

CNC

**8058/8060
8065**

Canned cycles (·M· model).

Ref: 2102



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 the product - CNC 8058 | 7 |
| About the product - CNC 8060 | 11 |
| About the product - CNC 8065 | 17 |
| Declaration of CE conformity and warranty conditions | 23 |
| Safety conditions | 25 |
| Returning conditions | 29 |
| CNC maintenance | 31 |
| New features..... | 33 |

CHAPTER 1 MILLING CANNED CYCLES.

| | | |
|-------|---|----|
| 1.1 | General concepts | 39 |
| 1.1.1 | Definition, Influence zone and cancellation of a canned cycle | 41 |
| 1.1.2 | Starting plane and reference plane | 42 |
| 1.1.3 | Programming the canned cycles | 43 |
| 1.1.4 | Programming a canned cycle in different planes | 44 |
| 1.1.5 | Modifying the parameters of a canned cycle | 46 |
| 1.2 | G81. Drilling canned cycle | 48 |
| 1.2.1 | Programming example | 49 |
| 1.3 | G82. Drilling canned cycle with variable peck | 50 |
| 1.3.1 | Programming example | 53 |
| 1.4 | G83. Deep-hole drilling canned cycle with constant peck | 54 |
| 1.4.1 | Programming example | 56 |
| 1.5 | G84. Tapping canned cycle | 57 |
| 1.5.1 | Programming example | 59 |
| 1.6 | G85. Reaming canned cycle | 60 |
| 1.6.1 | Programming example | 61 |
| 1.7 | G86. Boring canned cycle | 62 |
| 1.7.1 | Programming example | 64 |
| 1.8 | G87. Rectangular pocket canned cycle | 65 |
| 1.8.1 | Programming example | 68 |
| 1.9 | G88. Circular pocket canned cycle | 69 |
| 1.9.1 | Programming example | 72 |
| 1.10 | G210. Bore milling canned cycle | 73 |
| 1.11 | G211. Inside thread milling cycle | 75 |
| 1.12 | G212. Outside thread milling cycle | 78 |

CHAPTER 2 MULTIPLE MACHINING

| | | |
|-------|---|----|
| 2.1 | G160. Multiple machining in straight line | 82 |
| 2.1.1 | Programming example | 84 |
| 2.2 | G161. Multiple machining in rectangular pattern | 85 |
| 2.2.1 | Programming example | 87 |
| 2.3 | G162. Multiple machining in grid pattern | 88 |
| 2.3.1 | Programming example | 90 |
| 2.4 | G163. Multiple machining in a full circle | 91 |
| 2.4.1 | Programming example | 93 |
| 2.5 | G164. Multiple machining in arc pattern | 94 |
| 2.5.1 | Programming example | 96 |
| 2.6 | G165. Multiple machining in a chord pattern | 97 |
| 2.6.1 | Programming example | 98 |

CHAPTER 3 CYCLE EDITOR

| | | |
|-----|--|-----|
| 3.1 | Configuring the cycle editor. | 102 |
| 3.2 | Configuring el cycle editor (rotary axis cycles). | 103 |
| 3.3 | Teach-in mode. | 104 |
| 3.4 | Selecting data, profiles and icons | 105 |
| 3.5 | Associate a multiple machining operation with a canned cycle | 106 |
| 3.6 | Simulate a canned cycle | 107 |

CHAPTER 4 WORK PLANES AND SHIFTING OF THE MACHINING OPERATIONS.

| | | |
|-------|--|-----|
| 4.0.1 | Value applied when the value of a parameter is 0 | 111 |
|-------|--|-----|



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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

| | | |
|-------|---------------------------------------|-----|
| 5.1 | Center punching. | 114 |
| 5.1.1 | Basic operation. | 115 |
| 5.2 | Drilling. | 116 |
| 5.2.1 | Basic operation. | 117 |
| 5.3 | Deep hole drilling. | 118 |
| 5.3.1 | Basic operation. | 120 |
| 5.4 | Bore milling. | 121 |
| 5.4.1 | Basic operation. | 122 |
| 5.5 | Tapping. | 123 |
| 5.5.1 | Basic operation. | 125 |
| 5.6 | Thread milling. | 126 |
| 5.6.1 | Basic operation. | 128 |
| 5.7 | Reaming. | 129 |
| 5.7.1 | Basic operation. | 130 |
| 5.8 | Boring. | 131 |
| 5.8.1 | Basic operation. | 132 |
| 5.9 | Boring with spindle orientation. | 133 |
| 5.9.1 | Basic operation. | 135 |

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

| | | |
|-------|--|-----|
| 6.1 | Simple rectangular pocket. | 138 |
| 6.1.1 | Basic operation. | 141 |
| 6.2 | Rectangular pocket with rounding. | 142 |
| 6.2.1 | Basic operation. | 146 |
| 6.3 | Circular pocket. | 147 |
| 6.3.1 | Basic operation. | 151 |
| 6.4 | Pre-empted circular pocket. | 152 |
| 6.4.1 | Basic operation. | 155 |
| 6.5 | Rectangular boss. | 157 |
| 6.5.1 | Basic operation. | 161 |
| 6.6 | Circular boss. | 162 |
| 6.6.1 | Basic operation. | 165 |

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

| | | |
|-------|---|-----|
| 7.1 | 2D profile pocket. | 168 |
| 7.1.1 | Executable pocket file. | 173 |
| 7.1.2 | Basic operation. | 174 |
| 7.1.3 | Examples of how to define 2D profiles. | 176 |
| 7.2 | 3D profile pocket with islands. | 179 |
| 7.2.1 | Executable pocket file. | 185 |
| 7.2.2 | Basic operation. | 186 |
| 7.2.3 | Examples of how to define 3D profiles. | 187 |

CHAPTER 8**CANNED CYCLES OF THE EDITOR. ROUGHING.**

| | | |
|-------|--------------------------------------|-----|
| 8.1 | Surface milling. | 192 |
| 8.1.1 | Basic operation. | 195 |
| 8.2 | Point-to-point profile milling. | 196 |
| 8.2.1 | Basic operation. | 199 |
| 8.3 | Free profile milling. | 200 |
| 8.3.1 | Basic operation. | 203 |
| 8.4 | Slot milling. | 204 |
| 8.4.1 | Basic operation. | 208 |

CHAPTER 9**CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.**

| | | |
|-------|---|-----|
| 9.1 | Multiple machining in straight line. | 212 |
| 9.1.1 | Programming example. | 213 |
| 9.2 | Multiple machining in arc pattern. | 214 |
| 9.2.1 | Programming example. | 216 |
| 9.3 | Multiple machining in rectangular pattern. | 217 |
| 9.3.1 | Programming example. | 218 |
| 9.4 | Multiple machining in grid pattern. | 219 |
| 9.4.1 | Programming example. | 220 |
| 9.5 | Random multiple machining. | 221 |
| 9.5.1 | Programming example. | 221 |

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

| | | |
|------|--------------------------------|-----|
| 10.1 | Multiple center-punching. | 224 |
|------|--------------------------------|-----|



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

Canned cycles (·M· model).

| | | |
|--------|--------------------------|-----|
| 10.1.1 | Basic operation. | 227 |
| 10.2 | Multiple drilling. | 228 |
| 10.2.1 | Basic operation. | 232 |
| 10.3 | Multiple tapping. | 233 |
| 10.3.1 | Basic operation. | 236 |
| 10.4 | Multiple reaming. | 237 |
| 10.4.1 | Basic operation. | 240 |
| 10.5 | Multiple boring. | 241 |
| 10.5.1 | Basic operation. | 245 |
| 10.6 | Multiple bore milling. | 246 |
| 10.6.1 | Basic operation. | 249 |
| 10.7 | Multiple thread milling. | 250 |
| 10.7.1 | Basic operation. | 254 |
| 10.8 | Multiple slot milling. | 255 |
| 10.8.1 | Basic operation. | 259 |

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

| | | |
|---------|--|-----|
| 11.1 | ZC/YZ plane profiling cycle. | 262 |
| 11.1.1 | Basic operation. | 266 |
| 11.2 | Rectangular pocket cycle in the ZC/YZ plane. | 267 |
| 11.2.1 | Basic operation. | 272 |
| 11.3 | Circular pocket cycle in the ZC/YZ plane. | 273 |
| 11.3.1 | Basic operation. | 277 |
| 11.4 | Circular pre-empted pocket cycle in the ZC/YZ plane. | 278 |
| 11.4.1 | Basic operation. | 282 |
| 11.5 | 2D profile pocket cycle in the ZC/YZ plane. | 283 |
| 11.5.1 | Basic operation. | 288 |
| 11.6 | XC/XY plane profiling cycle. | 289 |
| 11.6.1 | Basic operation. | 293 |
| 11.7 | Rectangular pocket cycle in the XC/XY plane. | 294 |
| 11.7.1 | Basic operation. | 299 |
| 11.8 | Circular pocket cycle in the XC/XY plane. | 300 |
| 11.8.1 | Basic operation. | 304 |
| 11.9 | Circular pre-empted pocket in the XC/XY plane. | 305 |
| 11.9.1 | Basic operation. | 309 |
| 11.10 | 2D profile pocket cycle in the XC/XY plane. | 310 |
| 11.10.1 | Basic operation. | 315 |

CHAPTER 12 USER CYCLES.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

BLANK PAGE

ABOUT THE PRODUCT - CNC 8058

BASIC CHARACTERISTICS.

| Basic characteristics. | 8058 M | 8058 T |
|--------------------------------------|-------------------------|--------|
| Number of axes. | 3 to 4 | 3 to 4 |
| Number of spindles. | 1 | 1 to 2 |
| Maximum number of axes and spindles. | 5 | 6 |
| Interpolated axes. | 4 | 4 |
| Number of tool magazines. | 1 | 1 |
| Number of execution channels. | 1 | 1 |
| Number of handwheels. | 1 to 3 | |
| Type of servo system. | Analog / Sercos Digital | |
| Communications. | Ethernet | |
| Integrated PLC. | | |
| PLC execution time. | < 1ms/K | |
| Digital inputs / Digital outputs. | 1024 / 1024 | |
| Marks / Registers. | 8192 / 1024 | |
| Timers / Counters. | 512 / 256 | |
| Symbols. | Unlimited | |

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

(*) TTL / Differential TTL / Sinusoidal1 Vpp / SSI protocol / FeeDat / EnDat



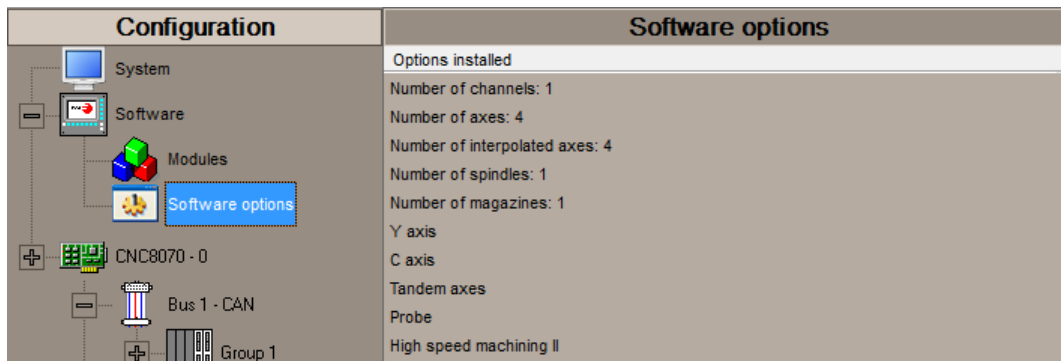
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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 8060 ADDIT AXES | Option to add axes to the default configuration. |
| SOFT 8060 ADDIT SPINDLES | Option to add spindles to the default configuration. |
| 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 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 TOOL RADIUS COMP | Option to enable radius compensation. This compensation programs the contour to be machined based on part dimensions without taking into account the dimensions of the tool that will be used later on. This avoids having to calculate and define the tool paths based on the tool radius. |
| SOFT PROFILE EDITOR | Option to enable the profile editor in edisimu mode and in the cycle editor. This editor can graphically, and in a guided way, define rectangular, circular profiles or any profile made up of straight and circular sections and it can also import dxf files. After defining the profile, the CNC generates the required blocks and add them to the program. |
| SOFT 60 HD GRAPHICS | 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). |

| Software option | Description. |
|--|---|
| SOFT 60 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 60 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 60 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. |
| 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 60 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 60 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 60 PPTRANS | Option to enable the program translator, which can convert programs written in other languages to Fagor ISO code. |
| 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). |

BLANK PAGE

ABOUT THE PRODUCT - CNC 8060

BASIC CHARACTERISTICS.

| Basic characteristics. | 8060 FL M | 8060 Power M | 8060 FL T | 8060 Power T | 8060 Power L | 8060 Power GL |
|--------------------------------------|-----------------------------------|--------------------|-----------------|--------------------|--------------------|---------------------|
| Number of axes. | 3 to 4 | 3 to 6 | 3 to 4 | 3 to 6 | 3 to 6 | 3 to 6 |
| Number of spindles. | 1 | 1 to 2 | 1 to 2 | 1 to 3 | 1 | 1 |
| Maximum number of axes and spindles. | 5 | 7 | 5 | 7 | 7 | 7 |
| Interpolated axes. | 4 | 4 | 4 | 4 | 4 | 4 |
| Number of tool magazines. | 1 | 1 | 1 | 1 to 2 | 1 | 1 |
| Number of execution channels. | 1 | 1 | 1 | 1 to 2 | 1 | 1 |
| Number of handwheels. | 1 to 3 | | | | | |
| Type of servo system. | Analog / Sercos Digital | | | | | |
| Communications. | RS485 / RS422 / RS232 Ethernet | | | | | |
| Integrated PLC. | | | | | | |
| PLC execution time. | < 1ms/K | | | | | |
| Digital inputs / Digital outputs. | 1024 / 1024 | | | | | |
| Marks / Registers. | 8192 / 1024 | | | | | |
| Timers / Counters. | 512 / 256 | | | | | |
| Symbols. | Unlimited | | | | | |
| Block processing time. | < 2.0 ms | < 1.5 ms | < 2.0 ms | < 1.5 ms | < 1 ms | < 1.5 ms |

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

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



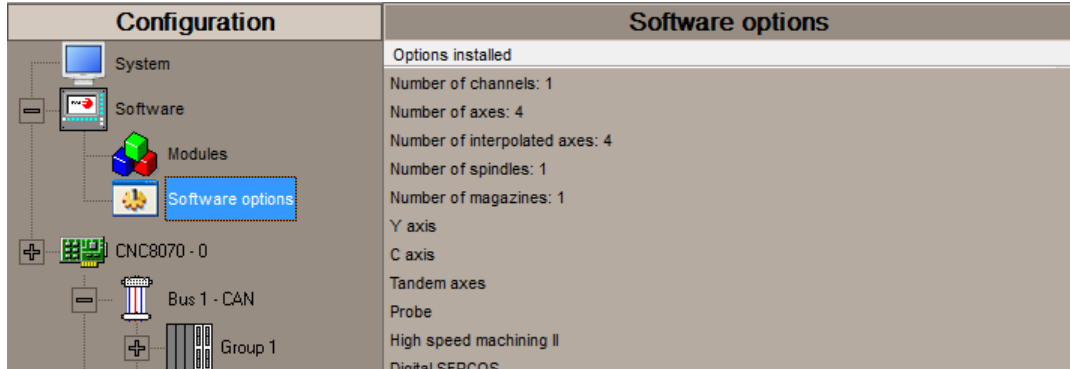
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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 8060 ADDIT AXES | Option to add axes to the default configuration. |
| SOFT 8060 ADDIT SPINDLES | Option to add spindles to the default configuration. |
| SOFT 8060 ADDIT TOOL MAGAZ | Option to add magazines to the default configuration. |
| SOFT 8060 ADDIT CHANNELS | Option to add channels to the default configuration. |
| SOFT DIGITAL SERCOS | Option for a Sercos digital bus. |
| SOFT THIRD PARTY I/Os | Option to enable non-Fagor remote modules. |
| SOFT MAB SYSTEM. | Sercos connection with MAB drives. |
| 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. |
| 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. |

| Software option | Description. |
|--|---|
| 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 as it can also import dxf files. After defining the profile, the CNC generates the required blocks and add them to the program. |
| SOFT 60 HD GRAPHICS | 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 60 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 60 RTCP | 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 60 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 60 Y AXIS | Option to enable lathe Y-axis kinematics and associated canned cycles. |
| SOFT 60 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. |
| SOFT 60 SYNCHRONISM | Option to enable the synchronization of paired axes and spindles, in speed or position, and through a given ratio. |
| 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. |



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

| Software option | Description. |
|---|--|
| SOFT 60 HSSA II MACHINING SYSTEM | Option to enable the HSSA-II (High Speed Surface Accuracy) algorithm for high-speed machining (HSC), which has the following advantages compared to the HSSA-I algorithm. <ul style="list-style-type: none"> • 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 60 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 60 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 60 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 60 PPTRANS | Option to enable the program translator, which can convert programs written in other languages to Fagor ISO code. |
| 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. |
| 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. |



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

| Software option | Description. |
|--------------------------------------|--|
| 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 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. |



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

BLANK PAGE

ABOUT THE PRODUCT - CNC 8065

BASIC CHARACTERISTICS.

| Basic characteristics. | 8065 M | | 8065 M Power | |
|--------------------------------------|--------|--------|--------------|---------|
| | Basic | Pack 1 | Basic | Pack 1 |
| Number of execution channels. | 1 | 1 | 1 | 1 to 4 |
| Number of axes. | 3 to 6 | 5 to 8 | 5 to 12 | 8 to 28 |
| Number of spindles. | 1 | 1 to 2 | 1 to 4 | 1 to 4 |
| Maximum number of axes and spindles. | 7 | 10 | 16 | 32 |
| Number of tool magazines. | 1 | 1 | 1 to 2 | 1 to 4 |
| Limited to 4 interpolated axes. | Option | Option | Option | Option |

| Basic characteristics. | 8065 T | | 8065 T Power | |
|--------------------------------------|--------|--------|--------------|---------|
| | Basic | Pack 1 | Basic | Pack 1 |
| Number of execution channels. | 1 | 1 to 2 | 1 to 2 | 1 to 4 |
| Number of axes. | 3 to 5 | 5 to 7 | 5 to 12 | 8 to 28 |
| Number of spindles. | 2 | 2 | 3 to 4 | 3 to 4 |
| Maximum number of axes and spindles. | 7 | 9 | 16 | 32 |
| Number of tool magazines. | 1 | 1 to 2 | 1 to 2 | 1 to 4 |
| Limited to 4 interpolated axes. | Option | Option | Option | Option |

| Basic characteristics. | 8065 M | 8065 M Power | 8065 T | 8065 T Power |
|-----------------------------------|--|--------------|--------|--------------|
| Number of handwheels. | 1 to 12 | | | |
| Type of servo system. | Analog / Digital Sercos / Digital Mechatrolink | | | |
| Communications. | RS485 / RS422 / RS232 Ethernet | | | |
| Integrated PLC. | | | | |
| PLC execution time. | < 1ms/K | | | |
| Digital inputs / Digital outputs. | 1024 / 1024 | | | |
| Marks / Registers. | 8192 / 1024 | | | |
| Timers / Counters. | 512 / 256 | | | |
| Symbols. | Unlimited | | | |
| Block processing time. | < 1 ms | | | |

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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

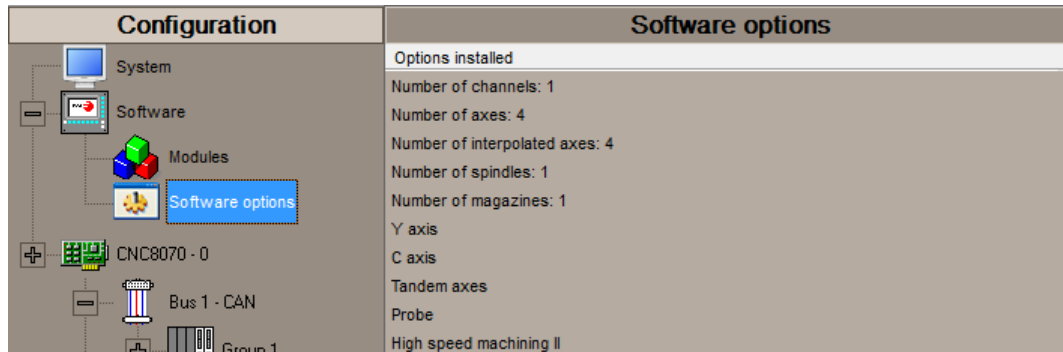
(*) TTL / Differential TTL / Sinusoidal 1 Vpp / SSI protocol / FeeDat / EnDat

Customizing (on an open system only).

PC-based open system, fully customizable.
 INI configuration files.
 Tool for display configuration FGUIM.
 Visual Basic®, Visual C++®, etc.
 Internal databases in Microsoft® Access.
 OPC compatible interface

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 I/Os | Option to enable non-Fagor remote 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. |
| 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. |



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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

| Software option | Description. |
|--|---|
| SOFT HSSA II MACHINING SYSTEM | <p>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.</p> <ul style="list-style-type: none"> • 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 FVC STANDARD SOFT FVC UP TO 10m3 SOFT FVC MORE TO 10m3 | <p>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.</p> <p>There are 3 options, which depend on the size of the machine.</p> <ul style="list-style-type: none"> • 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. |
| SOFT TANGENTIAL CONTROL | <p>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.</p> |
| SOFT PROBE | <p>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.</p> |
| SOFT CONV USER CYCLES | <p>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.</p> |
| SOFT PROGTL3 | <p>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.</p> |
| SOFT PPTRANS | <p>Option to enable the program translator, which can convert programs written in other languages to Fagor ISO code.</p> |

| Software option | Description. |
|---|---|
| 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 IEC 61131 LANGUAGE This feature requires the MP-PLUS (83700201) processor. | IEC 61131 is a PLC programming language that is very popular in alternative markets, which is slowly entering into the machine-tool market. With this feature, the PLC may be programmed either in the usual Fagor language or in IEC 61131 format. |



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

DECLARATION OF CE CONFORMITY AND WARRANTY CONDITIONS

DECLARATION OF CONFORMITY

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

WARRANTY TERMS

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



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

BLANK PAGE

SAFETY CONDITIONS

Read the following safety measures in order to prevent harming people or damage to this product and those products connected to it. Fagor Automation shall not be held responsible of any physical 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.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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: <ul style="list-style-type: none"> 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.

Canned cycles (·M· model).

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

CNC 8058
CNC 8060
CNC 8065

REF: 2102

BLANK PAGE

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 AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

BLANK PAGE

CNC MAINTENANCE

CLEANING

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

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

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

PRECAUTIONS BEFORE CLEANING THE UNIT

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

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

BLANK PAGE

NEW FEATURES.

Manual reference: Ref: 2102
Date of publication: February, 2021
Associated software: v6.21.12

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

| List of features. | Manual |
|--|----------|
| Unified version for milling machine, lathe and laser. | |
| New integrated CNC 8058 model, with 10K monitor. | [HARD] |
| SO W10. New Windows 10 operating system. The CNC must meet the following requirements. <ul style="list-style-type: none"> • 16 GB NandFlash. • 4 GB FRAM memory. | |
| OPC UA protocol. | |
| “Lantek Expert Inside ” application. | |
| New RIOS-E Inline modules. <ul style="list-style-type: none"> • IB IL 24 DI 32/HD-PAC • IB IL 24 DO 32/HD-PAC | [RIOS-E] |
| This version must be installed in administrator mode, so that the process properly registers the new FCombo.ocx control. | [INST] |
| Incompatibility in gap control. With version v6.20, the CNC always considers the approach coordinate in absolute coordinates, regardless of the active G90/G91 function. To program the approach coordinate in incremental coordinates, add the “INC” command to the #GAPCTRL instruction. | [INST] |
| Incompatibility in the leapfrog. With version v6.20, the CNC always considers the coordinate programmed in the “POS” command in absolute coordinates, regardless of the G90/G91 command. To program the coordinate at the end of the leap in incremental coordinates, use the “INCPOS” command. | [INST] |
| New language for the HMI. Turkish. <ul style="list-style-type: none"> • Machine parameters: LANGUAGE | [INST] |
| The HEADREF parameter values have changed to “Tool base” (previously “Yes”), “Head base” (previously “No”) and there is a new “Auto tool base” value. 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), in case any TDATA has changed manually. | [INST] |
| <div style="display: flex; justify-content: space-around;"> <div style="text-align: center;"> <p>HEADREF=Head base</p> </div> <div style="text-align: center;"> <p>HEADREF=Tool base</p> </div> </div> | |

- [CYC-M]..... Machining canned cycles (·M· model).
- [CYC-T]..... Machining canned cycles (·T· model).
- [HARD]..... Hardware configuration.
- [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.
- [RIOS-E]..... EtherCAT remote modules (RIOW-E Inline)
- [VAR]..... CNC variables.



FAGOR AUTOMATION

CNC 8058
 CNC 8060
 CNC 8065

REF: 2102

| List of features. | Manual |
|---|--------------|
| 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] |
| New TMOOPERATION=12 operation for asynchronous magazine. Leave the spindle tool in the magazine and take a tool from the same magazine (as TMOOPERATION=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. | [INST] |
| For a multi-axis where the same motor controls several axes, the CNC allows dynamic management of the multi-axis. This type of management consists of activating the axes of the group as they are programmed. To carry out the dynamic management of a multi-axis group, the axes of the group should be configured as dead axes and the DWELL>0 parameter defined. The axes use the DWELL parameter as a waiting time before giving an error for not having the SWITCH mark active. The PLC monitors that the ENABLE mark is active in order to engage that axis and activate its mark SWITCH, SERVOON, SPENA and DRENA mark. | [INST] |
| Gap control at the drive. • Parameters: GAPANAINTYPE GAPANAINID | [INST] |
| Connect the gap sensor via the EtherCAT. • Parameters: GAPANAINTYPE GAPANAINID | [INST] |
| 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. | [INST] |
| 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] |
| Changing the FFGAIN for the drive (PP216) in synchronous mode. | [INST] |
| DMC. Power peak detection. • Machine parameters: DMCPEAKSIZE • Variable: (V.)[ch].MPG.DMCPEAKSIZE | [INST] [VAR] |
| DMC. Detection of excessive power consumption. • Machine parameters: DMCPWRFACTOR • Variable: (V.)[ch].MPG.DMCPWRFACTOR | [INST] [VAR] |
| 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] |
| New API functions for common technology tables. | [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] |
| Translator 8055. Translation of the G72+G16+G15 sequence. | [OPT] |
| "Intel Graphics Control Panel" application. Configuring the brightness and contrast of the monitors. | [OPT] |
| Utilities mode. Encryption of scripts (scp files) and subroutines associated with user cycles. | [OPT] |
| 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. | [OPT] |
| Kinematics calibration. The cycle only allows part of the travel of the rotary axes module to be calibrated. The part of the non-calibrated path is not compensated. | [OPT] |
| Lathe HD graphics take into account offsets in the X axis of part zero. | |

- [CYC-M].....Machining canned cycles (.M· model).
- [CYC-T].....Machining canned cycles (.T· model).
- [HARD].....Hardware configuration.
- [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.
- [RIOS-E].....EtherCAT remote modules (RIOW-E Inline)
- [VAR].....CNC variables.

REF: 2102



CNC 8058
CNC 8060
CNC 8065

| List of features. | Manual | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
|---|----------------------------|------------|---|------------|---|--|--|--------|--|----------|---|------------|---|------------|---|-----------|--|------------|--|------------|---|------------|------------|------------|------------|-----|------------|------------|------------|------------|------|------------|------------|------------|------------|------|------------|------------|------------|------------|-------|------------|------------|------------|--|------|------------|------------|------------|--|------|------------|------------|------------|------------|-------|------------|------------|------------|------------|-------|
| <p>Active offsets table. The user tables offer a new table for showing the active values in different functions; G92, G159, G201, etc.</p> <table border="1"> <thead> <tr> <th colspan="5">Channel 1 : Active Offsets</th> </tr> <tr> <th></th> <th>X (mm)</th> <th>Y (mm)</th> <th>Z (mm)</th> <th>B (deg.)</th> </tr> </thead> <tbody> <tr><td>G92</td><td>00000.0000</td><td>00000.0000</td><td>00000.0000</td><td>00000.0000</td></tr> <tr><td>G54, G159</td><td>00000.0000</td><td>00000.0000</td><td>00000.0000</td><td>00000.0000</td></tr> <tr><td>G158</td><td>00000.0000</td><td>00000.0000</td><td>00000.0000</td><td>00000.0000</td></tr> <tr><td>FIX</td><td>00000.0000</td><td>00000.0000</td><td>00000.0000</td><td>00000.0000</td></tr> <tr><td>G191</td><td>00000.0000</td><td>00000.0000</td><td>00000.0000</td><td>00000.0000</td></tr> <tr><td>G201</td><td>00000.0000</td><td>00000.0000</td><td>00000.0000</td><td>00000.0000</td></tr> <tr><td>ACSCS</td><td>00000.0000</td><td>00000.0000</td><td>00000.0000</td><td></td></tr> <tr><td>RTCP</td><td>00000.0000</td><td>00000.0000</td><td>00000.0000</td><td></td></tr> <tr><td>HEAD</td><td>00000.0000</td><td>00000.0000</td><td>00000.0000</td><td>00000.0000</td></tr> <tr><td>PLCOF</td><td>00000.0000</td><td>00000.0000</td><td>00000.0000</td><td>00000.0000</td></tr> </tbody> </table> | Channel 1 : Active Offsets | | | | | | X (mm) | Y (mm) | Z (mm) | B (deg.) | G92 | 00000.0000 | 00000.0000 | 00000.0000 | 00000.0000 | G54, G159 | 00000.0000 | 00000.0000 | 00000.0000 | 00000.0000 | G158 | 00000.0000 | 00000.0000 | 00000.0000 | 00000.0000 | FIX | 00000.0000 | 00000.0000 | 00000.0000 | 00000.0000 | G191 | 00000.0000 | 00000.0000 | 00000.0000 | 00000.0000 | G201 | 00000.0000 | 00000.0000 | 00000.0000 | 00000.0000 | ACSCS | 00000.0000 | 00000.0000 | 00000.0000 | | RTCP | 00000.0000 | 00000.0000 | 00000.0000 | | HEAD | 00000.0000 | 00000.0000 | 00000.0000 | 00000.0000 | PLCOF | 00000.0000 | 00000.0000 | 00000.0000 | 00000.0000 | [OPT] |
| Channel 1 : Active Offsets | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | X (mm) | Y (mm) | Z (mm) | B (deg.) | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| G92 | 00000.0000 | 00000.0000 | 00000.0000 | 00000.0000 | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| G54, G159 | 00000.0000 | 00000.0000 | 00000.0000 | 00000.0000 | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| G158 | 00000.0000 | 00000.0000 | 00000.0000 | 00000.0000 | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| FIX | 00000.0000 | 00000.0000 | 00000.0000 | 00000.0000 | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| G191 | 00000.0000 | 00000.0000 | 00000.0000 | 00000.0000 | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| G201 | 00000.0000 | 00000.0000 | 00000.0000 | 00000.0000 | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| ACSCS | 00000.0000 | 00000.0000 | 00000.0000 | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| RTCP | 00000.0000 | 00000.0000 | 00000.0000 | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| HEAD | 00000.0000 | 00000.0000 | 00000.0000 | 00000.0000 | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| PLCOF | 00000.0000 | 00000.0000 | 00000.0000 | 00000.0000 | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| <p>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.</p> | [OPT] | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| <p>Logging data and events in a user log.</p> <ul style="list-style-type: none"> Variable: (V.)E.USERLOG <table border="1"> <thead> <tr> <th colspan="2">USER LOG</th> </tr> </thead> <tbody> <tr><td>May 19 13:20:53, Pulsada tecla del panel de jog , KEYBOARD1 OVR = 100</td><td></td></tr> <tr><td>May 19 13:21:31, Reset del CNC, 0.00000</td><td></td></tr> <tr><td>May 19 13:21:40, Cierre del CNC, 0.00000</td><td></td></tr> <tr><td>May 19 13:22:24, Arranque del CNC, 0.00000</td><td></td></tr> <tr><td>May 19 13:22:29, Activación de fichero de material, 0.00000, 001_MS1_50</td><td></td></tr> <tr><td>May 19 13:22:29, Pulsada tecla del panel de jog , KEYBOARD1 OVR = 100</td><td></td></tr> <tr><td>May 19 13:23:07, Reset del CNC, 0.00000</td><td></td></tr> <tr><td>May 19 13:23:17, Cierre del CNC, 0.00000</td><td></td></tr> <tr><td>May 19 13:24:02, Arranque del CNC, 0.00000</td><td></td></tr> <tr><td>May 19 13:24:06, Activación de fichero de material, 0.00000, 001_MS1_50</td><td></td></tr> </tbody> </table> | USER LOG | | May 19 13:20:53, Pulsada tecla del panel de jog , KEYBOARD1 OVR = 100 | | May 19 13:21:31, Reset del CNC, 0.00000 | | May 19 13:21:40, Cierre del CNC, 0.00000 | | May 19 13:22:24, Arranque del CNC, 0.00000 | | May 19 13:22:29, Activación de fichero de material, 0.00000, 001_MS1_50 | | May 19 13:22:29, Pulsada tecla del panel de jog , KEYBOARD1 OVR = 100 | | May 19 13:23:07, Reset del CNC, 0.00000 | | May 19 13:23:17, Cierre del CNC, 0.00000 | | May 19 13:24:02, Arranque del CNC, 0.00000 | | May 19 13:24:06, Activación de fichero de material, 0.00000, 001_MS1_50 | | [OPT] | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| USER LOG | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| May 19 13:20:53, Pulsada tecla del panel de jog , KEYBOARD1 OVR = 100 | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| May 19 13:21:31, Reset del CNC, 0.00000 | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| May 19 13:21:40, Cierre del CNC, 0.00000 | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| May 19 13:22:24, Arranque del CNC, 0.00000 | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| May 19 13:22:29, Activación de fichero de material, 0.00000, 001_MS1_50 | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| May 19 13:22:29, Pulsada tecla del panel de jog , KEYBOARD1 OVR = 100 | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| May 19 13:23:07, Reset del CNC, 0.00000 | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| May 19 13:23:17, Cierre del CNC, 0.00000 | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| May 19 13:24:02, Arranque del CNC, 0.00000 | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| May 19 13:24:06, Activación de fichero de material, 0.00000, 001_MS1_50 | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| <p>In programs that include #SET AX (change configuration of axes) and #DEFGRAP (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.</p> | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| <p>If the CNC reaches the end of a program without an M02/M30, then it ends the execution without prompting a warning.</p> | [PRG] | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| <p>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.</p> | [INST] [PRG] | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| <p>Table+spindle kinematics (type 52). From this version onwards, the CNC does not take into account the axes of the table that are not programmed.</p> <ul style="list-style-type: none"> Statement: #SELECT ORI. | [PRG] | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| <p>Table+spindle kinematics (type 52). Take into account the position of the table when orienting the tool. ALL command.</p> <ul style="list-style-type: none"> Statement: #SELECT ORI. | [PRG] | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| <p>Table+spindle kinematics (type 52). When activating the RTCP, define the part of the kinematics (table or spindle) to be used and the type of RTCP (static or dynamic). HEAD, TABLE commands</p> <ul style="list-style-type: none"> Statement: #RTCP. | [PRG] | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| <p>Table+spindle kinematics (type 52). When activating the RTCP, define the part of the kinematics (table or spindle) to be used and the type of RTCP (static or dynamic). HEAD, TABLE commands.</p> <ul style="list-style-type: none"> Statement: #RTCP. | [PRG] | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| <p>Table+spindle kinematics (type 52). Activation of the RTCP according to the values defined in the machine parameters (TDATA). CLEAR command.</p> <ul style="list-style-type: none"> Statement: #RTCP. | [PRG] | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| <p>Table+spindle kinematics (type 52). Rotate the part coordinate system when rotating the table. COROT command.</p> <ul style="list-style-type: none"> Statement: #RTCP. | [PRG] | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | | |

- [CYC-M]..... Machining canned cycles (·M· model).
- [CYC-T]..... Machining canned cycles (·T· model).
- [HARD]..... Hardware configuration.
- [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.
- [RIOS-E]..... EtherCAT remote modules (RIOW-E Inline)
- [VAR]..... CNC variables.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

| List of features. | Manual | |
|--|----------|----------|
| Management of discontinuities in the orientation of rotary axes. Orient the tool in the programmed direction of the main or secondary rotary axis. DIRF, DIRS commands. • Statement: #DEFROT. | [PRG] | |
| OEM subroutines for kinematics calibration can execute motions in two axes at the same time. | | |
| New mode for defining a tilted plane (MODE 7). Mode for defining an inclined plane rotating first around 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. | [PRG] | |
| Activate a kinematic displaying the tool tip dimensions. TIP command. • Statement: #KIN ID. • Variable: (V.)G.KINIDMODE | [PRG] | [VAR] |
| Defining the path and the name of a program in a macro. • Statement: #DEF. | [PRG] | |
| Does not prompt an error for operations using #OPEN, #WRITE y #CLOSE. New command MUTED. • Statement: #OPEN. | [PRG] | |
| New access mode R(READ). Verify if the file exists. • Statement: #OPEN. | [PRG] | |
| Change the HMI variables from the workpiece programme. • Statement: #MSGVAR. | [PRG] | |
| DMC. If a minimum and maximum override with value 100 is programmed, the DMC monitors the power but without changing the override to keep the power constant. The actions to take when increasing the power will be defined by the OEM, for example, from the PLC. • Statement: #DMC. | [PRG] | |
| Identifying the installation folder. • Variable: (V.)G.FOLDERID. | [VAR] | |
| Identification of the language selected in the CNC. • Variable: (V.)G.LANGUAGEID. | [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] | |
| Working with RTCP in spindle+table kinematics. • Variable: (V.)G.RTCPMODE | [VAR] | |
| Softkey to hide the Fagor cycles from the editor and conversational mode. | [CYC-M] | [CYC-T] |
| | [OPT-MC] | [OPT-TC] |
| The canned cycles may be defined by the CZ plane. | [CYC-M] | [CYC-T] |
| | [OPT-MC] | [OPT-TC] |
| The #CYL instruction admits programming of the CZ plane (#CYL[C,Z]). | | |
| The profile editor shows the CZ plane oriented correctly. | | |
| The #PLC instruction allows symbols defined by the PLC program (PDEF command). • Instruction #PLC. | [LSR] | |
| Gap control. The CNC always considers the approach coordinate in absolute coordinates, regardless of the active G90/G91 function. To program the approach coordinate in incremental coordinates, add the "INC" command to the #GAPCTRL instruction. • Statement: #GAPCTRL | [LSR] | |
| Leapfrog. The CNC always considers the coordinate programmed in the "POS" command in absolute coordinates, regardless of the G90/G91 command. To program the coordinate at the end of the leap in incremental coordinates, use the "INCPOS" command. • Instruction #LEAP. | [LSR] | |
| Power control through the PWM frequency (FREQ command). • Instruction #PWRCTRL. • Variables: (V.)G.PWRFREQON (V.)G.PWRFREQOVRMIN (V.)G.PWRFREQOVRMAX (V.)G.PWRFREQFMIN (V.)G.PWRFREQFMAX | [LSR] | |
| Power control set by the OEM (FMASTER command). When the laser cut has a configuration where the feedrate between the nozzle and the sheet is not dependent on the XY axes (for example, tube cutting where only one axis is rotated), the feedrate can be defined from the PLC by writing in the G.FEEDPWRCTRL variable and activating the power control through the FMASTER command. • Instruction #PWRCTRL. • Variable: (V.)G.FEEDPWRCTRL | [LSR] | |

- [CYC-M]..... Machining canned cycles (.M· model).
- [CYC-T]..... Machining canned cycles (.T· model).
- [HARD]..... Hardware configuration.
- [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.
- [RIOS-E]..... EtherCAT remote modules (RIOW-E Inline)
- [VAR]..... CNC variables.



CNC 8058
CNC 8060
CNC 8065

REF: 2102

Canned cycles (·M· model).

| List of features. | Manual | |
|--|--------|-------|
| Generic technology tables in the laser model. <ul style="list-style-type: none"> Instruction #TECHTABLE. Variables: (V.)TT.tableName_varName (V.)[ch].G.ACTIVEMATERIAL (V.)[ch].G.TECHTABLE Subroutines: nombreTablaON.fst nombreTablaOFF.fst | [LSR] | |
| Technology tables. Adding a parameter after the selected position. | [LSR] | |
| Improved refresh speed of the CUT_VIEW. | [LSR] | |
| CUT_VIEW mode. Improving the graphic display with many piercing points. | [LSR] | |
| CUT_VIEW mode. New method for selecting piercing points; "Restart PP". | [LSR] | |
| CUT_VIEW mode. Select the current point as the piercing point. | [LSR] | |
| Change the orientation of the XY axes of the CUT_VIEW. | [LSR] | |
| Refresh the CUT_VIEW. <ul style="list-style-type: none"> CVIEWREFRESH mark. | [INST] | [LSR] |
| Executing within a piercing point selection. | [LSR] | |
| Keep in mind that the angle of the sheet when resuming the cut using "Break PP" or "Restart PP". | [LSR] | |
| The CNC can limit the preview of the program to a specific area by using the instructions #PREVIEW BEGIN and #PREVIEW END. The #PREVIEW BEGIN instruction defines the start of the cut preview and the #PREVIEW END instruction the end of the cut preview, both for the program selector and in the CUT_VIEW page in automatic mode. <ul style="list-style-type: none"> Instruction #PREVIEW BEGIN. | [LSR] | |
| Automatic mode. "Verify nozzle" softkey. | [LSR] | |
| Position and dimensions of the minimum rectangle containing the drawing. <ul style="list-style-type: none"> Variables: (V.)G.FRAMEOFF1 (V.)G.FRAMEOFF2 (V.)G.INCMAX1 (V.)G.INCMAX2 | [LSR] | |
| G137 as the new default value for the transition between blocks (parameter IRCOMP). For laser models, the default value is G137; for other models it is G136. | [LSR] | |
| New algorithm for G137 machining. | [LSR] | |
| Defining a profile (#PIERCING and #CUTTING). Before now, the beginning of a laser profile started with transition G0 to G1 and ended with transition G1 to G0. And from now on, the instructions #PIERCING and #CUTTING ON indicate the beginning of a profile (it is only possible to program one of these) and #CUTTING OFF is to end the profile. | [LSR] | |
| Cutting of several sheets in a single program. To cut several sheets on the same table, each with a different angle, different programs can be executed for each sheet using the #EXEC instruction. If the execution is interrupted and resumed using a "Break PP", then the CNC will recall the rotation of the corresponding program. | [LSR] | |
| Cutting sheets longer than the table feed. | [LSR] | |

[CYC-M]..... Machining canned cycles (·M· model).
 [CYC-T]..... Machining canned cycles (·T· model).
 [HARD]..... Hardware configuration.
 [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.
 [RIOS-E]..... EtherCAT remote modules (RIOW-E Inline)
 [VAR]..... CNC variables.



FAGOR AUTOMATION

CNC 8058
 CNC 8060
 CNC 8065

REF: 2102

BLANK PAGE

MILLING CANNED CYCLES.

1

1.1 General concepts

Some canned cycles are edited in ISO code (described in this chapter) and others are generated in conversational mode. The canned cycles edited in ISO code are defined using a "G" function and its relevant parameters.

| | |
|------|---|
| G81 | Drilling canned cycle. |
| G82 | Drilling canned cycle with variable peck (drilling step). |
| G83 | Deep hole drilling canned cycle with constant peck (drilling step). |
| G84 | Tapping canned cycle. |
| G85 | Reaming canned cycle. |
| G86 | Boring canned cycle. |
| G87 | Rectangular pocket canned cycle. |
| G88 | Circular pocket canned cycle. |
| G210 | Bore milling canned cycle |
| G211 | Inside thread milling canned cycle. |
| G212 | Outside thread milling canned cycle. |

Other functions related to canned cycles:

| | |
|-----|---|
| G80 | Canned cycle cancellation. |
| G98 | The tool, after the canned cycle is done, returns to the starting plane. |
| G99 | The tool, after the canned cycle is done, returns to the reference plane. |

Machining cycles may be executed in any plane. The penetration is carried out along the longitudinal axis, selected with function G20 or with the instruction "#TOOL AX", or when missing, along the axis perpendicular to the active plane.

Combined (dual-purpose) machines Milling and turning canned cycles available at the same CNC.

On dual-purpose machines, those where milling and turning operations may be carried out, the CNC offers the possibility to run canned cycles of both machines. Since both types of canned cycles share the same 'G' functions, the user can select which cycles to execute. By default, it executes the cycles of the software installed.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

On a mill model CNC (milling software installed).

By default, it will execute the milling canned cycles. To execute the turning canned cycles, use the following instructions:

- #LATHECY ON - To activate the turning canned cycles.
- #LATHECY OFF - To deactivate the turning canned cycles.

| | |
|--------------|--|
| G81 ... | Drilling canned cycle. |
| #LATHECY ON | Activates the turning canned cycles. |
| G81 ... | |
| G87 ... | |
| #LATHECY OFF | Deactivates the turning canned cycles. |

On a lathe model CNC (lathe software installed).

By default, it will execute the turning canned cycles. To execute the milling canned cycles, use the following instructions:

- #MILLCY ON - To activate the milling canned cycles.
- #MILLCY OFF - To deactivate the milling canned cycles.

| | |
|-------------|--|
| G81 ... | Turning canned cycle with straight sections. |
| #MILLCY ON | Activates the milling canned cycles. |
| G81 ... | |
| G86 ... | |
| #MILLCY OFF | Deactivates the milling canned cycles. |

1.

MILLING CANNED CYCLES.
General concepts



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

1.1.1 Definition, Influence zone and cancellation of a canned cycle.

Definition of a canned cycle.

The canned cycles are defined using the relevant "G" function and its associated parameters. The canned cycle may be defined anywhere in the program, in the main program as well as in a subroutine.

Executing a canned cycle does not change the history of the previous "G" functions and maintains the spindle turning direction. If it is stopped, it starts clockwise (M03).

Influence zone of a canned cycle

The canned cycle is modal. Once a canned cycle has been defined, by program or MDI/MDA, it stays active until its cancellation is programmed or until one of the conditions that cancels it occurs.

While the canned cycle is active, all the blocks that are programmed are under the influence of that canned cycle. If inside the influence zone of the canned cycle, a motion block is executed, the CNC makes the programmed move and then executes the machining operation corresponding to the canned cycle. If there is a motionless block inside the influence zone of a canned cycle, the CNC does not repeat the active canned cycle.

If inside the influence zone of a canned cycle, a motion block is programmed containing a number of repetitions (NR command), the CNC executes the programmed movement and the canned cycle the programmed number of times. If the number of repetitions is zero, NR0, the CNC will only execute the programmed movement.

```
G99 G81 Z2 I-20
    Definition and execution of the drilling canned cycle.
G90 G01 X85
    Move to point X85 and drill a new hole.
G91 Y85 NR3
    The CNC repeats the movement and the drilling three times.
G90 G01 X0 NR0
    Move to point X0 without drilling.
```

Canned cycle cancellation

A cycle is cancelled as follows.

- Function G80, which can be programmed in any block.
- After defining a new canned cycle. The new canned cycle cancels and replaces any other cycle that may be active.
- After executing M02, M30 or after an emergency or reset.
- When doing a home search using function G74.
- Selecting another longitudinal axis, with G20 or with #TOOL AX.
- Selecting a new work plane.

1.

MILLING CANNED CYCLES.
General concepts

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

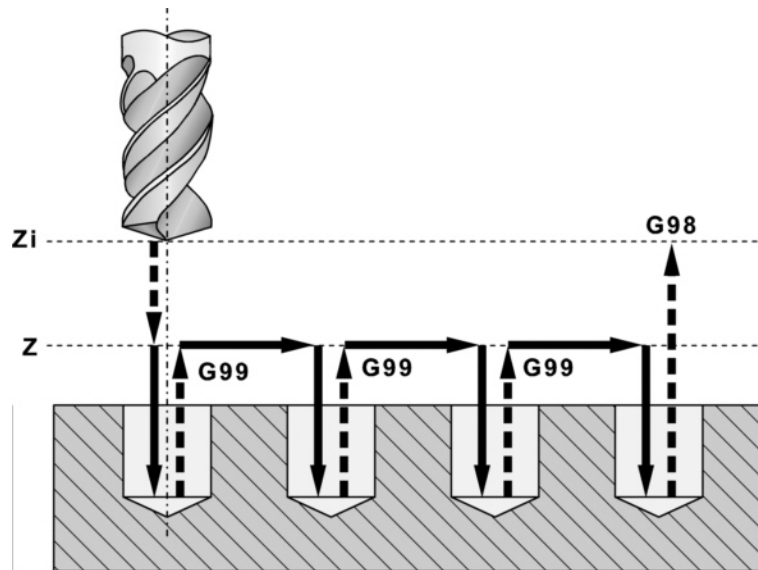
1.1.2 Starting plane and reference plane.

There are two coordinates in the machining cycles along the longitudinal axis that are described next because they are important:

- Starting plane (Zi). Tool coordinate (position) when defining the cycle.
- Reference plane (Z). This plane is programmed in the cycle and represents a part approaching coordinate. This plane may be programmed in absolute or incremental coordinates, in which case, it will be referred to the reference plane.

Functions G98 and G99 indicate where the tool returns after machining. Both functions are modal and G98 is assumed by default. These function may be used in the canned cycle calling block and in the blocks that are under its influence.

- G98 Return (withdraw) to the starting plane (Zi).
G99 Return (withdraw) to the reference plane (Z).



| | |
|--------------|--|
| G99 G1 X0 Y0 | (Movement) |
| G81 Z I K | (Defines and executes the drilling canned cycle) |
| X1 Y1 | (Move and drill) |
| X2 Y2 | (Move and drill) |
| G98 X3 Y3 | (Move and drill) |
| G80 | (Canned cycle cancellation) |

1.1.3 Programming the canned cycles.

As a general rule, the structure of a canned cycle defining block is the following.

```
[G functions] G8x [Machining point] cycle parameters [F S T D M]
```

First program the G functions, where the G function calling the cycle must be the last one. Then, the machining point may be programmed (except the longitudinal axis), both in Polar coordinates and Cartesian coordinates. After defining the point, define the parameters corresponding to the canned cycle and finally the desired complementary functions F S T D M.

```
N10 G99 G1 G81 X60 Y0 Z2 I-20 F1000 S2000 M4
```

It is also possible to add the definition of the canned cycle (calling function and parameters) at the end of any block.

```
N10 G99 G1 X60 Y0 F1000 S2000 M4 G81 Z2 I-20
```

Define a canned cycle in the influence zone of another canned cycle.

Here are two examples for defining a canned cycle in the influence zone of another active cycle.

1st example. Block N20 cancels the active canned cycle and block N40 activates the second canned cycle. If block N20 is not programmed, block N30 repeats the canned cycle defined in block N10.

```
G00 G90 Z25
  (Starting plane; Z25).
N10 G81 Z2 I-20
  (Canned cycle definition).
N15 X160 Y50 F3000
  (Movement to point X60 Y0 and drilling).
  (Withdraw to the reference plane; Z2).
N20 G80
  (Canned cycle cancellation).
N30 G1 X200 Y200
  (Movement to X200 Y200).
N40 G83 Z2 I-2 J5
  (Canned cycle definition).
N50 X220
  (Drilling).
  (Withdraw to the reference plane; Z2).
N60 M30
```

2nd example. The active canned cycle defined in N10 is canceled when defining a new one in N30. When executing block N30, it first moves the axes to X200 Y200 and then it executes the canned cycle G83.

```
G00 G90 Z25
  (Starting plane; Z25).
N10 G81 Z2 I-20
  (Canned cycle definition).
N15 X160 Y50 F3000
  (Movement to point X60 Y0 and drilling).
  (Withdraw to the reference plane; Z2).
N30 G1 X200 Y200 G83 Z2 I-2 J5
  (Movement to X200 Y200).
  (Drilling).
  (Withdraw to the reference plane; Z2).
N50 X220
  (Movement to point X220 and drilling).
  (Withdraw to the reference plane; Z2).
N60 M30
```

1.

MILLING CANNED CYCLES.
General concepts

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

1.1.4 Programming a canned cycle in different planes.

The programming format is always the same, it does not depend on the work plane. The following examples show how to drill in both axes of the XY plane in both directions; X axis as abscissa axis and Y axis as ordinate axis

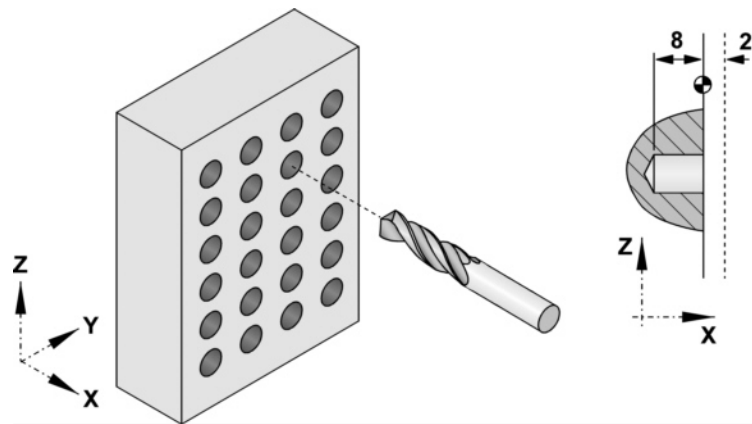
Function G81 defines the drilling canned cycle. The call parameters have the following meaning.

| | |
|-------|---|
| X/Y/Z | Reference coordinate along the longitudinal axis. |
| I | Drilling depth. |
| K | Dwell at the bottom. |

For each type of machine and machining operation the tool's longitudinal axis must be selected using the #TOOL AX instruction so the CNC knows the machining direction.

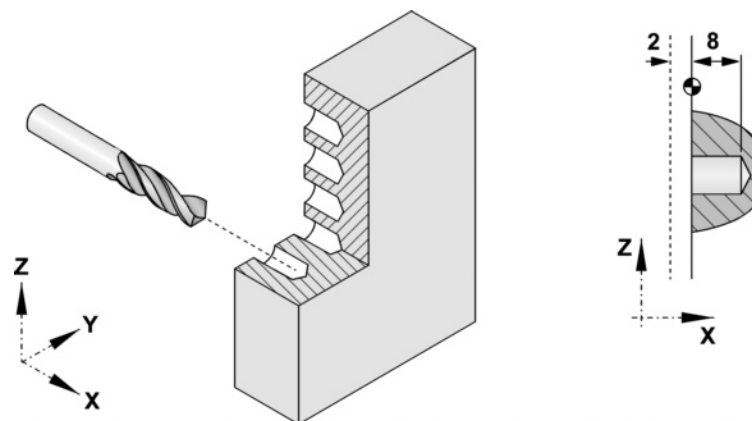
In the following examples, the part surface has a 0 coordinate, the holes are 8 mm deep and the reference coordinate is 2 mm above the surface.

Example 1:



```
G19
#TOOL AX [X+]
G1 X25 F1000 S1000 M3
G81 X2 I-8 K1
```

Example 2:



```
G19
#TOOL AX [X-]
G1 X-25 F1000 S1000 M3
G81 X-2 I8 K1
```

1.

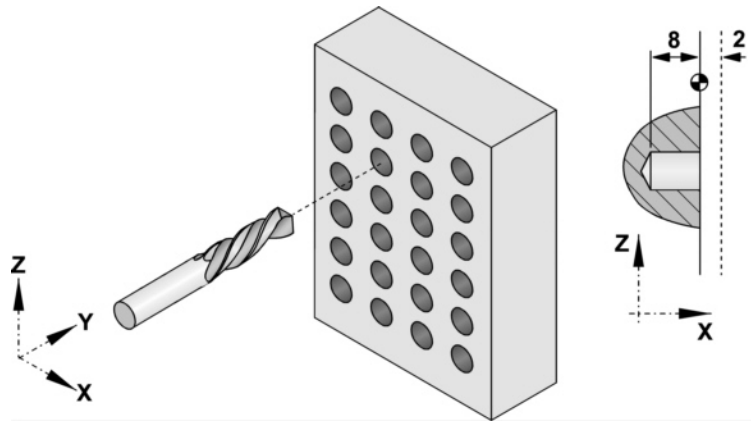
MILLING CANNED CYCLES.
General concepts

FAGOR 
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

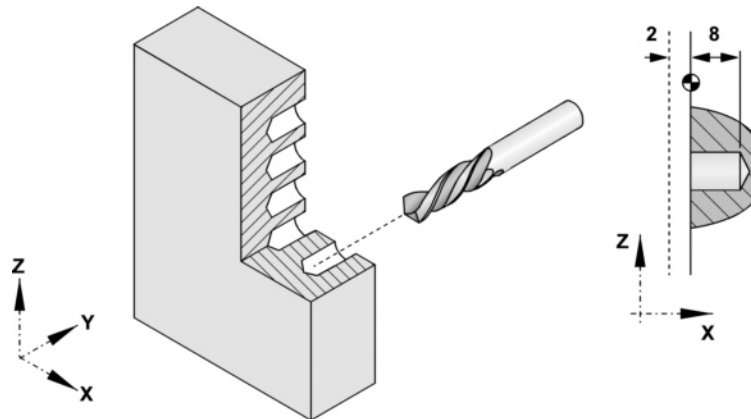
REF: 2102

Example 3:



```
G18
#TOOL AX [Y-]
G1 Y25 F1000 S1000 M3
G81 Y2 I-8 K1
```

Example 4:



```
G18
#TOOL AX [Y+]
G1 Y-25 F1000 S1000 M3
G81 Y-2 I8 K1
```

If working in the U V plane and the tool is located on the longitudinal axis X2, it is programmed as follows:

```
#SET AX [U,V,X2]
#TOOL AX [X2+]
G1 X2=25 F1000 S1000
G81 X2=2 I-8 K1
```

1.

MILLING CANNED CYCLES.
General concepts



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

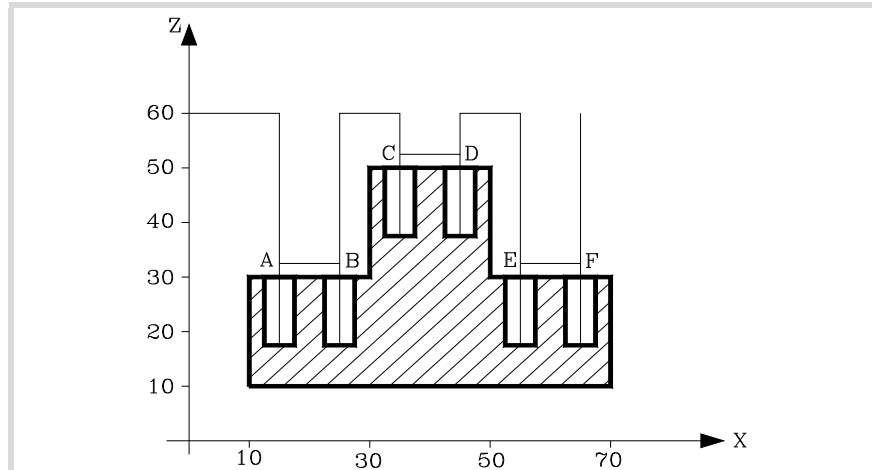
REF: 2102

1.1.5 Modifying the parameters of a canned cycle.

Inside the influence zone of a canned cycle, it is possible to modify one or several parameters of the cycle without having to redefine it. After modifying the parameters, the CNC keeps the canned cycle active and carries out the machining operations with the updated parameters.

The parameters of the cycle are modified with the V.C.A variable for parameter ·A·, V.C.B for parameter ·B· and so on. The values of these variables are defined in absolute coordinates with respect to part zero.

Here are two programming examples where the work plane is XY (X as abscissa axis and Y as ordinate axis) and Z as longitudinal axis.



```

T1 M6
G00 G90 X0 Y0 Z60 F1000
    Starting point.
G99 G91 X15 Y25 G81 Z-28 I-14
    Drilling canned cycle. Drilling A.
G98 G90 X25
    Drilling canned cycle. Drilling B.
V.C.Z=52 V.C.I=40
    New reference plane and machining penetration.
G99 X35
    Drilling canned cycle. Drilling C.
G98 X45
    Drilling canned cycle. Drilling D.
V.C.Z=32 V.C.I=18
    New reference plane and machining penetration.
G99 X55
    Drilling canned cycle. Drilling E.
G98 X65
    Drilling canned cycle. Drilling F.
M30
  
```

1.

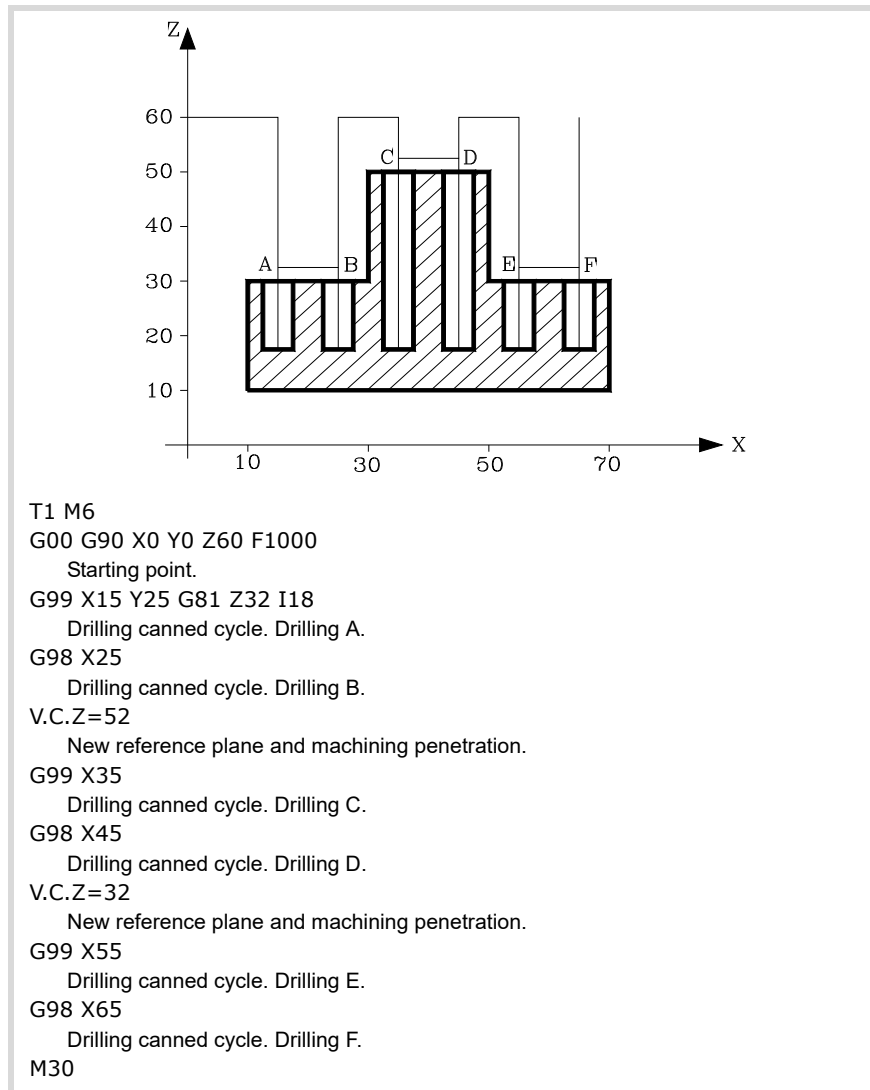
MILLING CANNED CYCLES.
General concepts



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102



1.

MILLING CANNED CYCLES.
 General concepts



FAGOR AUTOMATION

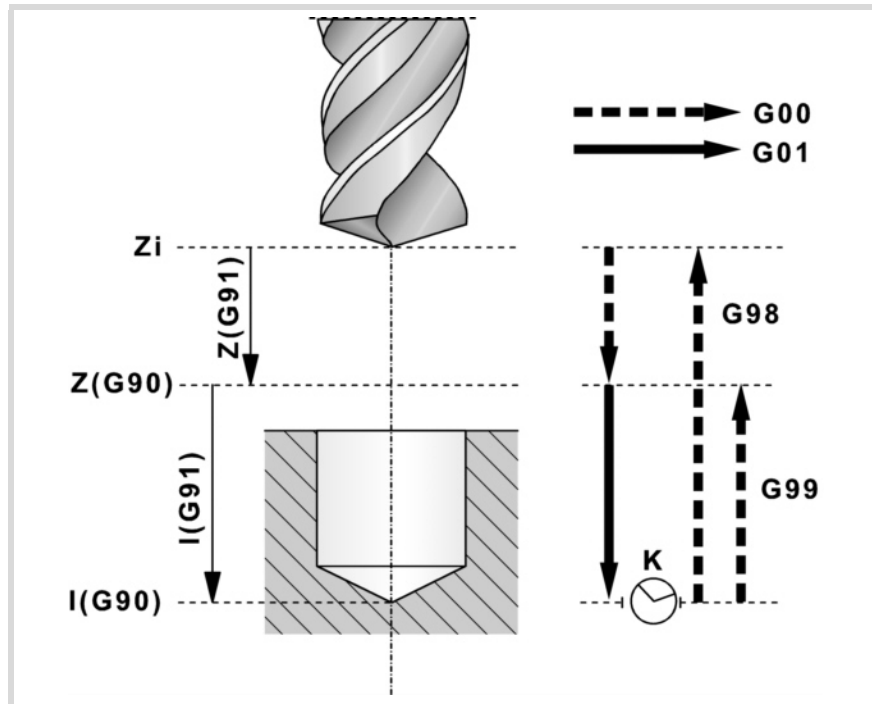
CNC 8058
 CNC 8060
 CNC 8065

REF: 2102

1.2 G81. Drilling canned cycle

Programming format in Cartesian coordinates:

G81 Z I K A



Parameter definition:

- Z** Reference plane.
 In G90, coordinate referred to part zero.
 In G91, coordinate referred to starting plane (Zi).
 If not programmed, it assumes as reference plane the current position of the tool (Z=Zi).
- I** Drilling depth.
 In G90, coordinate referred to part zero.
 In G91, coordinate referred to reference plane (Z).
- K** Delay, in seconds, between the drilling and the withdrawal movement.
 If not programmed, it assumes K0.
- A** Spindle behavior when going into the hole and coming out of it.
 A0: The tool goes into the hole and comes out turning.
 A1: The tool goes into the hole turning and comes out stopped.
 If not programmed, it assumes a value of A0.

Basic operation:

- 1 If the spindle was previously running, it maintains the turning direction. If it is stopped, it starts clockwise (M03).
- 2 Rapid movement (G0) of the longitudinal axis from the starting plane (Zi) to the reference plane (Z).
- 3 Drill the hole. Movement of the longitudinal axis at work feedrate, to the bottom of the hole programmed in "I".
- 4 Dwell, in seconds, if it has been programmed.
- 5 Rapid withdrawal (G0) to the starting plane (Zi) if function G98 is active or to the reference plane (Z) if function G99 is active. The spindle will come out turning or stopped depending on parameter "A".

1.

MILLING CANNED CYCLES.
 G81. Drilling canned cycle

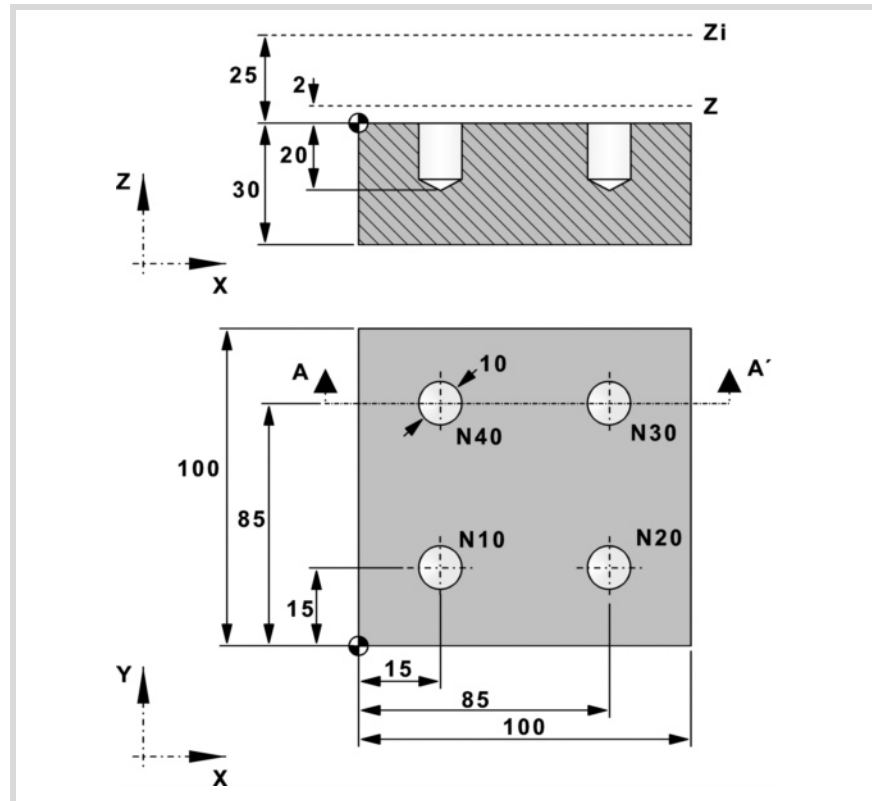


FAGOR AUTOMATION

CNC 8058
 CNC 8060
 CNC 8065

REF: 2102

1.2.1 Programming example



Absolute programming:

```
T1 D1 M6
S1000 M3 M8 M41
G0 G90 X0 Y0 Z25 F200
N10 G99 X15 Y15 G81 Z2 I-20
N20 X85
N30 Y85
N40 G98 X15
M30
```

Incremental programming:

```
T1 D1 M6
S1000 M3 M8 M41
G0 G90 X0 Y0 Z25 F200
N10 G99 G91 X15 Y15 G81 Z-23 I-22
N20 X70
N30 Y70
N40 G98 X-70
M30
```

1.

MILLING CANNED CYCLES:
G81. Drilling canned cycle



FAGOR AUTOMATION

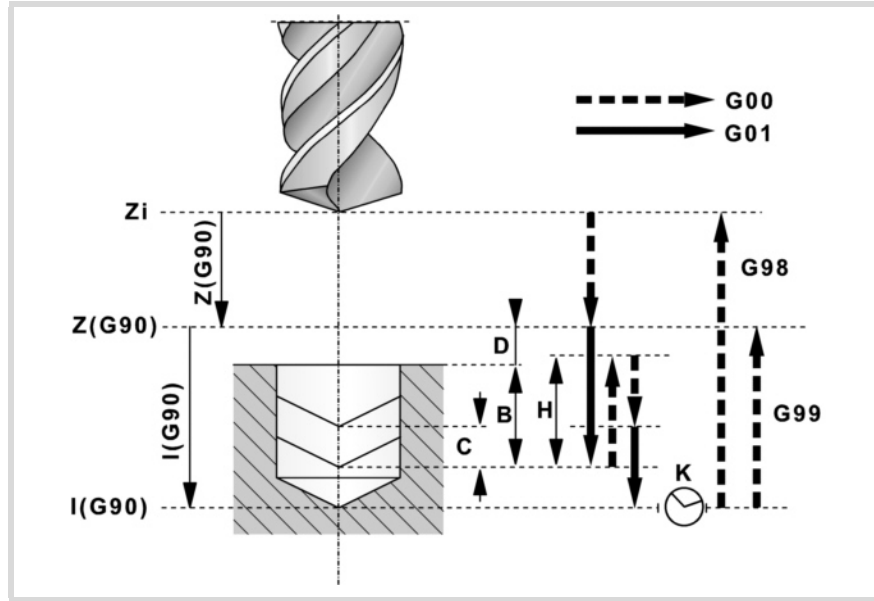
CNC 8058
CNC 8060
CNC 8065

REF: 2102

1.3 G82. Drilling canned cycle with variable peck

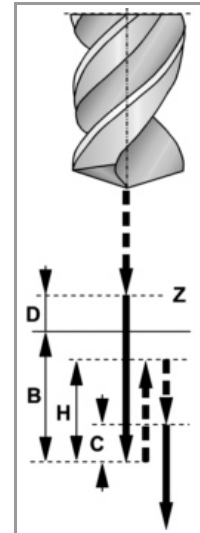
Programming format in Cartesian coordinates:

G82 Z I D B H C J K R L A



Parameter definition:

- Z** Reference plane.
 In G90, coordinate referred to part zero.
 In G91, coordinate referred to starting plane (Z_i).
 If not programmed, it assumes as reference plane the current position of the tool ($Z=Z_i$).
- I** Drilling depth.
 In G90, coordinate referred to part zero.
 In G91, coordinate referred to reference plane (Z).
- D** Distance between the reference plane and the part surface.
 If not programmed, it assumes a value of 0.
- B** Drilling peck (step).
 All the pecks have this value, except the last one that is adjusted to the total depth.
- H** Distance or coordinate it returns to, in rapid (G0), after each drilling step.
 "J" other than 0 means the distance and "J=0" indicates the relief coordinate or absolute coordinate it withdraws to.
 If not programmed, it returns to the reference plane.
- C** Approach coordinate.
 It defines the rapid approach (G0) distance of the longitudinal axis from the previous drilling peck to carry out a new drilling peck.
 If not programmed, it assumes 1 mm.
 It issues an error message if "C=0" is programmed.
- J** It defines after how many drilling pecks the tool returns in rapid (G0) to the reference plane (Z).
 With "J" greater than 1, after each peck, the tool returns the distance indicated by "H" and every "J" pecks to the reference plane (Z).
 With "J=1", it returns to the reference plane (Z) after each peck.
 If "J" is not programmed or "J=0" is programmed, it returns to the relief coordinate indicated by "H" after each peck.



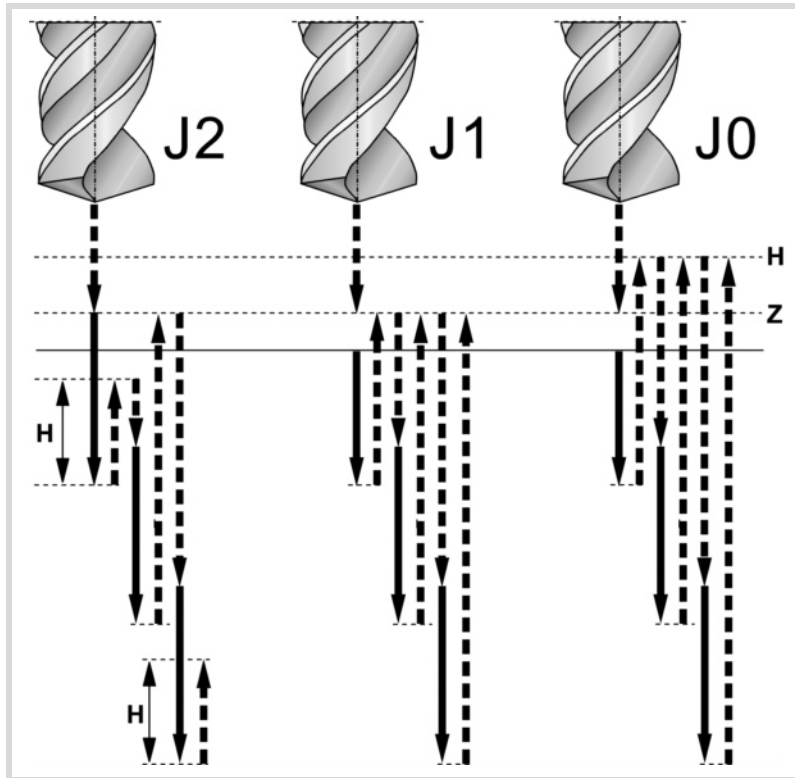
1.

MILLING CANNED CYCLES.
 G82. Drilling canned cycle with variable peck

FAGOR 
 FAGOR AUTOMATION

CNC 8058
 CNC 8060
 CNC 8065

REF: 2102



- K Dwell, in seconds, at the bottom of the hole.
If not defined, it assumes a value of 0.
- R Factor that increases or reduces the drilling peck (step) "B".
The first peck will be "B", the second "RB", the third "R(RB)" and so on.
If it is not programmed or "R=0" is programmed, it assumes "R=1". With "R=1", all the drilling pecks will have the value of "B".
- L Minimum value for the drilling peck. It is used with "R" values other than 1. If not programmed or programmed with a 0 value, there is no minimum drilling distance.
- A Spindle behavior when going into the hole and coming out of it.
 - A0: The tool goes into the hole and comes out turning.
 - A1: The tool goes into the hole turning and comes out stopped.
 If not programmed, it assumes a value of A0.

Basic operation:

- 1 If the spindle was previously running, it maintains the turning direction. If it is stopped, it starts clockwise (M03).
- 2 Rapid movement (G0) of the longitudinal axis from the starting plane (Zi) to the reference plane (Z).
- 3 First drilling penetration, at work feedrate. The distance indicated by "B", from the part surface.
- 4 Drilling loop until reaching the total drilling depth programmed in "I".
 - Rapid withdrawal (G0).
With "J=1", it returns to the reference plane (Z) after each peck.
If "J" is not programmed or "J=0" is programmed, it returns to the relief coordinate indicated by "H" after each peck.
With "J" greater than 1, after each peck, the tool returns the distance indicated by "H" and every "J" pecks to the reference plane (Z).
 - Rapid approach (G0) to a distance "C" or up to 1 mm from the previous drilling step (peck).
 - New drilling peck, at work feedrate. The distance indicated by "B" and "R".

1.

MILLING CANNED CYCLES.
G82. Drilling canned cycle with variable peck



FAGOR AUTOMATION

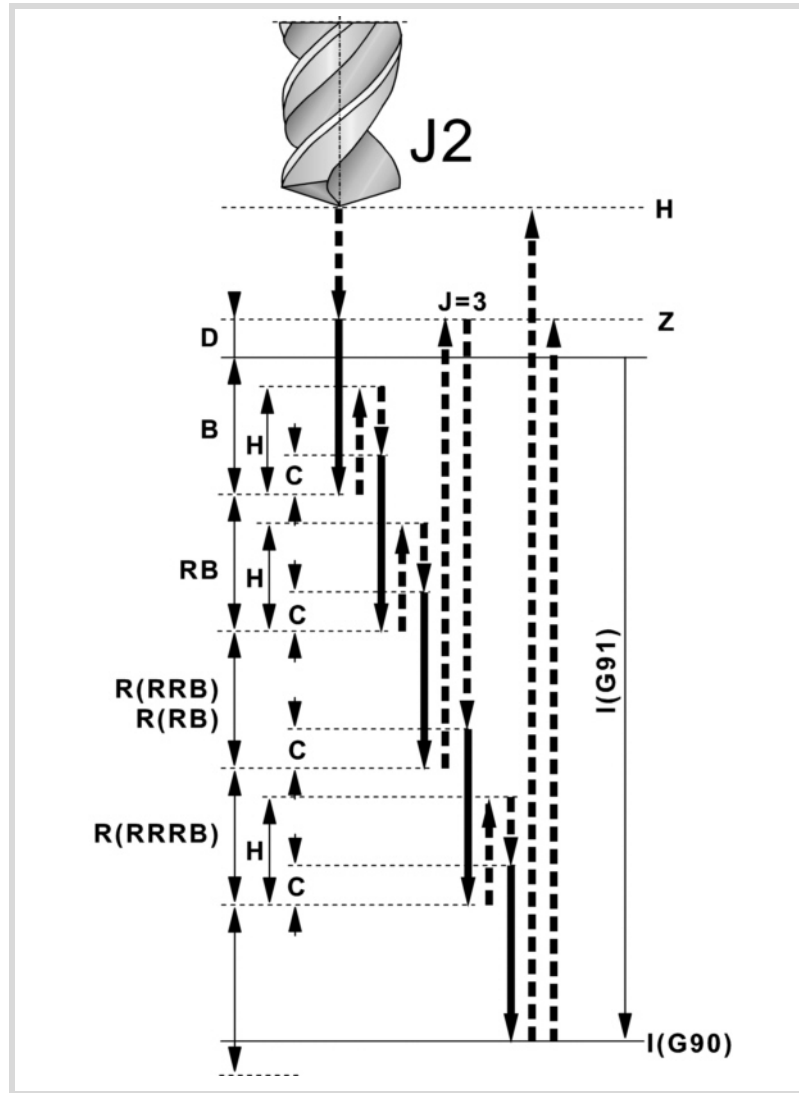
CNC 8058
CNC 8060
CNC 8065

REF: 2102

1.

MILLING CANNED CYCLES.

G82. Drilling canned cycle with variable peck



- 5 Dwell at the bottom of the hole. The time indicated by "K" in seconds.
- 6 Rapid withdrawal (G0) to the starting plane (Zi) if function G98 is active or to the reference plane (Z) if function G99 is active. The spindle will come out turning or stopped depending on parameter "A".

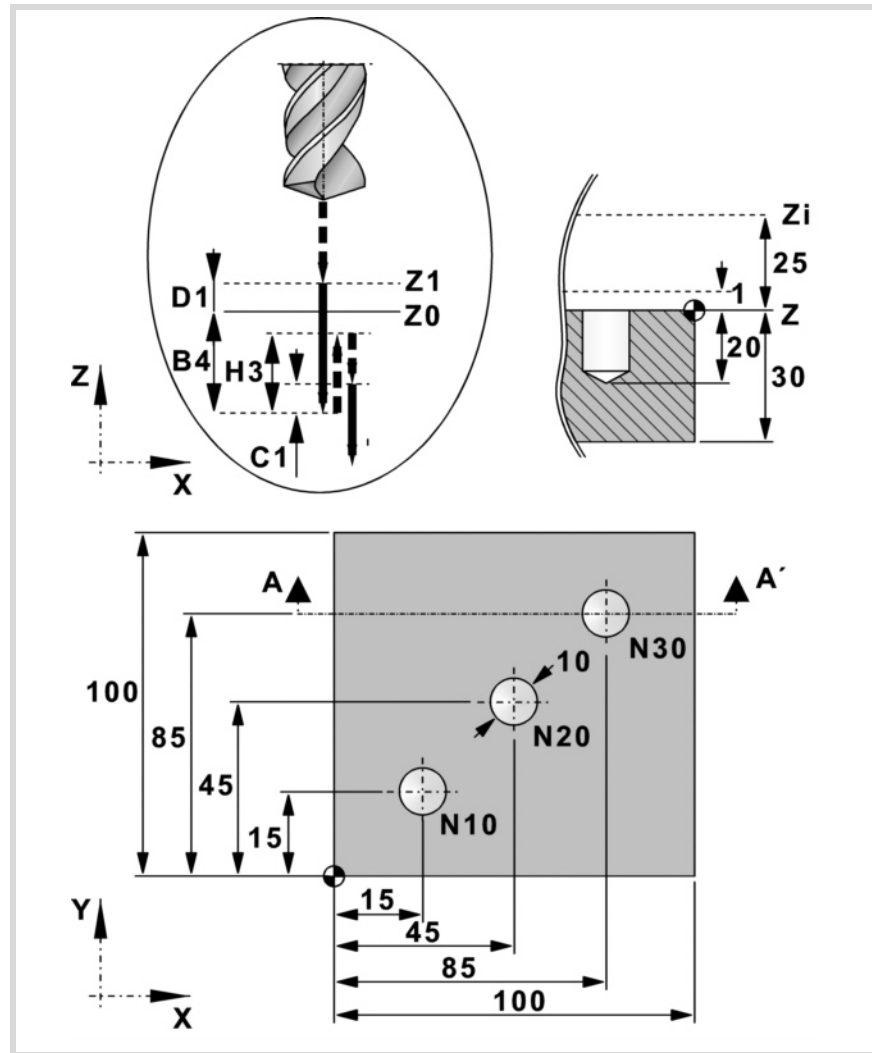


FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

1.3.1 Programming example



Absolute programming:

```
T2 D1 M6
S1000 M3 M8 M41
G0 G90 X0 Y0 Z25 F200
N10 G99 X15 Y15
G82 Z1 I-20 D1 B4 H3 C1 J3 K1 R0.8 L3
N20 X45 Y45
N30 G98 X85 Y85
M30
```

Incremental programming:

```
T2 D1 M6
S1000 M3 M8 M41
G0 G90 X0 Y0 Z25 F200
N10 G99 G91 X15 Y15
G82 Z-24 I-21 D1 B4 H3 C1 J3 K1 R0.8 L3
N20 X30 Y30
N30 G98 X40 Y40
M30
```

1.

MILLING CANNED CYCLES.

G82. Drilling canned cycle with variable peck



FAGOR AUTOMATION

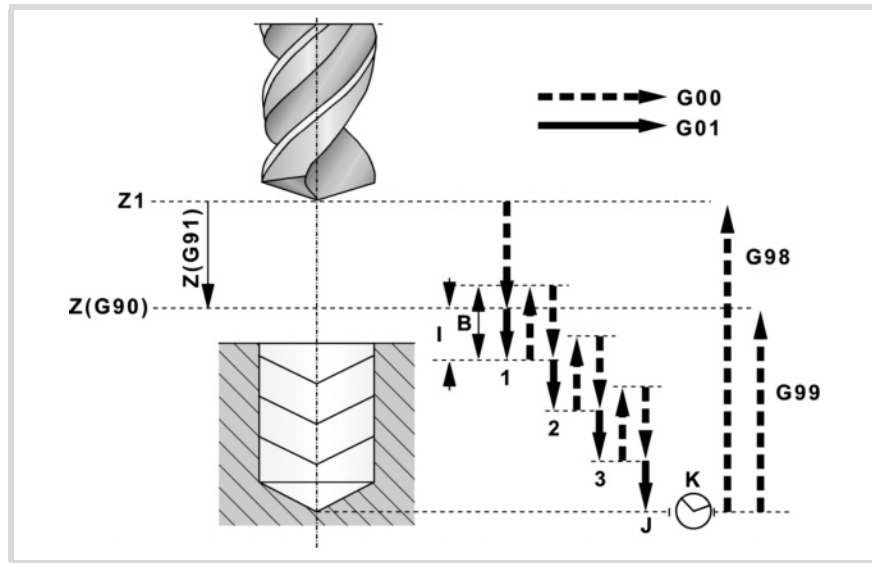
CNC 8058
CNC 8060
CNC 8065

REF: 2102

1.4 G83. Deep-hole drilling canned cycle with constant peck

Programming format in Cartesian coordinates:

G83 Z I J B K



Parameter definition:

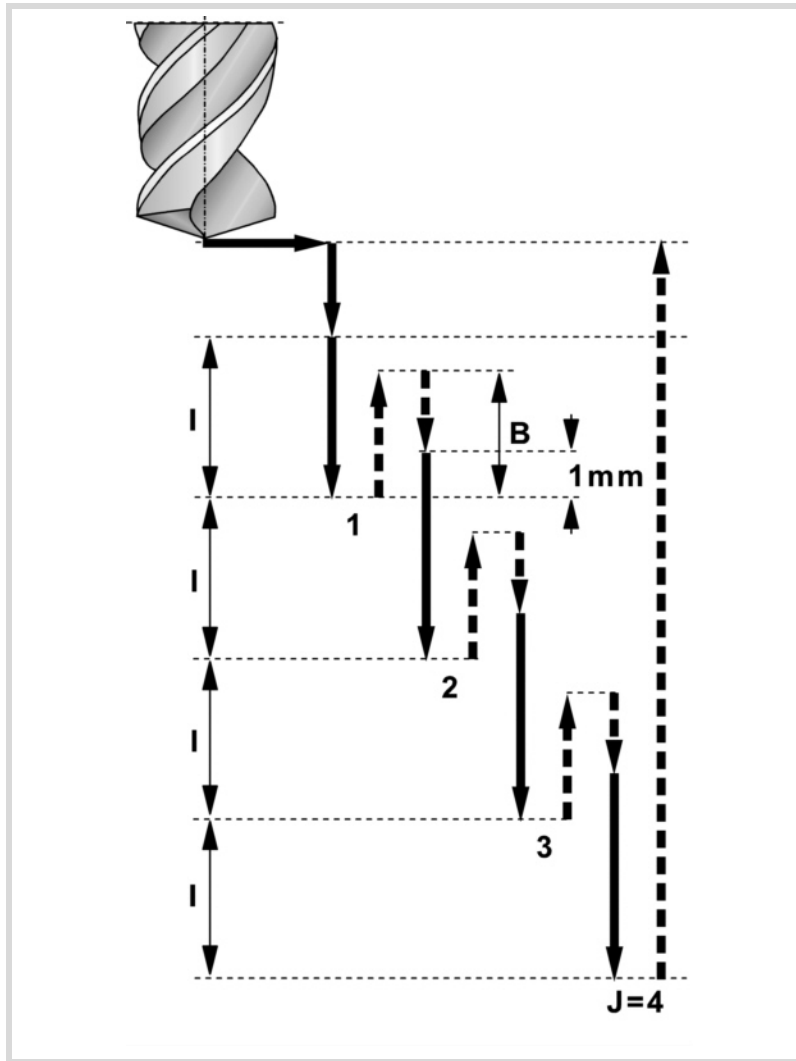
- Z** Reference plane.
 In G90, coordinate referred to part zero.
 In G91, coordinate referred to starting plane (Z1).
 If not programmed, it assumes as reference plane the current position of the tool (Z=Z1).
- I** Drilling peck (step).
 The sign indicates the machining direction. Positive towards plus coordinate and negative towards minus. In the figure "I".
- J** Number of pecks required by the drilling operation.
- B** Rapid withdraw (G0) distance after each drilling step.
 If not programmed, it returns to the reference plane.
- K** Dwell, in seconds, at the bottom of the hole.
 If not defined, it assumes a value of 0.

Basic operation:

- 1 If the spindle was previously running, it maintains the turning direction. If it is stopped, it starts clockwise (M03).
- 2 Rapid movement (G0) of the longitudinal axis from the starting plane (Z1) to the reference plane (Z).
- 3 Drilling loop. The following steps are repeated "J" times.
 - Drilling peck, at work feedrate. The distance indicated by "I".
 - Rapid withdrawal (G0). The "B" distance or to the reference plane.
 - Rapid approach (G0) up to 1 mm from the previous drilling step (peck).

1.

MILLING CANNED CYCLES.
G83. Deep-hole drilling canned cycle with constant peck



- 4 Dwell at the bottom of the hole. The time indicated by "K" in seconds.
- 5 Rapid withdrawal (G0) to the starting plane (Zi) if function G98 is active or to the reference plane (Z) if function G99 is active.

1.

MILLING CANNED CYCLES.

G83. Deep-hole drilling canned cycle with constant peck

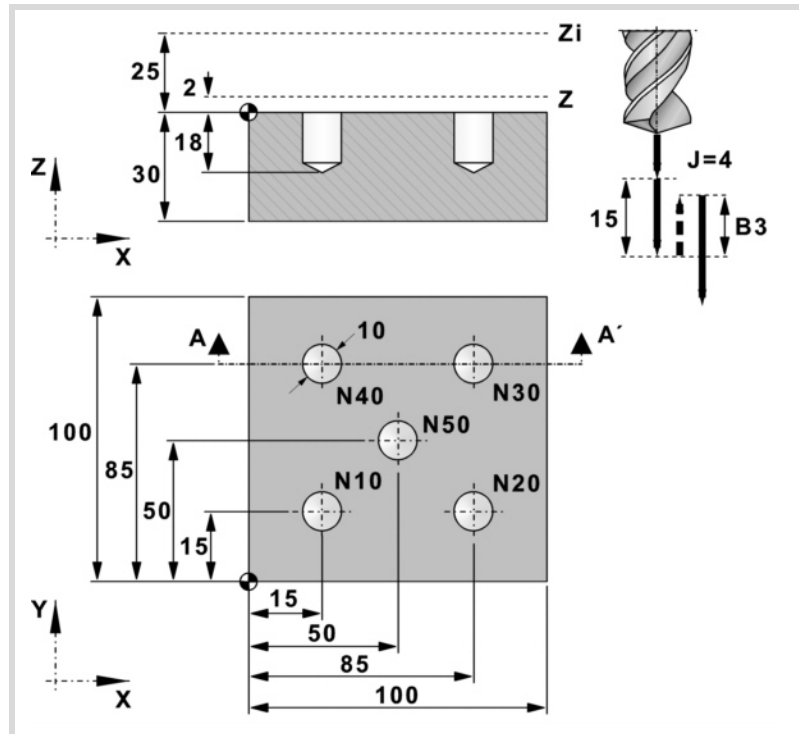
FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

1.4.1 Programming example



1.

MILLING CANNED CYCLES.
G83. Deep-hole drilling canned cycle with constant peck

Absolute programming:

```
T3 D1 M6
S1000 M3 M8 M41
G0 G90 X0 Y0 Z25 F200
N10 G99 X15 Y15
G83 Z2 I-5 J4 B3 K1
N20 X85
N30 Y85
N40 X15
N50 G98 X50 Y50
M30
```

Incremental programming:

```
T3 D1 M6
S1000 M3 M8 M41
G0 G90 X0 Y0 Z25 F200
N10 G99 G91 X15 Y15
G83 Z-23 I-5 J4 B3 K1
N20 X70
N30 Y70
N40 X-70
N50 G98 X35 Y-35
M30
```

FAGOR 
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

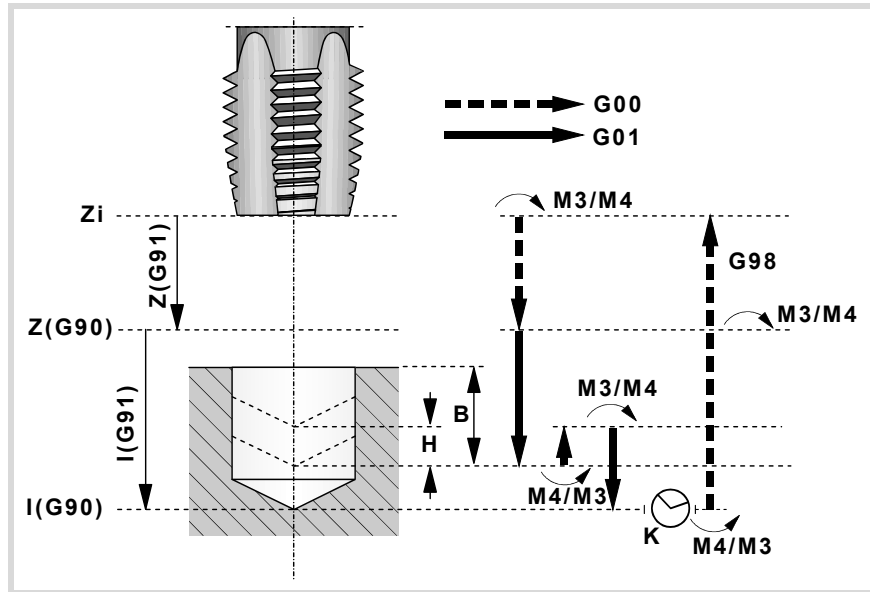
REF: 2102

1.5 G84. Tapping canned cycle

Both tapping with a clutch and rigid tapping are possible. For rigid tapping, the spindle must have a motor-drive system and a spindle encoder.

Programming format in Cartesian coordinates:

G84 Z I K R J B H



Parameter definition:

- Z Reference plane.
In G90, coordinate referred to part zero.
In G91, coordinate referred to starting plane (Zi).
If not programmed, it assumes as reference plane the current position of the tool (Z=Zi).
- I Tap depth.
In G90, coordinate referred to part zero.
In G91, coordinate referred to reference plane (Z).
- K Delay, in seconds, between the tapping and the withdrawal movement.
If not programmed, it assumes K0.
- R Type of tapping.
R0: normal tapping.
R1: rigid tapping.
- J Withdrawal feedrate factor.
When rigid tapping, the returning feedrate will be J times the tapping feedrate. When not programmed or programmed J1, they will both be the same.
- B Penetration step when tapping with chip removal (only for R1). 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.
This parameter may be used to execute the cycle in several passes, withdrawing a distance after each pass (parameter H) to remove the chips.
- H Withdrawal distance after each penetration step (only for R1). 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 J).

1.

MILLING CANNED CYCLES.
G84. Tapping canned cycle

FAGOR 
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

1.

MILLING CANNED CYCLES.
G84. Tapping canned cycle

Basic operation:

- 1 If the spindle was previously running, it maintains the turning direction. If it is stopped, it starts clockwise (M03).
- 2 Rapid movement (G0) of the longitudinal axis from the starting plane (Zi) to the reference plane (Z).
- 3 Tapping, at 100% of the feedrate "F" and spindle speed "S" programmed. Tapping cannot be interrupted.

R=0 Regular tapping.

R=1 Rigid tapping.

If parameter B has not been programmed, the cycle will execute the tap in a single pass. If parameter B has been programmed, the cycle executes the tap in several passes withdrawing an "H" distance after each pass.

- 4 If "K" other than 0, spindle stop (M05) and dwell.
- 5 The spindle reverses turning direction. Withdrawal and exit the tap to the reference plane. The cycle does the withdrawal taking the feedrate factor into account (parameter J). The thread exit cannot be interrupted.
- 6 Depending on the type of tap programmed.
 - R=0 Reverse the spindle turning direction restoring the initial turning direction.
 - R=1 Spindle orientation (M19).
- 7 If function G98 is active, rapid withdraw to the starting plane (Zi).

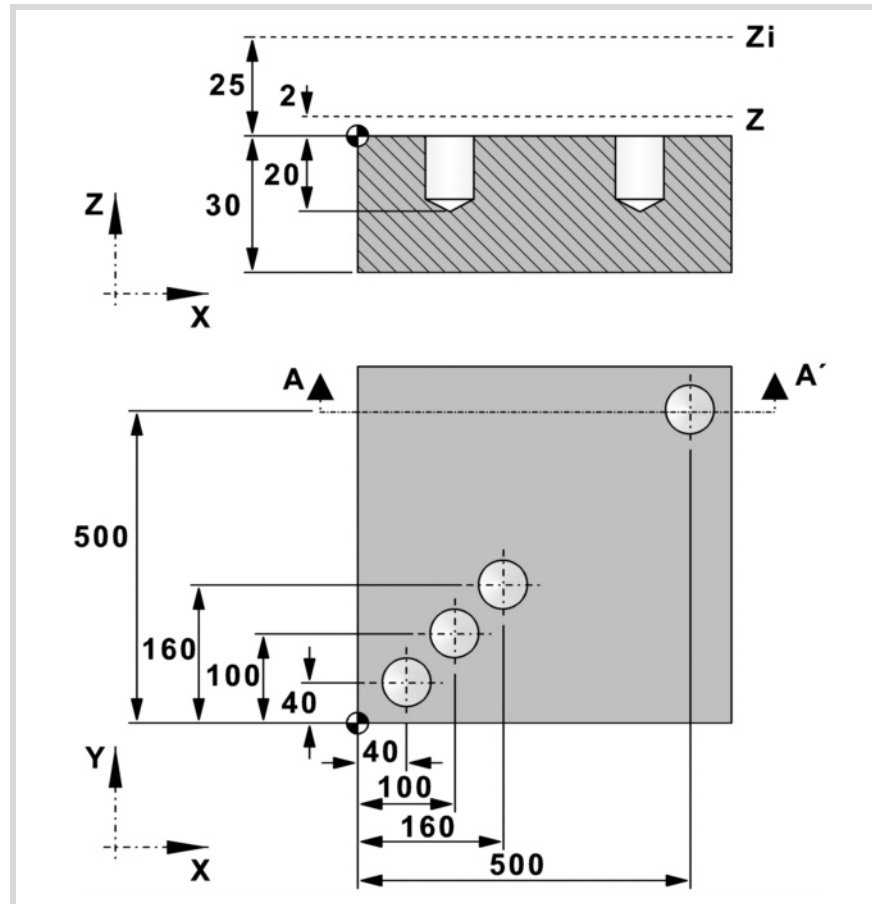


FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

1.5.1 Programming example



Absolute programming:

```
T4 D1 M6
S1000 M3 M8 M41
G0 G90 X0 Y0 Z25 F200
N10 G99 X40 Y40 G84 Z2 I-20 K1 R0
N20 X100 Y100
N30 X160 Y160
N40 G98 X500 Y500
M30
```

Incremental programming:

```
T4 D1 M6
S1000 M3 M8 M41
G0 G90 X0 Y0 Z25 F200
N10 G99 G91 X40 Y40 G84 Z-23 I-22 K1 R0
$FOR P0=1,2,1
X60 Y60
$ENDFOR
G98 X340 Y340
M30
```

1.

MILLING CANNED CYCLES:
G84. Tapping canned cycle



FAGOR AUTOMATION

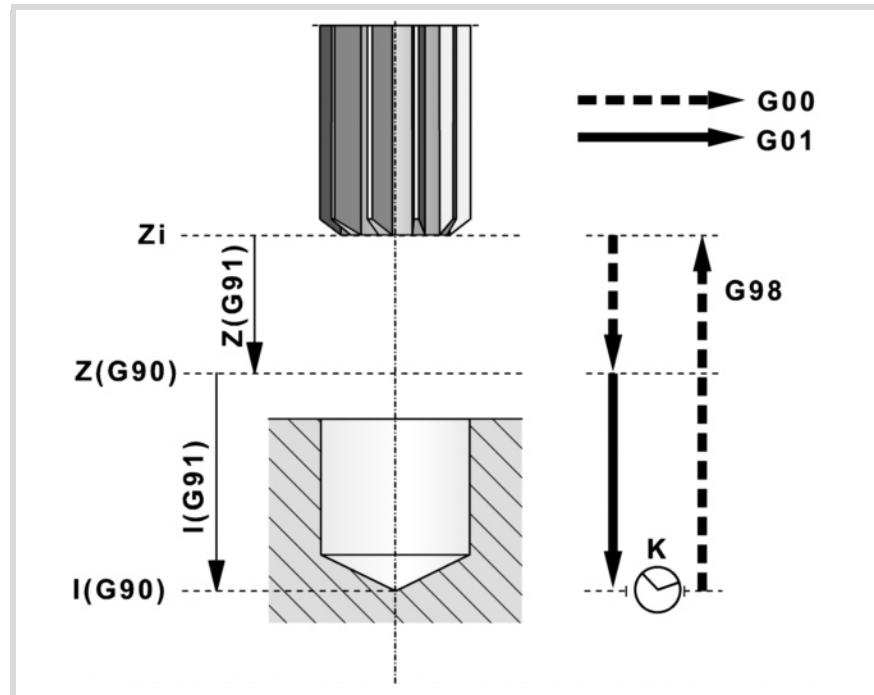
CNC 8058
CNC 8060
CNC 8065

REF: 2102

1.6 G85. Reaming canned cycle

Programming format in Cartesian coordinates:

G85 Z I K



Parameter definition:

- Z** Reference plane.
 In G90, coordinate referred to part zero.
 In G91, coordinate referred to starting plane (Z_i).
 If not programmed, it assumes as reference plane the current position of the tool ($Z=Z_i$).
- I** Reaming depth.
 In G90, coordinate referred to part zero.
 In G91, coordinate referred to reference plane (Z).
- K** Delay, in seconds, between the reaming and the withdrawal movement.
 If not programmed, it assumes $K0$.

Basic operation:

- 1 If the spindle was previously running, it maintains the turning direction. If it is stopped, it starts clockwise (M03).
- 2 Rapid movement (G0) of the longitudinal axis from the starting plane (Z_i) to the reference plane (Z).
- 3 Reaming the hole. Movement of the longitudinal axis at work feedrate, to the bottom of the hole programmed in "I".
- 4 Dwell, in seconds, if it has been programmed.
- 5 Withdrawal, at work feedrate (G01) up to the reference plane (Z).
- 6 If function G98 is active, rapid withdraw to the starting plane (Z_i).

1.

MILLING CANNED CYCLES.
 G85. Reaming canned cycle

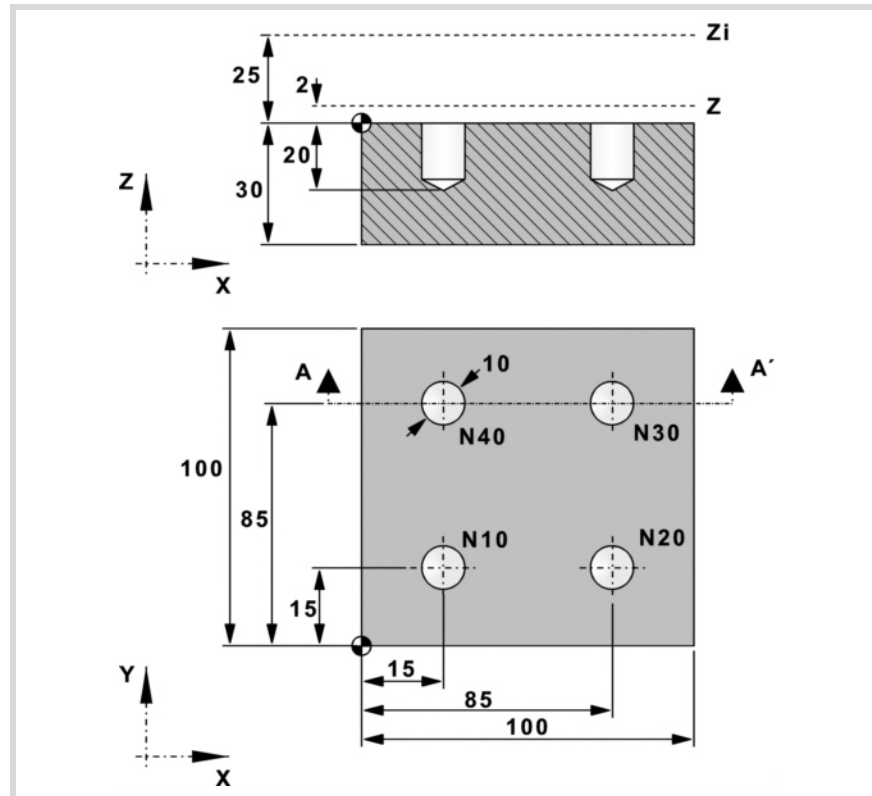


FAGOR AUTOMATION

CNC 8058
 CNC 8060
 CNC 8065

REF: 2102

1.6.1 Programming example



Absolute programming:

```
T5 D1 M6
S1000 M3 M8 M41
G0 G90 X0 Y0 Z25 F200
N10 G99 X15 Y15 G85 Z2 I-20
N20 X85
N30 Y85
N40 G98 X15
M30
```

Incremental programming:

```
T5 D1 M6
S1000 M3 M8 M41
G0 G90 X0 Y0 Z25 F200
N10 G99 G91 X15 Y15 G85 Z-23 I-22
N20 X70
N30 Y70
N40 G98 X-70
M30
```

1.

MILLING CANNED CYCLES:
G85. Reaming canned cycle



FAGOR AUTOMATION

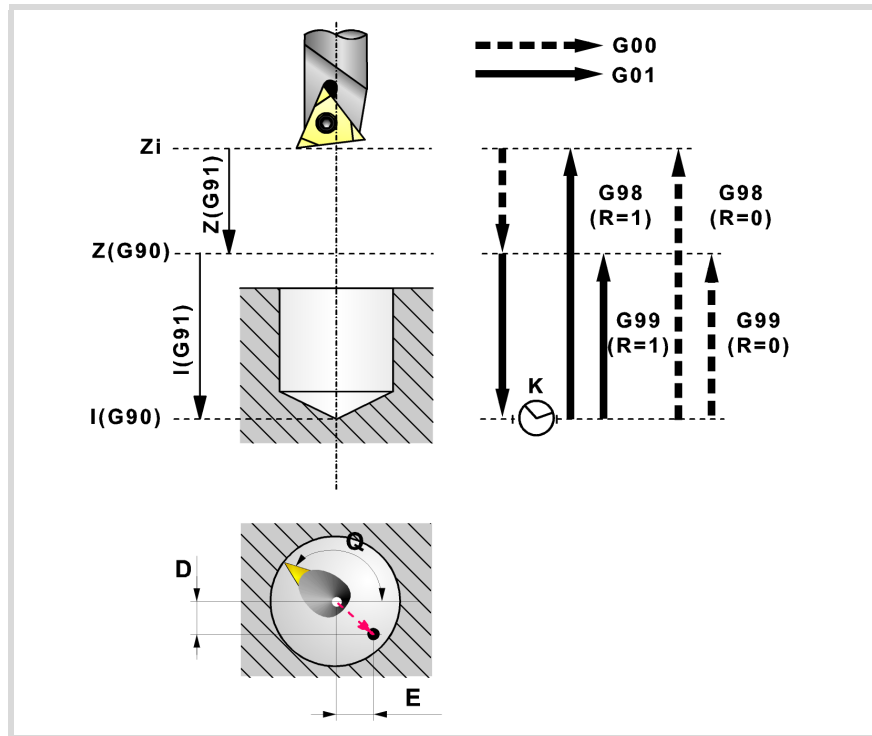
CNC 8058
CNC 8060
CNC 8065

REF: 2102

1.7 G86. Boring canned cycle

Programming format in Cartesian coordinates:

G86 Z I K R A Q D E



Parameter definition:

- Z** Reference plane.
 In G90, coordinate referred to part zero.
 In G91, coordinate referred to starting plane (Zi).
 If not programmed, it assumes as reference plane the current position of the tool ($Z=Zi$).
- I** Boring depth.
 In G90, coordinate referred to part zero.
 In G91, coordinate referred to reference plane (Z).
- K** Delay, in seconds, between the boring and the withdrawal movement.
 If not programmed, it assumes K0.
- R** Type of withdrawal, when parameter A not programmed.
 R0: Rapid withdrawal (G0) with the spindle stopped.
 R1: Withdraw at work feedrate (G1).
 If not programmed, it assumes a value of R0.
- A** Spindle behavior when going into the hole and coming out of it.
 A0: The tool goes into the hole turning and comes out stopped.
 A1: The tool goes into the hole stopped and comes out turning.
 If not programmed, the tool goes into the hole and comes out turning. When programming A0 or A1, define the spindle stop with parameters "Q", "D" and "E".
- Q** Spindle position, in degrees, to separate the cutter from the wall of the hole.
 When programming A0 or A1, this parameter sets the tool orientation and parameters "D" and "E" set the distance the tool withdraws off (away from) the walls of the hole.
- D** Distance to withdraw the cutter off the wall of the hole along the abscissa axis.
 Bear in mind the orientation of the spindle to define the moving direction; with the wrong direction, the tool may collide with the hole instead of moving away from it.
 The cycle takes this parameter into consideration only when parameter "Q" has been programmed.

1.

MILLING CANNED CYCLES.
 G86: Boring canned cycle

FAGOR 
 FAGOR AUTOMATION

CNC 8058
 CNC 8060
 CNC 8065

REF: 2102

- E Distance to withdraw the cutter off the wall of the hole along the ordinate axis.
 Bear in mind the orientation of the spindle to define the moving direction; with the wrong direction, the tool may collide with the hole instead of moving away from it.
 The cycle takes this parameter into consideration only when parameter "Q" has been programmed.

Basic operation:

- 1 Depending on parameter "A", spindle start or stop.
 - If the spindle must go in turning and it is already turning, it maintains its turning direction. If the spindle is stopped, it starts clockwise (M03).
 - If the spindle must go in stopped and it is already turning, it stops.
- 2 Rapid movement (G0) of the longitudinal axis from the starting plane (Zi) to the reference plane (Z).
- 3 Depending on parameter "A", the spindle will go into the hole and come out of it either turning or stopped.
 - If the spindle must go in turning, hole boring. Movement of the longitudinal axis at work feedrate, to the bottom of the hole programmed in "I".
 - If the spindle must go in stopped, the tool is oriented in the position set by parameter "Q" and moves off the wall the distance programmed in parameters "D" and "E". Movement of the longitudinal axis at work feedrate, to the bottom of the hole programmed in "I". The spindle approaches the wall of the hole the distance programmed in parameters "D" and "E".
- 4 Dwell, in seconds, if it has been programmed.
- 5 Depending on parameter "A", the spindle will come out of the hole and come out of it either turning or stopped.
 - not A If not programmed, withdrawal to the starting plane (Zi) if function G98 is active or to the reference plane (Z) if function G99 is active. The withdrawal is done in rapid (G0) and the spindle stopped when programmed "R=0" or at work feedrate (G01) and the spindle running when programmed "R=1".
 - A0 Stop the spindle. The tool is oriented in the position set by parameter "Q" and moves off the wall the distance programmed in parameters "D" and "E". Withdrawal to the starting plane (Zi) if function G98 is active or to the reference plane (Z) if function G99 is active. The spindle approaches the wall of the hole the distance programmed in parameters "D" and "E".
 - A1 The spindle starts in the same direction as it was turning before. Withdrawal to the starting plane (Zi) if function G98 is active or to the reference plane (Z) if function G99 is active.
- 6 If the spindle is stopped, it will start in the same direction as it was turning before.



MILLING CANNED CYCLES.
G86. Boring canned cycle

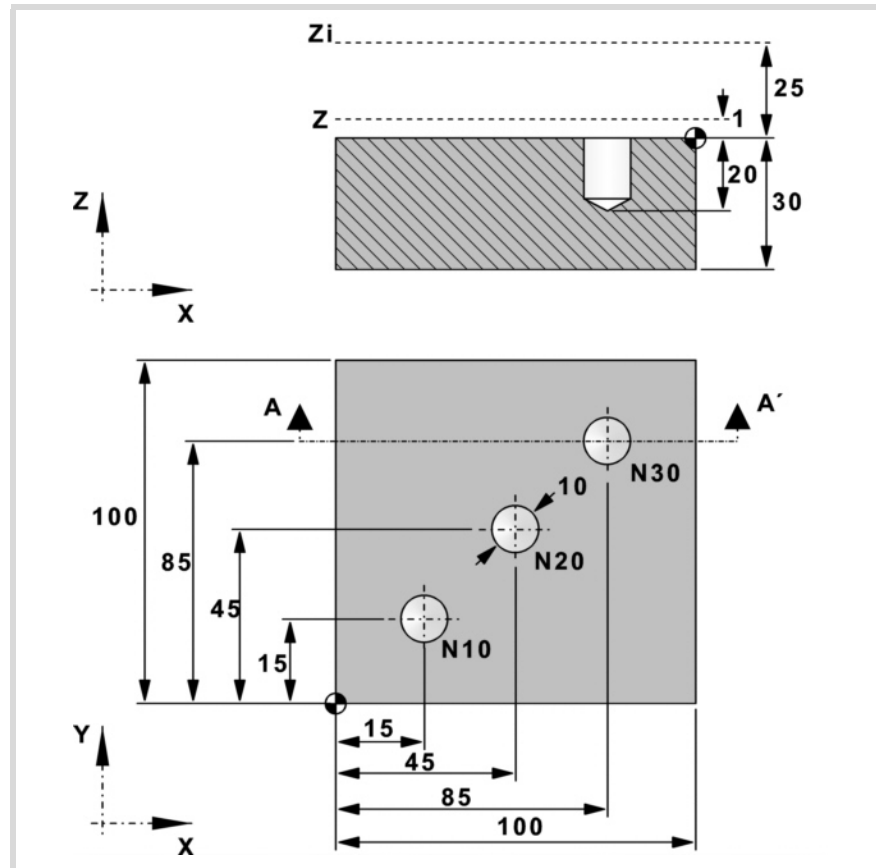


FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

1.7.1 Programming example



1.

MILLING CANNED CYCLES.
G86: Boring canned cycle

Absolute programming with R=0:

```
T6 D1 M6
S1000 M3 M8 M41
G0 G90 X0 Y0 Z25 F200
N10 G99 X15 Y15 G86 Z2 I-20 K3 R0
N20 X45 Y45
N30 G98 X85 Y85
M30
```

Incremental programming with R=1:

```
T6 D1 M6
S1000 M3 M8 M41
G0 G90 X0 Y0 Z25 F200
N10 G99 G91 X15 Y15 G86 Z-23 I-22 K3 R1
N20 X30 Y30
N30 G98 X40 Y40
M30
```

FAGOR 
FAGOR AUTOMATION

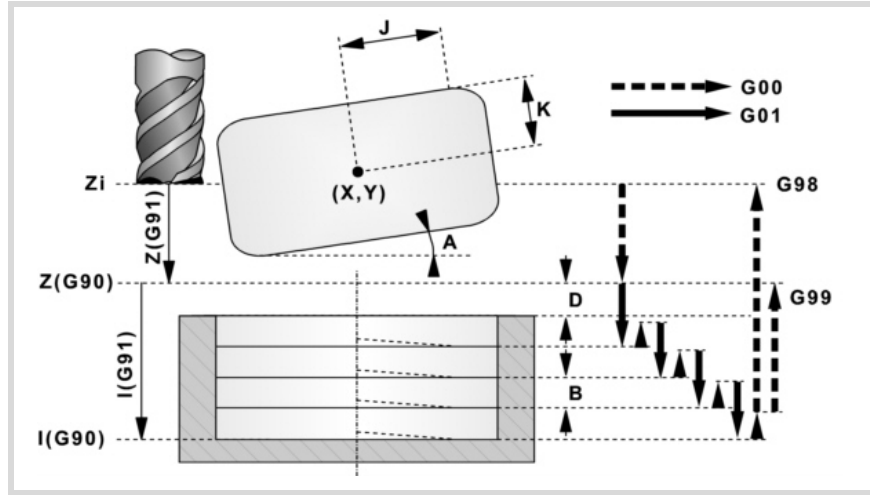
CNC 8058
CNC 8060
CNC 8065

REF: 2102

1.8 G87. Rectangular pocket canned cycle.

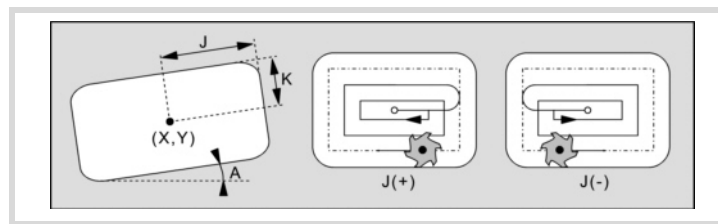
Programming format in Cartesian coordinates:

G87 Z I D A J K M Q B C L H V

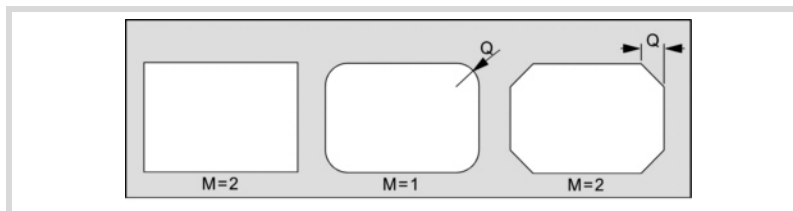


Parameter definition:

- Z Reference plane.
In G90, coordinate referred to part zero.
In G91, coordinate referred to starting plane (Zi).
If not programmed, it assumes as reference plane the current position of the tool (Z=Zi).
- I Pocket depth.
In G90, coordinate referred to part zero.
In G91, coordinate referred to reference plane (Z).
- D Distance between the reference plane and the part surface. If not programmed, it assumes a value of 1 mm.
- A Angle, in degrees, between the pocket and the abscissa axis. If not programmed, it assumes a value of 0.
- J Half length of the pocket.
The sign indicates the pocket machining direction:
(J+) clockwise, (J-) counterclockwise.



- K Half width of the pocket.
- M Type of corner. (0) square, (1) rounded, (2) chamfered. If not programmed, it assumes a value of 0.



- Q Rounding radius or chamfer size.

1.

MILLING CANNED CYCLES.
G87. Rectangular pocket canned cycle.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

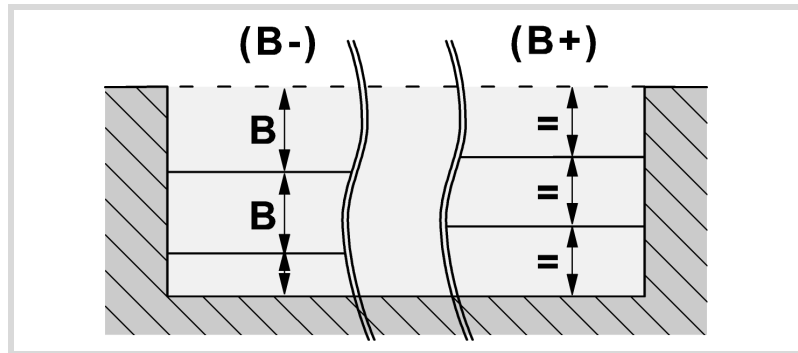
1.

MILLING CANNED CYCLES.
G87. Rectangular pocket canned cycle.

B Depth of pass.

If programmed with a positive sign (B+), 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 (B-), the pocket is machined with the given pass (step) except the last pass that machines the rest.

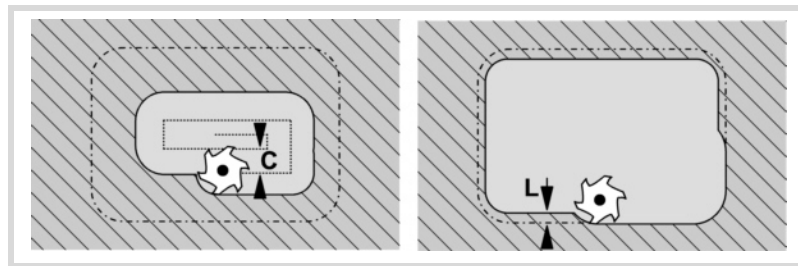


C Milling pass or width.

If not programmed or programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

If it is the same as parameter "J" or "K" (half length/half width of the pocket) it only runs the finishing pass.

If programmed with a value greater than the tool diameter, the CNC issues the relevant error message.



L Finishing pass.

If not programmed or programmed with a 0 value, it does not run the finishing pass.

H Feedrate for the finishing pass. If not programmed or programmed with a 0 value, it is carried out at the roughing feedrate.

V Tool penetrating feedrate. If not programmed or programmed with a 0 value, it is carried out at 50% of the feedrate in the plane.

Basic operation:

- 1 If the spindle was previously running, it maintains the turning direction. If it is stopped, it starts clockwise (M03).
- 2 Rapid movement (G0) of the longitudinal axis from the starting plane (Zi) to the reference plane (Z).



FAGOR AUTOMATION

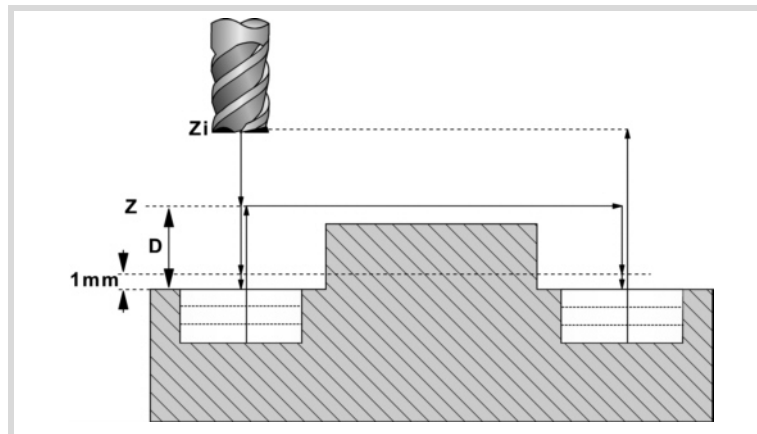
CNC 8058
CNC 8060
CNC 8065

REF: 2102

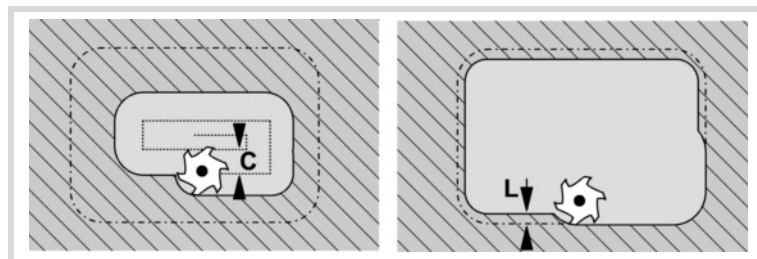
Canned cycles (·M· model).

- Rapid movement (G0) of the longitudinal axis up to 1 mm off the part surface.

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



- Penetration. The longitudinal axis penetrates into the part the distance indicated by "B" and at the feedrate indicated by "V".
 - Milling of the pocket surface at work feedrate in the passes defined by "C" up to a distance "L" (finishing pass) from the pocket wall. It is carried out in the direction indicated by parameter "J".
 - Finishing milling, "L" amount, at the work feedrate defined by "H".
- In order to obtain good part finish when machining the pocket walls, the finishing passes are carried out with tangential entry and exit.
- Rapid withdrawal (G0) to the center of the pocket, 1 mm off the machined surface.



- New milling surfaces until reaching the total depth of the pocket.
 - Penetration, at the feedrate indicated in "F" up to a distance "B" from the previous surface.
 - Milling of the new surface following the steps indicated in points 5, 6 and 7.
- Withdrawal to the starting plane (Z_i) if function G98 is active or to the reference plane (Z) if function G99 is active.

1.

MILLING CANNED CYCLES.

G87. Rectangular pocket canned cycle.



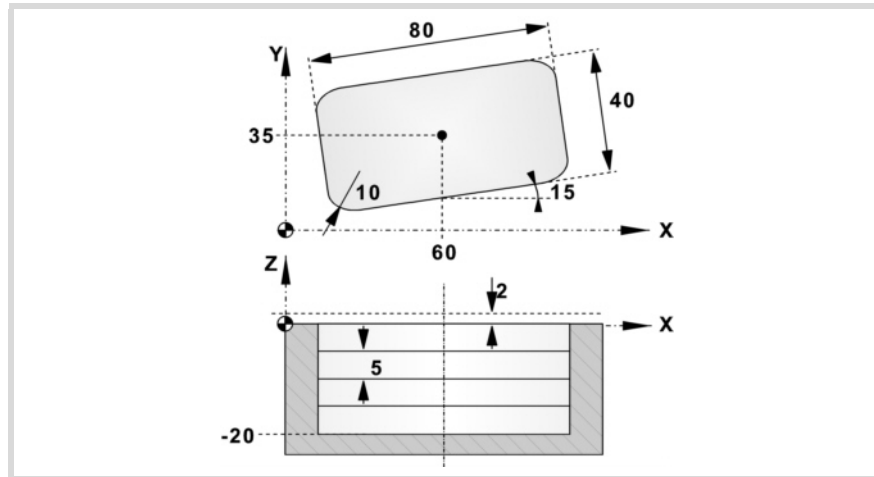
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

1.8.1 Programming example

To machine a 80x40 pocket centered at (X60, Y35) and rotated 15°. The pocket surface is at Z0 and it is to be emptied down to Z-20. The reference plane is at Z2.



```
G90 G0 X60 Y35
G87 Z2 I-20 D2 A15 J40 K20 .....
```

The pocket corners are to be rounded with a 10 mm radius.

```
G87 Z2 I-20 D2 A15 J40 K20 M1 Q10 .....
```

The penetrating pass is 5 mm and it is carried out at a feedrate of 50 mm/min.

```
G87 Z2 I-20 D2 A15 J40 K20 M1 Q10 B5 .....
```

The milling is carried out with a 5 mm wide roughing pass and at a feedrate of 800 mm/min. Since the milling feedrate must be selected before the execution of the cycle, it is defined in the previous block.

```
G90 G0 X60 Y35 F800
G87 Z2 I-20 D2 A15 J40 K20 M1 Q10 B5 C5 .....
```

It will leave a finishing stock of 1 mm that will be machined at a feedrate of 300 mm/min.

```
G87 Z2 I-20 D2 A15 J40 K20 M1 Q10 B5 C5 L1 H300 V50
```

We now show how to execute a pocket and repeated in several positions (X200 Y135) and (X350 Y235).

Absolute programming:

```
T7 D1 M6
G0 G90 X0 Y0 Z25 S1000 M3 M8 M41 F800
N10 G99 X60 Y35
G87 Z2 I-20 D2 A15 J40 K20 M1 Q10 B5 C5 L1 H300 V50
N20 X200 Y135
N30 G98 X350 Y235
M30
```

Incremental programming:

```
T7 D1 M6
G0 G90 X0 Y0 Z25 S1000 M3 M8 M41 F800
N10 G99 G91 X60 Y35
G87 Z-23 I-22 D2 A15 J40 K20 M1 Q10 B5 C5 L1 H300 V50
N20 X140 Y100
N30 G98 X150 Y100
M30
```

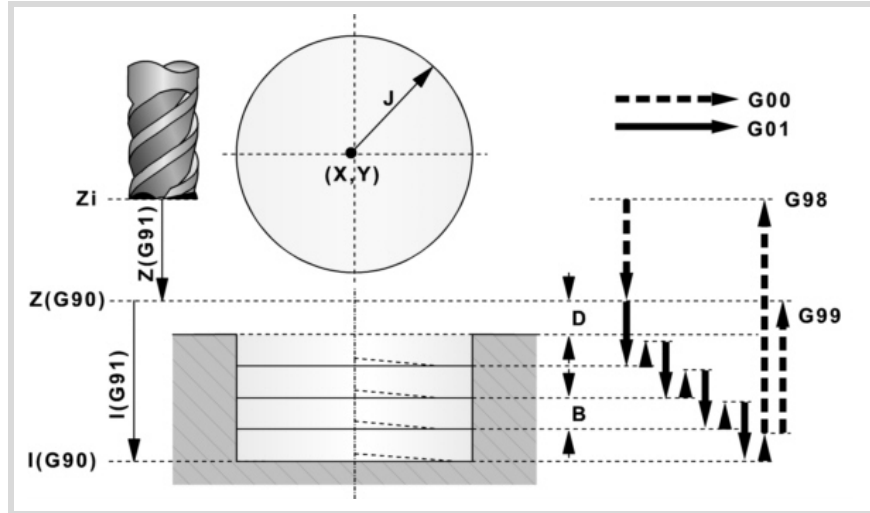
1.

MILLING CANNED CYCLES.
G87. Rectangular pocket canned cycle.

1.9 G88. Circular pocket canned cycle

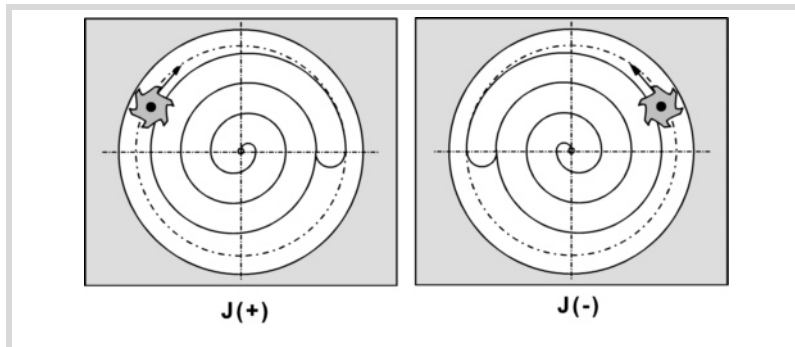
Programming format in Cartesian coordinates:

G88 Z I D J B C L H V



Parameter definition:

- Z Reference plane.
In G90, coordinate referred to part zero.
In G91, coordinate referred to starting plane (Zi).
If not programmed, it assumes as reference plane the current position of the tool (Z=Zi).
- I Pocket depth.
In G90, coordinate referred to part zero.
In G91, coordinate referred to reference plane (Z).
- D Distance between the reference plane and the part surface. If not programmed, it assumes a value of 1 mm.
- J Pocket radius.
The sign indicates the pocket machining direction:
(J+) clockwise, (J-) counterclockwise.



- B Depth of pass.
If programmed with a positive sign (B+), 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 (B-), the pocket is machined with the given pass (step) except the last pass that machines the rest.

1.

MILLING CANNED CYCLES.
G88. Circular pocket canned cycle



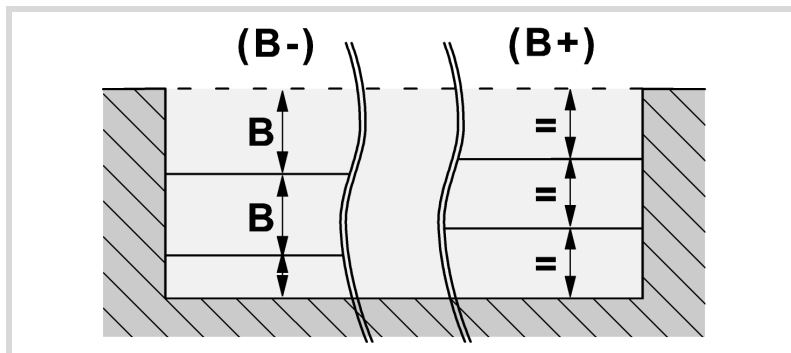
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

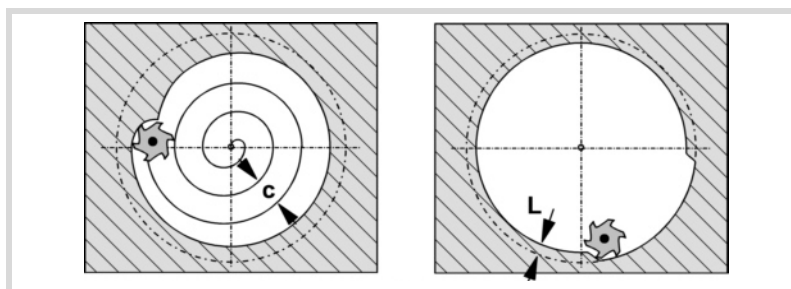
REF: 2102

1.

MILLING CANNED CYCLES.
G88. Circular pocket canned cycle



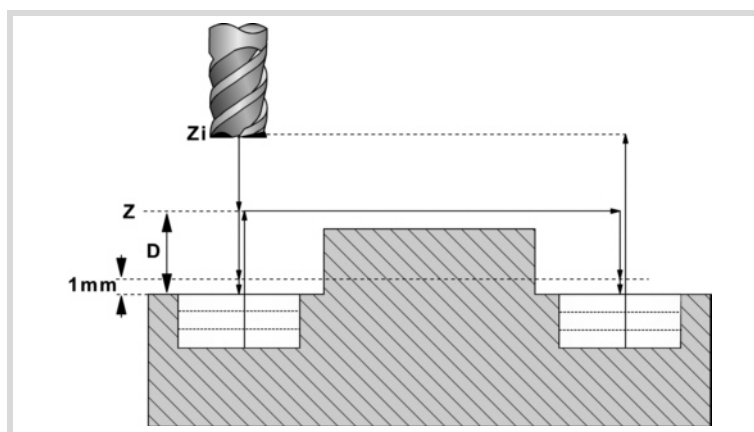
- C Milling pass or width.
If not programmed or programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.
If it is the same as parameter "J" (pocket radius), it only runs the finishing pass.
If programmed with a value greater than the tool diameter, the CNC issues the relevant error message.



- L Finishing pass.
If not programmed or programmed with a 0 value, it does not run the finishing pass.
- H Feedrate for the finishing pass. If not programmed or programmed with a 0 value, it is carried out at the roughing feedrate.
- V Tool penetrating feedrate. If not programmed or programmed with a 0 value, it is carried out at 50% of the feedrate in the plane.

Basic operation:

- 1 If the spindle was previously running, it maintains the turning direction. If it is stopped, it starts clockwise (M03).
- 2 Rapid movement (G0) of the longitudinal axis from the starting plane (Zi) to the reference plane (Z).
- 3 Rapid movement (G0) of the longitudinal axis up to 1 mm off the part surface.
The movement permits, as in the case of the figure, a fast approach to the machining surface when the safety coordinate (Z) is far away from the surface.

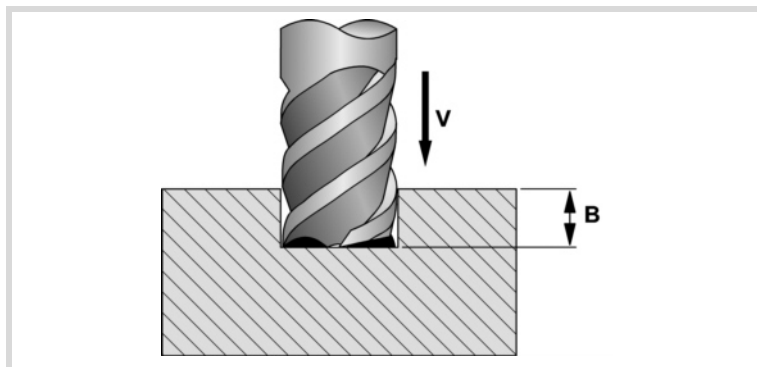


FAGOR 
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

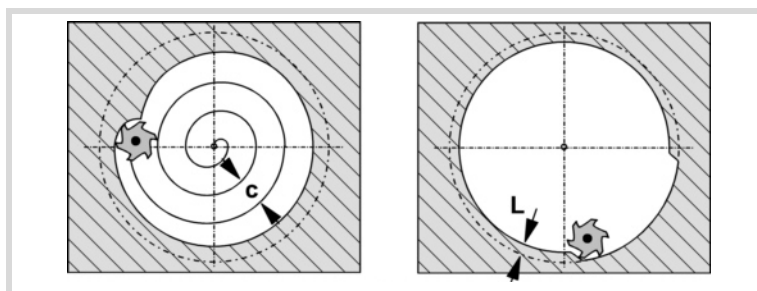
REF: 2102

- 4 Penetration. The longitudinal axis penetrates into the part the distance indicated by "B" and at the feedrate indicated by "V".



- 5 Milling of the pocket surface at work feedrate in the passes defined by "C" up to a distance "L" (finishing pass) from the pocket wall.

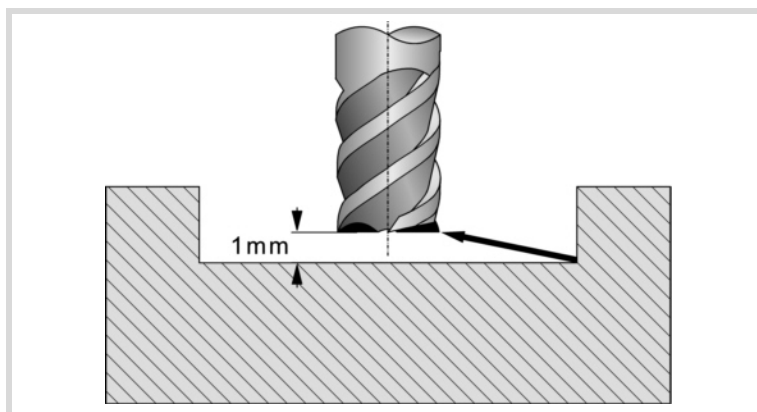
It is carried out in the direction indicated by parameter "J".



- 6 Finishing milling, "L" amount, at the work feedrate defined by "H".

In order to obtain good part finish when machining the pocket walls, the finishing passes are carried out with tangential entry and exit.

- 7 Rapid withdrawal (G0) to the center of the pocket, 1 mm off the machined surface.



- 8 New milling surfaces until reaching the total depth of the pocket.
- Penetration, at the feedrate indicated in "F" up to a distance "B" from the previous surface.
 - Milling of the new surface following the steps indicated in points 5, 6 and 7.
- 9 Withdrawal to the starting plane (Zi) if function G98 is active or to the reference plane (Z) if function G99 is active.

1.

MILLING CANNED CYCLES.
G88. Circular pocket canned cycle

FAGOR 

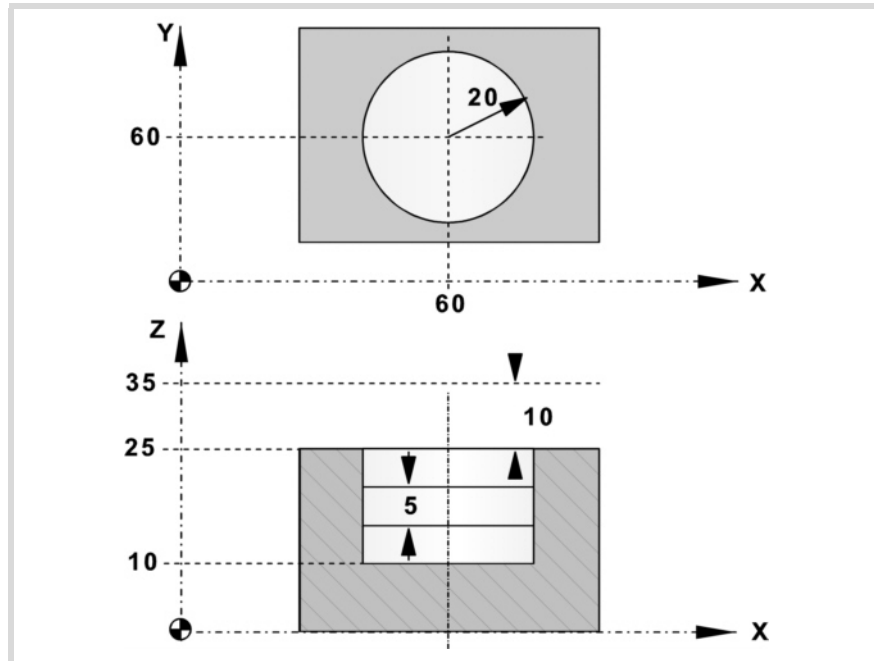
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

1.9.1 Programming example

To machine a 20 mm radius pocket centered in (X60, Y60). The pocket surface is at Z25 and it is to be emptied down to Z10. The reference plane is at Z35.



```
G90 G0 X60 Y60
G88 Z35 I10 D10 J20 .....
```

The penetrating pass is 5 mm and it is carried out at a feedrate of 50 mm/min.

```
G88 Z35 I10 D10 J20 B5 ..... V50
```

The milling is carried out with a 5 mm wide roughing pass and at a feedrate of 800 mm/min. Since the milling feedrate must be selected before the execution of the cycle, it is defined in the previous block.

```
G90 G0 X60 Y60 F800
G88 Z35 I10 D10 J20 B5 C5 ..... V50
```

It will leave a finishing stock of 1 mm that will be machined at a feedrate of 300 mm/min.

```
G88 Z35 I10 D10 J20 B5 C5 L1 H300 V50
```

We now show how to execute a pocket and repeated in several positions (X200 Y135) and (X350 Y235).

Absolute programming:

```
T8 D1 M6
G0 G90 X0 Y0 Z45 S1000 M3 M8 M41 F800
N10 G99 X60 Y60
G88 Z35 I10 D10 J20 B5 C5 L1 H300 V50
N20 X200 Y135
N30 G98 X350 Y235
M30
```

Incremental programming:

```
T8 D1 M6
G0 G90 X0 Y0 Z45 S1000 M3 M8 M41 F800
N10 G99 G91 X60 Y60
G88 Z-10 I-25 D10 J20 B5 C5 L1 H300 V50
N20 X140 Y75
N30 G98 X150 Y100
M30
```

1.

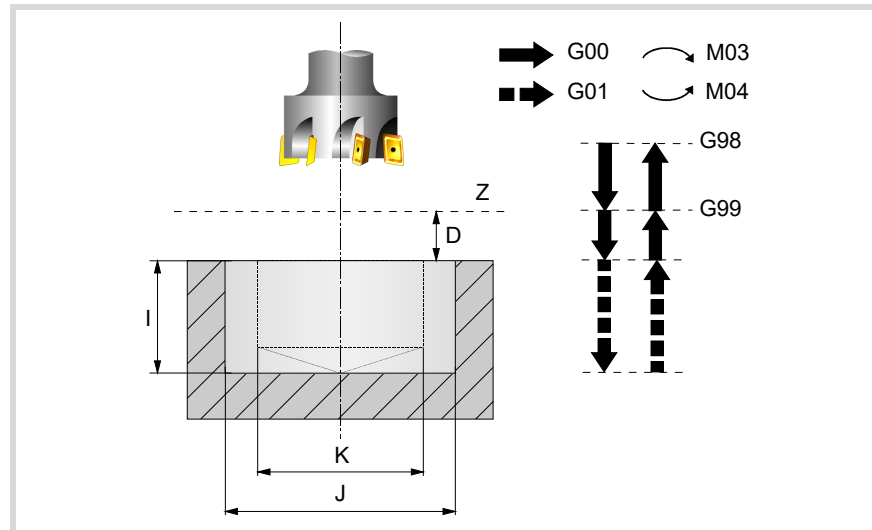
MILLING CANNED CYCLES.
G88. Circular pocket canned cycle

1.10 G210. Bore milling canned cycle

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.

Programming format in Cartesian coordinates:

G210 Z D I J K B



Parameter definition:

- Z** Reference plane.
Defines the reference plane coordinate. It may be programmed either in absolute or incremental coordinates, in which case it will be referred to the starting plane. If not programmed, it assumes as reference plane the current position of the tool.
- D** Safety distance.
Defines the distance between the reference plane and the surface of the part where the milling is to be done. If not programmed, it assumes 0.
- I** Machining depth.
Defines the machining depth. It may be programmed either in absolute or incremental coordinates, in which case it will be referred to the reference plane.
- J** Hole diameter.
Defines the nominal diameter of the hole. The sign indicates the direction of the helical path associated with the machining of the hole (positive if clockwise and negative if counterclockwise).
If not programmed or programmed with a value smaller than the diameter of the active tool, the CNC issues the relevant error message.
- K** 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** Penetration step.
It defines the penetration step when machining the hole.
- With a positive sign, it will mill the bottom of the hole.
 - With a negative sign, it will not mill the bottom of the hole.

1.

MILLING CANNED CYCLES.
G210. Bore milling canned cycle

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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 parameter J, 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).

1.

MILLING CANNED CYCLES.
G210. Bore milling canned cycle



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

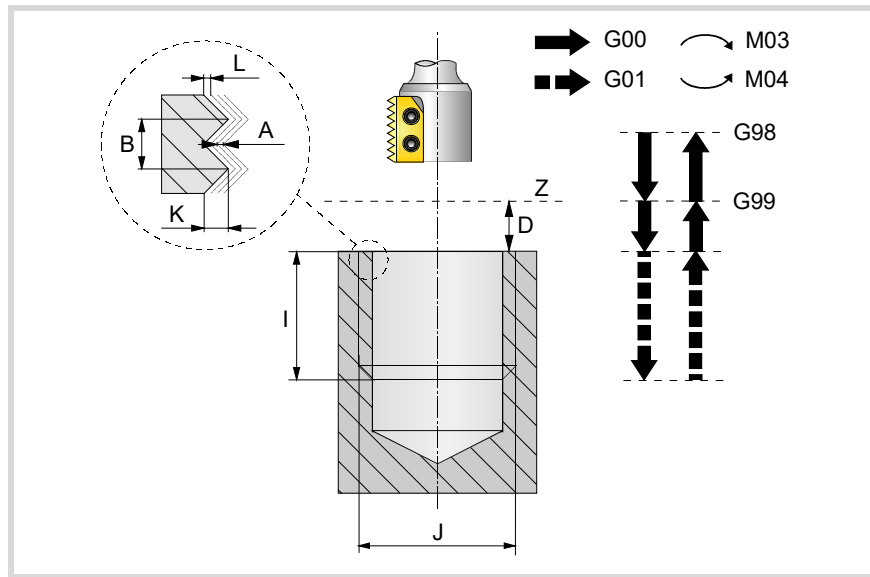
REF: 2102

1.11 G211. Inside thread milling cycle

This cycle may be used to make an inside thread through a helical movement of the tool.

Working in Cartesian coordinates, the basic structure of the block is as follows:

G211 Z D I J K B C L A E Q



Parameter definition:

- Z** Reference plane.
Defines the reference plane coordinate. It may be programmed either in absolute or incremental coordinates, in which case it will be referred to the starting plane.
- D** Safety distance.
Defines the distance between the reference plane and the surface of the part where the milling is to be done. If not programmed, it assumes 0.
- I** Machining depth.
Defines the threading depth. It may be programmed either in absolute or incremental coordinates, in which case it will be referred to the reference plane.
- J** Thread diameter.
Defines the nominal diameter of the thread. The sign indicates the machining direction of the thread (positive if clockwise and negative if counterclockwise).
- K** Thread depth.
It defines the distance between the crest and the root of the thread.
- B** Thread pitch.
Defines the thread pitch.
- With a positive sign, the direction of the thread pitch is from the surface of the part to the bottom.
 - With a negative sign, the direction of the thread pitch is from the bottom to the surface of the part.
- C** Type of threading
Defines the type of tapping to be carried out. This parameter depends on the type of tool being used.
- When programming C=0, the threading will be done in a single pass.
 - When programming C=1, it will make one thread per each pass (single-edge cutter).
 - When programming C=n (where n is the number of cutting edges of the cutter), it will make n threads per each pass.
- If not programmed, a value of C=1 is assumed.

1.

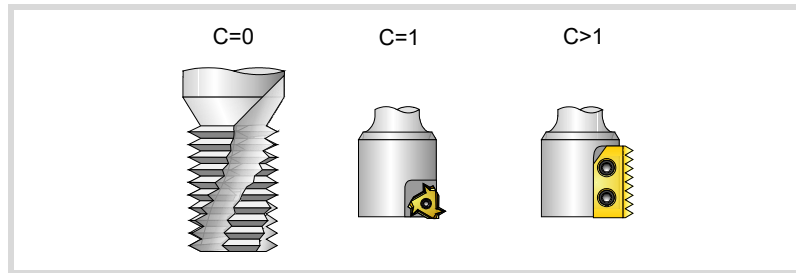
MILLING CANNED CYCLES.
G211. Inside thread milling cycle

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102



1.

MILLING CANNED CYCLES.
G211. Inside thread milling cycle

- L** Finishing stock.
It defines the finishing stock at the bottom of the thread. If not programmed, a value of 0 is assumed.
- A** Maximum penetration step.
Defines the maximum penetrating pass of the thread. If not programmed or programmed with a 0 value, it will run a single pass up to the finishing stock.
- E** Approach distance.
Approach distance to the thread entry. If not programmed, it will enter the thread from the center of the hole.
- Q** 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. If not programmed, a value of 0 is assumed.

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 (it only makes this movement if parameter E has been programmed).
- 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 value of parameter C.

- C=0 Helical movement, in the direction indicated in parameter J, to the bottom of the thread (the movement will only be one revolution). Then, helical thread exiting movement, tangent to the previous helical path. If parameter E has not been programmed, the exit point will correspond with the coordinates of the hole center.

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.

- C=1 Helical movement, with the pitch and direction given in parameter J, to the bottom of the thread. Then, helical thread exiting movement, tangent to the previous helical path. If parameter E has not been programmed, the exit point will correspond with the coordinates of the hole center.

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.

- C=n Threading loop until reaching the bottom of the thread.
 - Helical movement, with the pitch and direction given in parameter J, (the movement will be one revolution).
 - Helical thread exiting movement, tangent to the previous helical path. If parameter E has not been programmed, the exit point will correspond with the coordinates of the hole center.
 - Rapid movement to the thread entry point of the next threading path. Rapid movement to the Z coordinate of the thread entry point of the next threading path.

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.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

Canned cycles (·M· model).

- 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).

1.

MILLING CANNED CYCLES.

G211. Inside thread milling cycle

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

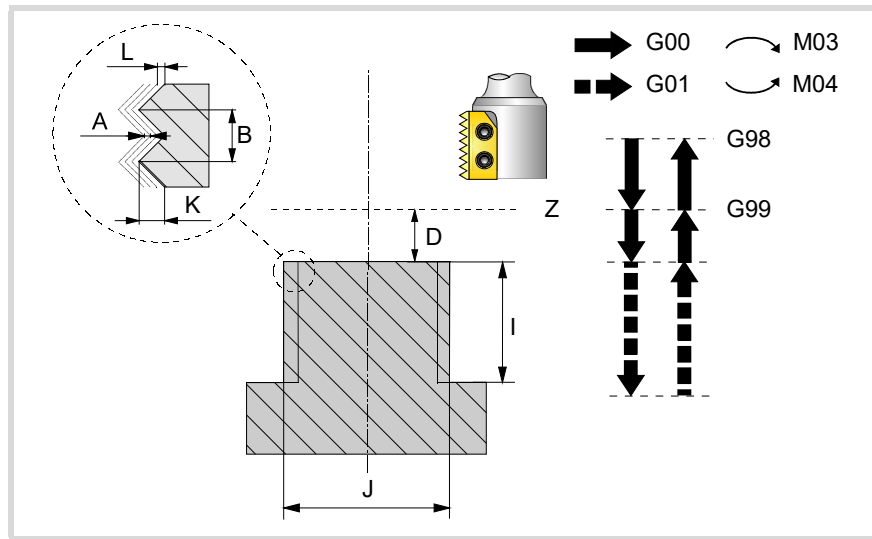
REF: 2102

1.12 G212. Outside thread milling cycle

This cycle may be used to make an outside thread through a helical movement of the tool.

Working in Cartesian coordinates, the basic structure of the block is as follows:

G212 Z D I J K B C L A E Q



Parameter definition:

- Z Reference plane.
Defines the reference plane coordinate. It may be programmed either in absolute or incremental coordinates, in which case it will be referred to the starting plane.
If not programmed, it assumes as reference plane the current position of the tool.
- D Safety distance.
Defines the distance between the reference plane and the surface of the part where the milling is to be done. If not programmed, it assumes 0.
- I Machining depth.
Defines the threading depth. It may be programmed either in absolute or incremental coordinates, in which case it will be referred to the reference plane.
- J Thread diameter.
Defines the nominal diameter of the thread. The sign indicates the machining direction of the thread (positive if clockwise and negative if counterclockwise).
- K Thread depth.
It defines the distance between the crest and the root of the thread.
- B Thread pitch.
Defines the thread pitch.
 - With a positive sign, the direction of the thread pitch is from the surface of the part to the bottom.
 - With a negative sign, the direction of the thread pitch is from the bottom to the surface of the part.
- C Type of tapping.
Defines the type of tapping to be carried out. This parameter depends on the type of tool being used.
 - When programming C=0, the threading will be done in a single pass.
 - When programming C=1, it will make one thread per each pass (single-edge cutter).
 - When programming C=n (where n is the number of cutting edges of the cutter), it will make n threads per each pass.
If not programmed, a value of C=1 is assumed.

1.

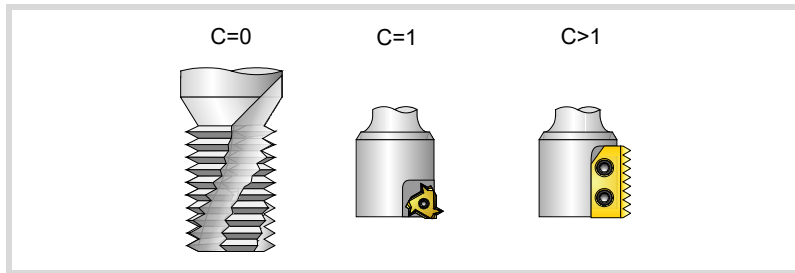
MILLING CANNED CYCLES.
G212. Outside thread milling cycle



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102



- L** Finishing stock.
It defines the finishing stock at the bottom of the thread. If not programmed, a value of 0 is assumed.
- A** Maximum penetration step.
Defines the maximum penetrating pass of the thread. If not programmed or programmed with a 0 value, it will run a single pass up to the finishing stock.
- E** Approach distance.
Approach distance to the thread entry.
- Q** 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. If not programmed, a value of 0 is assumed.

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 (it only makes this movement if parameter E has been programmed).
- 4 Rapid movement to the thread entry point coordinate along the longitudinal axis.
- 5 Rapid movement to the thread entry point (movement interpolated in 3 axes).
- 6 Thread entry with a helical movement tangent to the first helical threading path.
- 7 Making the thread according to the value of parameter C.
 - C=0 Helical movement, in the direction indicated in parameter J, to the bottom of the thread (the movement will only be one revolution). Then, 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.
 - C=1 Helical movement, with the pitch and direction given in parameter J, to the bottom of the thread. Then, 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.
 - C=n Threading loop until reaching the bottom of the thread.
 - Helical movement, with the pitch and direction given in parameter J, (the movement will be one revolution).
 - Helical thread exiting movement, tangent to the previous helical path.
 - Rapid movement to the Z coordinate of the thread entry point of the next threading path.
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 Rapid movement to the reference plane (G99).
- 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).

1.

MILLING CANNED CYCLES.

G212. Outside thread milling cycle

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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

1.

MILLING CANNED CYCLES.

G212. Outside thread milling cycle



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

The programmer selects the type of machining that could be any canned cycle.

Programming

The machining paths are defined by the following functions:

| | |
|------|--|
| G160 | Multiple machining in a straight line. |
| G161 | Multiple machining in a parallelogram pattern. |
| G162 | Multiple machining in a grid pattern. |
| G163 | Multiple machining in a full circle. |
| G164 | Multiple machining in an arc. |
| G165 | Machining programmed with an arc-chord. |

These functions may be executed in any work plane and must be defined every time they are used because they are not modal.

The machining operation to be repeated **MUST BE** active. In other words, these functions will only make sense if they are under the influence (affected by) a canned cycle.

Follow these steps to carry out a multiple machining operation:

- 1 Move the tool to the first point where the multiple machining will take place.
- 2 Define the canned cycle to be repeated at all the points.
- 3 Define the multiple machining operation to be carried out.

Considerations

All the machining operations programmed with these functions are carried out under the same working conditions (T, D, F, S) that were selected when the canned cycle was defined.

Once the programmed multiple machining has been executed, the program will restore the history that it had before starting the machining operation, the canned cycle will even remain active. F now being the feedrate for the feedrate programmed for the canned cycle.

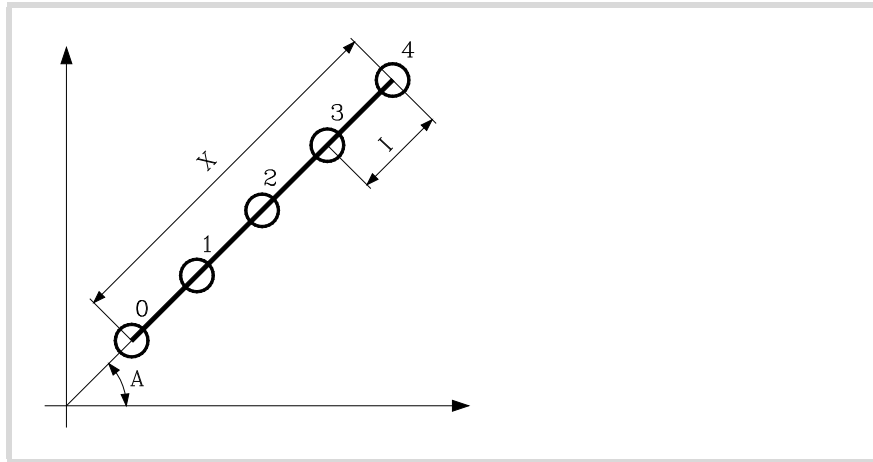
Likewise, the tool will be positioned at the last point where the programmed machining operation was carried out.

A detailed description is given of the multiple machining operations assuming in all of them that the work plane is formed by the X and Y axis.

2.1 G160. Multiple machining in straight line

The programming format for this cycle is. When defining the machining operation, only use two parameters of the group "X", "I" and "K".

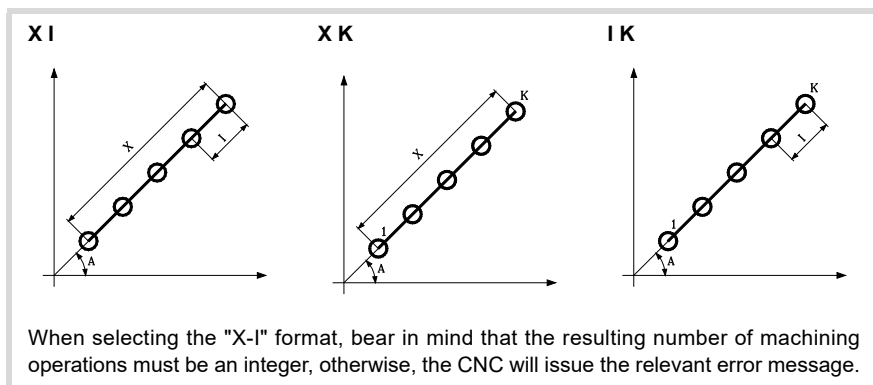
G160 A X I K P Q R S T U V



- A Angle, in degrees of the machining path with respect to the abscissa axis.
If not programmed, a value of $A = 0$ is assumed.

When defining the machining operation, only two of parameters "X", "I" and "K" are required.

- X Length of the machining path.
I Step between machining operations.
K Total number of machining operations in the section, including that of the machining definition point.



- P,Q,R,S,T,U,V These parameters are optional and are used to indicate at which points or between which points of the ones programmed the machining operation will NOT be carried out. If these parameters are not programmed, the CNC executes the machining operation at all the points of the programmed path.

Thus, programming "P7" means that no machining operation takes place at point 7. Programming "Q10.013" means that no machining takes place at points 10, 11, 12 and 13.

When defining a set of points (Q10.013), bear in mind that the last point must be defined with three digits because, for example, "Q10.13" is the same as programming "Q10.130".

The programming order for these parameters is "P" "Q" "R" "S" "T" "U" "V" and the numbering sequence for the points assigned to them must also be respected; In other words, the numbering sequence of the points assigned to "Q" must be greater than the one of those assigned to "P" and smaller than the one for those assigned to "R".

Example of proper programming: P5.006 Q12.015 R20.022

Example of wrong programming: P5.006 Q20.022 R12.015

2.

MULTIPLE MACHINING
G160. Multiple machining in straight line

FAGOR 
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

Basic operation

Multiple machining is executed as follows:

- 1 The multiple machining calculates the next programmed point to machine.
- 2 Rapid movement (G00) to that point.
- 3 The multiple machining will execute the selected canned cycle after the movement.
- 4 The CNC will repeat steps 1-2-3 until completing the programmed multiple machining operation.

After completing the multiple machining, the tool will remain positioned at the last point of the programmed path where the machining operation took place.

2.

MULTIPLE MACHINING

G160. Multiple machining in straight line

FAGOR 

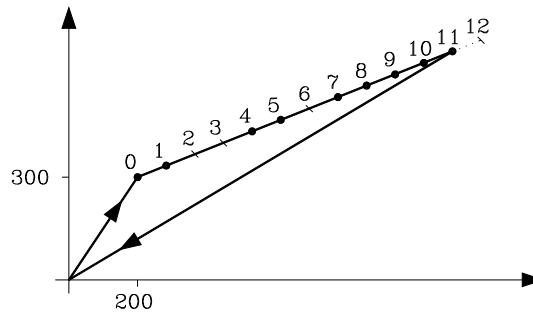
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

2.1.1 Programming example

Programming example assuming that the work plane is formed by the X and Y axes, that the Z axis is the longitudinal axis and that the starting point is X0 Y0 Z0:



```
G00 G91 X200 Y300 F100 S500
G98 G81 Z-8 I-22
G160 A30 X1200 I100 P2.003 Q6 R12
G80
G90 X0 Y0
M30
```

Multiple machining may also be defined as follows:

```
G160 A30 X1200 K13 P2.003 Q6 R12
G160 A30 I100 K13 P2.003 Q6 R12
```

2.

MULTIPLE MACHINING
G160. Multiple machining in straight line

FAGOR 

FAGOR AUTOMATION

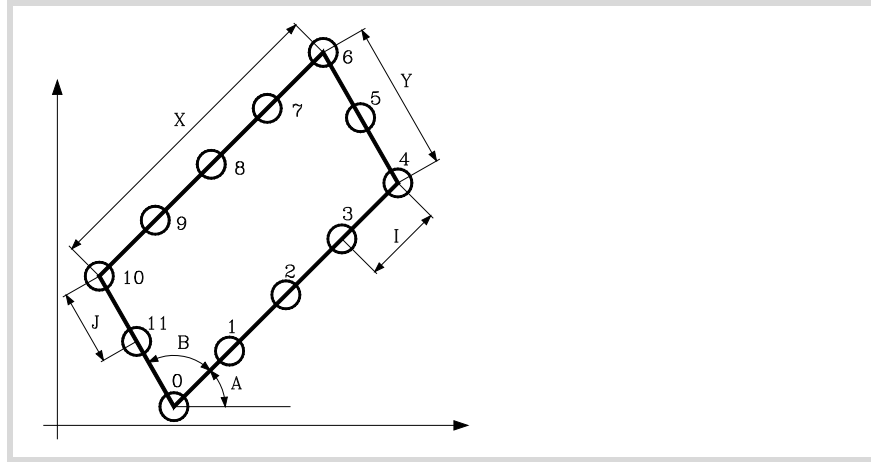
CNC 8058
CNC 8060
CNC 8065

REF: 2102

2.2 G161. Multiple machining in rectangular pattern

The programming format for this cycle is. When defining the machining operation, only use two parameters of the group "X", "I" and "K" and two of the group "Y", "J", "D".

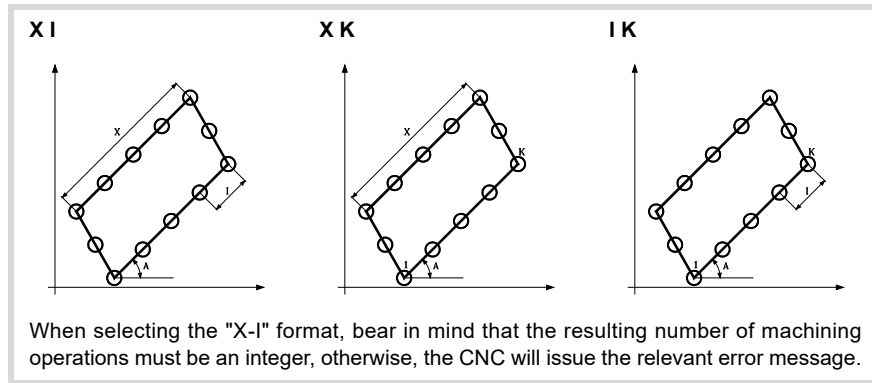
G161 A B X I K Y J D P Q R S T U V



- A Angle, in degrees of the machining path with respect to the abscissa axis.
If not programmed, a value of A = 0 is assumed.
- B Angle between both machining paths.
If not programmed, a value of B = 90 is assumed.

When defining the length of the parallelogram, only two of parameters "X", "I" and "K" are required.

- X Length of the parallelogram.
- I Step between machining operations along the path.
- K Total number of machining operations along the path, including that of the machining definition point.



When defining the width of the parallelogram, only two of parameters "Y", "J" and "D" are required.

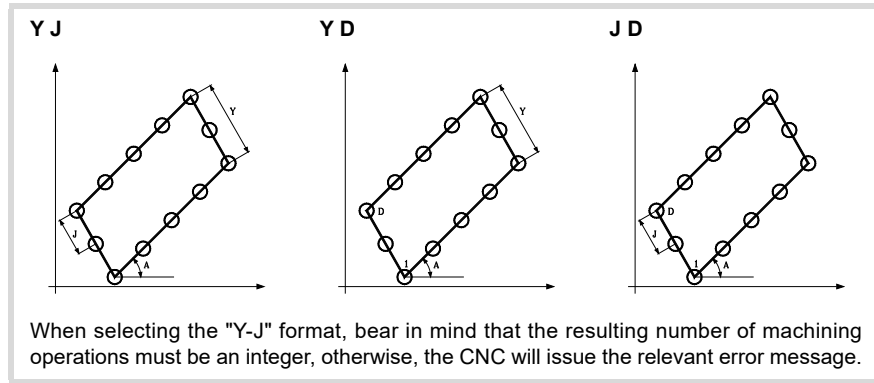
- Y Width of the parallelogram.
- J Step between machining operations along the path.
- D Total number of machining operations along the path, including that of the machining definition point.

2.

MULTIPLE MACHINING
G161. Multiple machining in rectangular pattern

2.

MULTIPLE MACHINING
 G161. Multiple machining in rectangular pattern



P,Q,R,S,T,U,V These parameters are optional and are used to indicate at which points or between which points of the ones programmed the machining operation will NOT be carried out. If these parameters are not programmed, the CNC executes the machining operation at all the points of the programmed path.

Thus, programming "P7" means that no machining operation takes place at point 7. Programming "Q10.013" means that no machining takes place at points 10, 11, 12 and 13.

When defining a set of points (Q10.013), bear in mind that the last point must be defined with three digits because, for example, "Q10.13" is the same as programming "Q10.130".

The programming order for these parameters is "P" "Q" "R" "S" "T" "U" "V" and the numbering sequence for the points assigned to them must also be respected; In other words, the numbering sequence of the points assigned to "Q" must be greater than the one of those assigned to "P" and smaller than the one for those assigned to "R".

Example of proper programming: P5.006 Q12.015 R20.022

Example of wrong programming: P5.006 Q20.022 R12.015

Basic operation

Multiple machining is executed as follows:

- 1 The multiple machining calculates the next programmed point to machine.
- 2 Rapid movement (G00) to that point.
- 3 The multiple machining will execute the selected canned cycle after the movement.
- 4 The CNC will repeat steps 1-2-3 until completing the programmed multiple machining operation.

After completing the multiple machining, the tool will remain positioned at the last point of the programmed path where the machining operation took place.



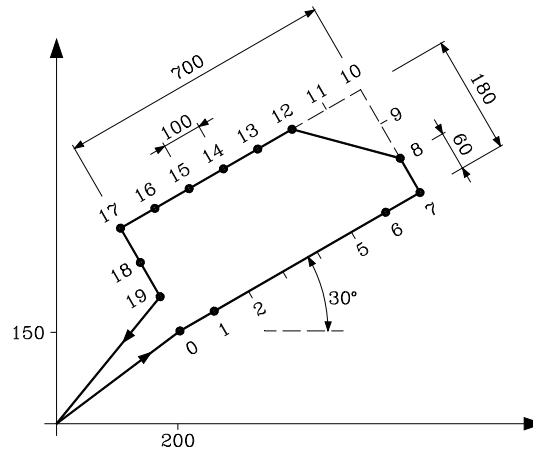
FAGOR AUTOMATION

CNC 8058
 CNC 8060
 CNC 8065

REF: 2102

2.2.1 Programming example

Programming example assuming that the work plane is formed by the X and Y axes, that the Z axis is the longitudinal axis and that the starting point is X0 Y0 Z0:



```
G00 G91 X100 Y150 F100 S500
G98 G81 Z-8 I-22
G161 A30 X700 I100 Y180 J60 P2.005 Q9.011
G80
G90 X0 Y0
M30
```

Multiple machining may also be defined as follows:
G161 A30 X700 K8 J60 D4 P2.005 Q9.011
G161 A30 I100 K8 Y180 D4 P2.005 Q9.011

2.

MULTIPLE MACHINING
G161. Multiple machining in rectangular pattern

FAGOR 

FAGOR AUTOMATION

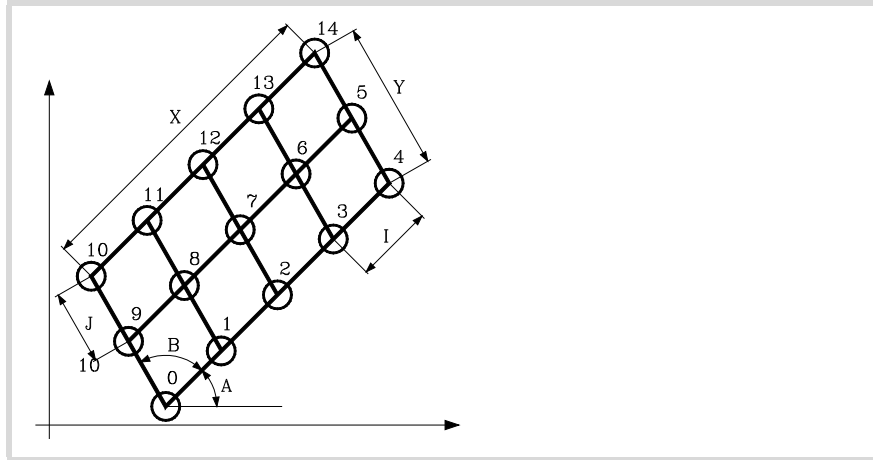
CNC 8058
CNC 8060
CNC 8065

REF: 2102

2.3 G162. Multiple machining in grid pattern

The programming format for this cycle is. When defining the machining operation, only use two parameters of the group "X", "I" and "K" and two of the group "Y", "J", "D".

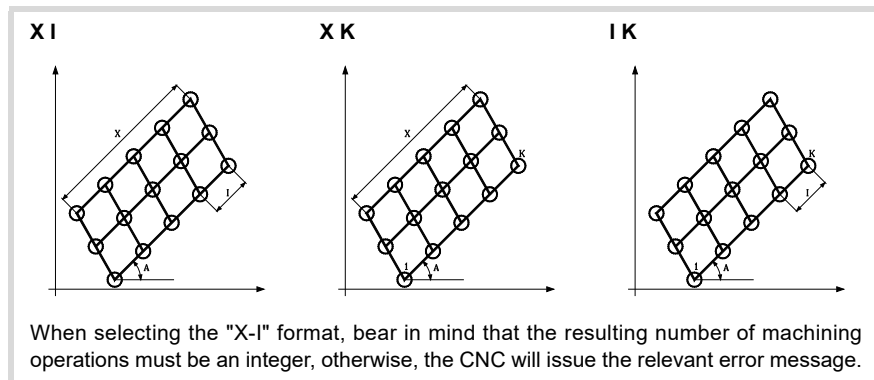
G162 A B X I K Y J D P Q R S T U V



- A Angle, in degrees of the machining path with respect to the abscissa axis.
If not programmed, a value of $A = 0$ is assumed.
- B Angle between both machining paths.
If not programmed, a value of $B = 90$ is assumed.

When defining the length of the grid only two of parameters "X", "I" and "K" are required.

- X Length of the grid.
- I Step between machining operations along the path.
- K Total number of machining operations along the path, including that of the machining definition point.



When defining the width of the grid, only two of parameters "Y", "J" and "D" are required.

- Y Width of the grid.
- J Step between machining operations along the path.
- D Total number of machining operations along the path, including that of the machining definition point.

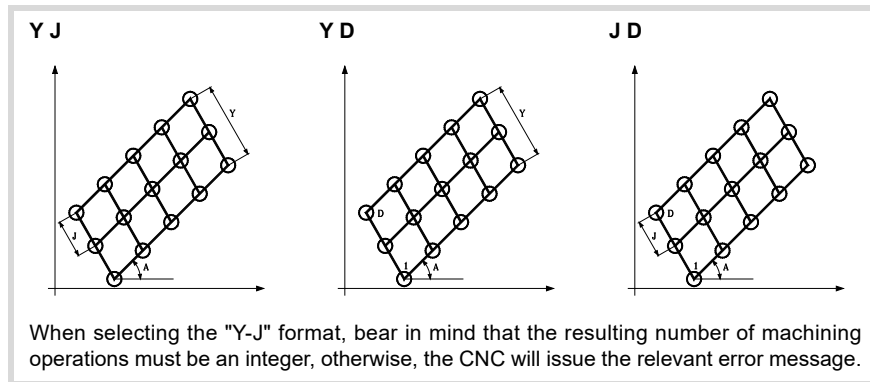
2.

MULTIPLE MACHINING
G162. Multiple machining in grid pattern

FAGOR 
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102



P,Q,R,S,T,U,V These parameters are optional and are used to indicate at which points or between which points of the ones programmed the machining operation will NOT be carried out. If these parameters are not programmed, the CNC executes the machining operation at all the points of the programmed path.

Thus, programming "P7" means that no machining operation takes place at point 7. Programming "Q10.013" means that no machining takes place at points 10, 11, 12 and 13.

When defining a set of points (Q10.013), bear in mind that the last point must be defined with three digits because, for example, "Q10.13" is the same as programming "Q10.130".

The programming order for these parameters is "P" "Q" "R" "S" "T" "U" "V" and the numbering sequence for the points assigned to them must also be respected; In other words, the numbering sequence of the points assigned to "Q" must be greater than the one of those assigned to "P" and smaller than the one for those assigned to "R".

Example of proper programming: P5.006 Q12.015 R20.022
 Example of wrong programming: P5.006 Q20.022 R12.015

Basic operation

Multiple machining is executed as follows:

- 1 The multiple machining calculates the next programmed point to machine.
- 2 Rapid movement (G00) to that point.
- 3 The multiple machining will execute the selected canned cycle after the movement.
- 4 The CNC will repeat steps 1-2-3 until completing the programmed multiple machining operation.

After completing the multiple machining, the tool will remain positioned at the last point of the programmed path where the machining operation took place.

2.

MULTIPLE MACHINING
 G162. Multiple machining in grid pattern



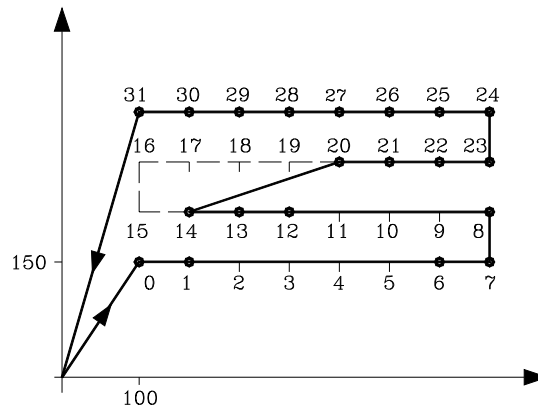
FAGOR AUTOMATION

CNC 8058
 CNC 8060
 CNC 8065

REF: 2102

2.3.1 Programming example

Programming example assuming that the work plane is formed by the X and Y axes, that the Z axis is the longitudinal axis and that the starting point is X0 Y0 Z0:



```
G00 G91 X100 Y150 F100 S500
G98 G81 Z-8 I-22
G162 X700 I100 Y180 J60 P2.005 Q9.011 R15.019
G80
G90 X0 Y0
M30
```

Multiple machining may also be defined as follows:

```
G162 X700 K8 J60 D4 P2.005 Q9.011 R15.019
G162 I100 K8 Y180 D4 P2.005 Q9.011 R15.019
```

2.

MULTIPLE MACHINING
G162. Multiple machining in grid pattern

FAGOR 
FAGOR AUTOMATION

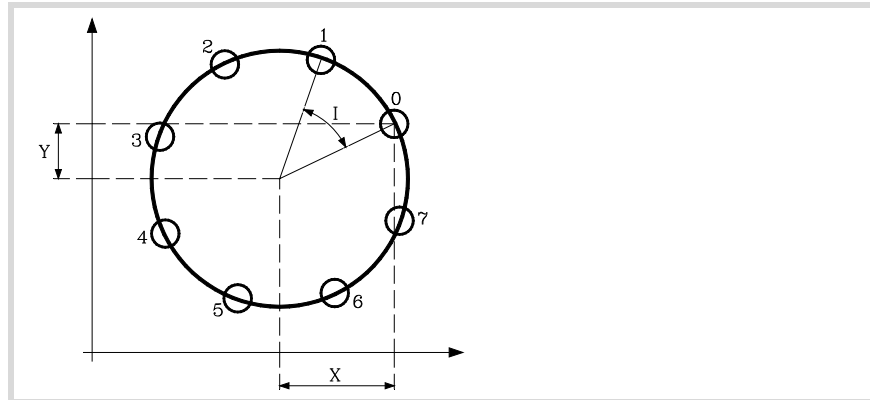
CNC 8058
CNC 8060
CNC 8065

REF: 2102

2.4 G163. Multiple machining in a full circle

The programming format for this cycle is. When defining the machining operation, only use one parameter of the group "I" and "K".

G163 X Y I K C F P Q R S T U V



Parameters "X" and "Y" define the center of the circle, same as "I" and "J" in circular interpolations (G02, G03).

X Distance from the starting point to the center along the abscissa axis.

Y Distance from the starting point to the center along the ordinate axis.

When defining the machining operation, only one of parameters "I" and "K" is required. If the angular step is programmed, bear in mind that the total angular movement must be 360°, otherwise, the CNC will issue the relevant error message.

I Angular step between machining operations.

When the movement between points is done in G00 or G01, the sign indicates the direction: "I+" counterclockwise and "I-" clockwise.

K Total number of machining operations including that of the machining definition point.

When the movement between points is done in G00 or G01, the machining operation is carried out counterclockwise.

C It indicates how it will move between the machining points. If not programmed, a value of C = 0 is assumed.

C=0 In rapid (G00).

C=1 Linear interpolation (G01).

C=2 In clockwise circular interpolation (G02).

C=3 In counterclockwise circular interpolation (G03).

F Feedrate for the movement between points. It will only be valid for "C" values other than zero.

P,Q,R,S,T,U,V These parameters are optional and are used to indicate at which points or between which points of the ones programmed the machining operation will NOT be carried out. If these parameters are not programmed, the CNC executes the machining operation at all the points of the programmed path.

Thus, programming "P7" means that no machining operation takes place at point 7. Programming "Q10.013" means that no machining takes place at points 10, 11, 12 and 13.

When defining a set of points (Q10.013), bear in mind that the last point must be defined with three digits because, for example, "Q10.13" is the same as programming "Q10.130".

The programming order for these parameters is "P" "Q" "R" "S" "T" "U" "V" and the numbering sequence for the points assigned to them must also be respected; In other words, the numbering sequence of the points assigned to "Q" must be greater than the one of those assigned to "P" and smaller than the one for those assigned to "R".

Example of proper programming: P5.006 Q12.015 R20.022

Example of wrong programming: P5.006 Q20.022 R12.015

2.

MULTIPLE MACHINING
G163. Multiple machining in a full circle

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

Basic operation

Multiple machining is executed as follows:

- 1 The multiple machining calculates the next programmed point to machine.
- 2 Movement to that point at the feedrate programmed with "C" (G00, G01, G02 or G03).
- 3 The multiple machining will execute the selected canned cycle after the movement.
- 4 The CNC will repeat steps 1-2-3 until completing the programmed multiple machining operation.

After completing the multiple machining, the tool will remain positioned at the last point of the programmed path where the machining operation took place.

2.

MULTIPLE MACHINING

G163. Multiple machining in a full circle



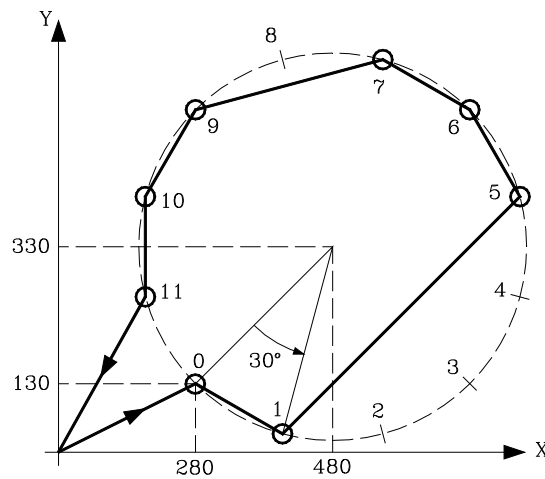
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

2.4.1 Programming example

Programming example assuming that the work plane is formed by the X and Y axes, that the Z axis is the longitudinal axis and that the starting point is X0 Y0 Z0:



```
G00 G91 X280 Y130 F100 S500
G98 G81 Z-8 I-22
G163 X200 Y200 I30 C1 F200 P2.004 Q8
G80
G90 X0 Y0
M30
```

Multiple machining may also be defined as follows:
G163 X200 Y200 K12 C1 F200 P2.004 Q8

2.

MULTIPLE MACHINING
G163. Multiple machining in a full circle

FAGOR 

FAGOR AUTOMATION

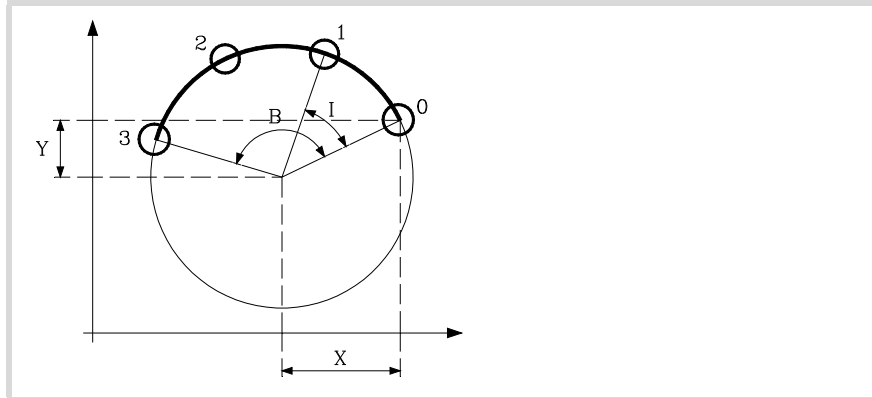
CNC 8058
CNC 8060
CNC 8065

REF: 2102

2.5 G164. Multiple machining in arc pattern

The programming format for this cycle is. When defining the machining operation, only use one parameter of the group "I" and "K".

G164 X Y B I K C F P Q R S T U V



Parameters "X" and "Y" define the center of the circle, same as "I" and "J" in circular interpolations (G02, G03).

- X Distance from the starting point to the center along the abscissa axis.
- Y Distance from the starting point to the center along the ordinate axis.
- B Angular distance in degrees of the machining path.

When defining the machining operation, only one of parameters "I" and "K" is required. If the angular step is programmed, bear in mind that the total angular movement must be the programmed angular distance "B", otherwise, the CNC will issue the relevant error message.

- I Angular step between machining operations.
When the movement between points is done in G00 or G01, the sign indicates the direction: "I+" counterclockwise and "I-" clockwise.
- K Total number of machining operations including that of the machining definition point.
When the movement between points is done in G00 or G01, the machining operation is carried out counterclockwise.
- C It indicates how it will move between the machining points. If not programmed, a value of C = 0 is assumed.
 - C=0 In rapid (G00).
 - C=1 Linear interpolation (G01).
 - C=2 In clockwise circular interpolation (G02).
 - C=3 In counterclockwise circular interpolation (G03).
- F Feedrate for the movement between points. It will only be valid for "C" values other than zero.
- P,Q,R,S,T,U,V These parameters are optional and are used to indicate at which points or between which points of the ones programmed the machining operation will NOT be carried out. If these parameters are not programmed, the CNC executes the machining operation at all the points of the programmed path.

Thus, programming "P7" means that no machining operation takes place at point 7. Programming "Q10.013" means that no machining takes place at points 10, 11, 12 and 13.

When defining a set of points (Q10.013), bear in mind that the last point must be defined with three digits because, for example, "Q10.13" is the same as programming "Q10.130".

The programming order for these parameters is "P" "Q" "R" "S" "T" "U" "V" and the numbering sequence for the points assigned to them must also be respected; In other words, the numbering sequence of the points assigned to "Q" must be greater than the one of those assigned to "P" and smaller than the one for those assigned to "R".

Example of proper programming: P5.006 Q12.015 R20.022

Example of wrong programming: P5.006 Q20.022 R12.015

2.

MULTIPLE MACHINING
G164. Multiple machining in arc pattern

FAGOR 
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

Basic operation

Multiple machining is executed as follows:

- 1 The multiple machining calculates the next programmed point to machine.
- 2 Movement to that point at the feedrate programmed with "C" (G00, G01, G02 or G03).
- 3 The multiple machining will execute the selected canned cycle after the movement.
- 4 The CNC will repeat steps 1-2-3 until completing the programmed multiple machining operation.

After completing the multiple machining, the tool will remain positioned at the last point of the programmed path where the machining operation took place.

2.

MULTIPLE MACHINING

G164. Multiple machining in arc pattern

FAGOR 

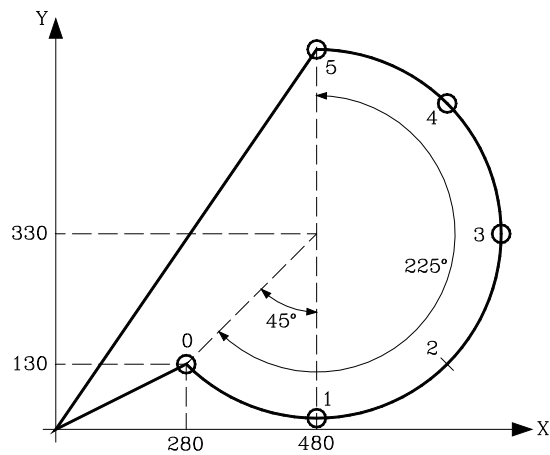
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

2.5.1 Programming example

Programming example assuming that the work plane is formed by the X and Y axes, that the Z axis is the longitudinal axis and that the starting point is X0 Y0 Z0:



```
G00 G91 X280 Y130 F100 S500
G98 G81 Z-8 I-22
G164 X200 Y200 B225 I45 C3 F200 P2
G80
G90 X0 Y0
M30
```

Multiple machining may also be defined as follows:
G164 X200 Y200 B225 K6 C3 F200 P2

2.

MULTIPLE MACHINING
G164. Multiple machining in arc pattern

FAGOR 
FAGOR AUTOMATION

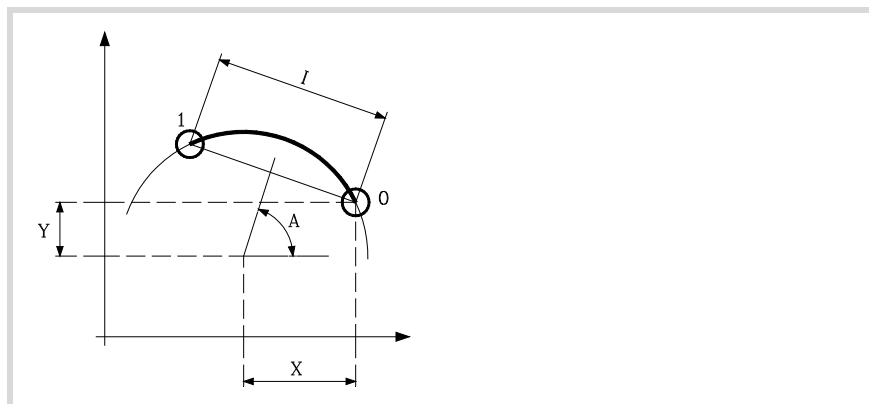
CNC 8058
CNC 8060
CNC 8065

REF: 2102

2.6 G165. Multiple machining in a chord pattern

With this function, it is possible to execute the active machining operation at the point programmed with an arch chord. The cycle will only execute one machining operation will be executed and its programming format is: When defining the machining operation, only use one parameter of the group "A" and "I".

G165 X Y A I C F



Parameters "X" and "Y" define the center of the circle, same as "I" and "J" in circular interpolations (G02, G03).

X Distance from the starting point to the center along the abscissa axis.

Y Distance from the starting point to the center along the ordinate axis.

When defining the machining operation, only one of parameters "A" and "I" is required.

A Angle, in degrees of the perpendicular bisector of the chord with respect to the abscissa axis.

I Length of the chord.

When the movement between points is done in G00 or G01, the sign indicates the direction: "I+" counterclockwise and "I-" clockwise.

C It indicates how it will move between the machining points. If not programmed, a value of C = 0 is assumed.

C=0 In rapid (G00).

C=1 Linear interpolation (G01).

C=2 In clockwise circular interpolation (G02).

C=3 In counterclockwise circular interpolation (G03).

F Feedrate for the movement between points. It will only be valid for "C" values other than zero.

Basic operation

Multiple machining is executed as follows:

- 1 The multiple machining calculates the programmed point to machine.
 - 2 Movement to that point at the feedrate programmed with "C" (G00, G01, G02 or G03).
 - 3 The multiple machining will execute the selected canned cycle after the movement.
- After the multiple machining, the tool will remain positioned at the programmed point.

2.

MULTIPLE MACHINING
G165. Multiple machining in a chord pattern

FAGOR 

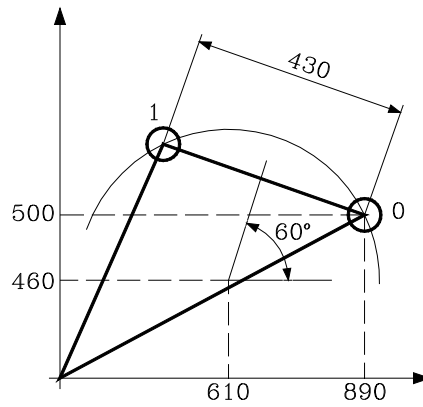
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

2.6.1 Programming example

Programming example assuming that the work plane is formed by the X and Y axes, that the Z axis is the longitudinal axis and that the starting point is X0 Y0 Z0:



```
G00 G91 X890 Y500 F100 S500
G98 G81 Z-8 I-22
G165 X-280 Y-40 A60 C1 F200
G80
G90 X0 Y0
M30
```

Multiple machining may also be defined as follows:

```
G165 X-280 Y-40 I430 C1 F200
```

2.

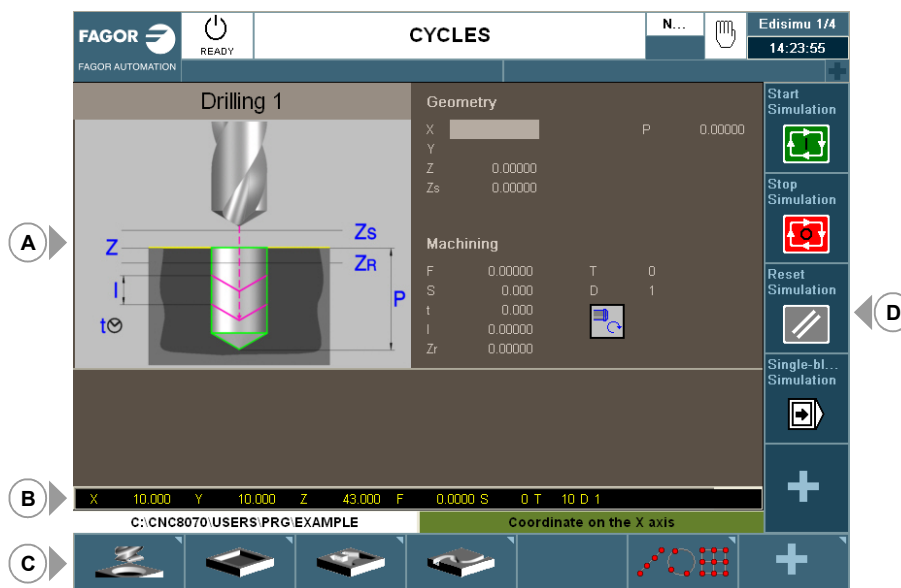
MULTIPLE MACHINING
G165. Multiple machining in a chord pattern

FAGOR 
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

The cycle editor is accessed from the EDISIMU mode, either directly from the softkey menu or by selecting a canned cycle in the part-program and pressing [RECALL]. When selecting a canned cycle, the editor displays the window for defining that canned cycle. Besides editing the cycles, with the cycle editor it is also possible to simulate the cycle graphically even if it is not included in the part-program.



- A Area for editing and simulating canned cycles.
- B Teach-in mode.
- C Softkey menu for selecting various cycles, activating the teach-in mode and configuring the cycle editor.
- D Softkey menu to simulate the cycle selected at the editor.

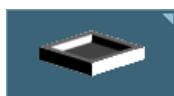
Select the machining cycles.

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.



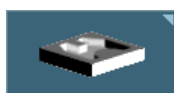
Z axis machining.

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-empted circular pocket, rectangular boss and circular boss.



2D/3D profile pockets.

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

3.

**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 104.

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).

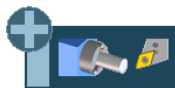
The "+" softkey displays the softkey to access the probe cycles of the milling model.

Accessing the milling cycles.

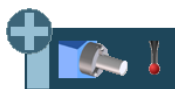
The "+" softkey displays the softkey to access the milling machining cycles.

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.

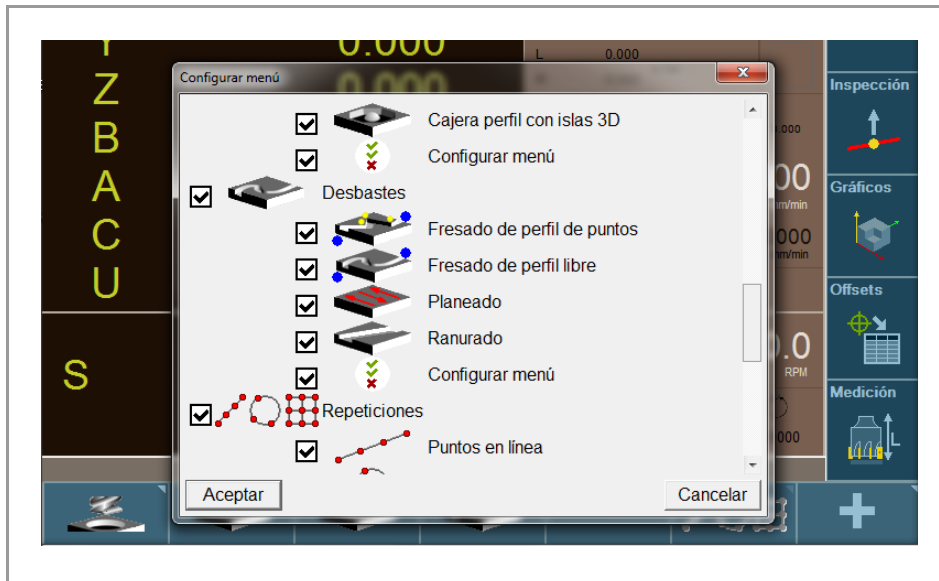
Accessing the probe cycles of the lathe model (if available).

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.

CYCLE EDITOR

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

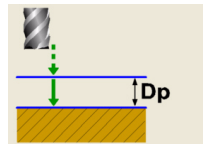
Enable the capability to program the next tool (the one that will be executed after the cycle) in the cycles. When it is a random magazine, it prepares the tool while the cycle is being executed, hence reducing the machining time.



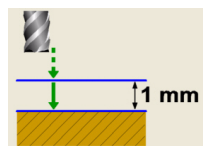
On the cycle screens, the display must be activated in order to see and define the next tool; otherwise, the data will not be displayed.

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.

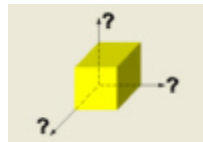


This option is enabled. The cycles show parameter ·Dp· for programming the part surface approach distance.



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.

CYCLE EDITOR
Configuring the cycle editor.

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.

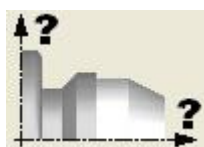
Enable the programming of M functions in the canned cycles, to execute them before each machining operation. This permits, for example, to execute subroutines associated with M functions before the various operations.

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.

CYCLE EDITOR
Configuring el cycle editor (rotary axis cycles).

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

CYCLE EDITOR
Teach-in mode.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

CYCLE EDITOR
Selecting data, profiles and icons

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

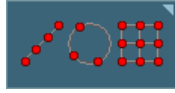
REF: 2102

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-empted 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.

Drilling 1

| Geometry | |
|----------|-----------|
| Z | P 0.00000 |
| X | |
| Y | 0.00000 |
| Ys | 0.00000 |

| Machining | | | |
|-----------|---------|---|---|
| F | 0.00000 | T | 0 |
| S | 0.000 | D | 1 |
| t | 0.000 | | |
| I | 0.00000 | | |
| Yr | 0.00000 | | |

| | |
|-------|---------|
| N | 1 |
| alpha | 0.00000 |
| L | 0.00000 |

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 area for the graphics. When quitting the graphics, it will assume as the new display area the one calculated last.

3.

CYCLE EDITOR
Simulate a canned cycle.

FAGOR

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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, first 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 AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

WORK PLANES AND SHIFTING OF THE MACHINING OPERATIONS.

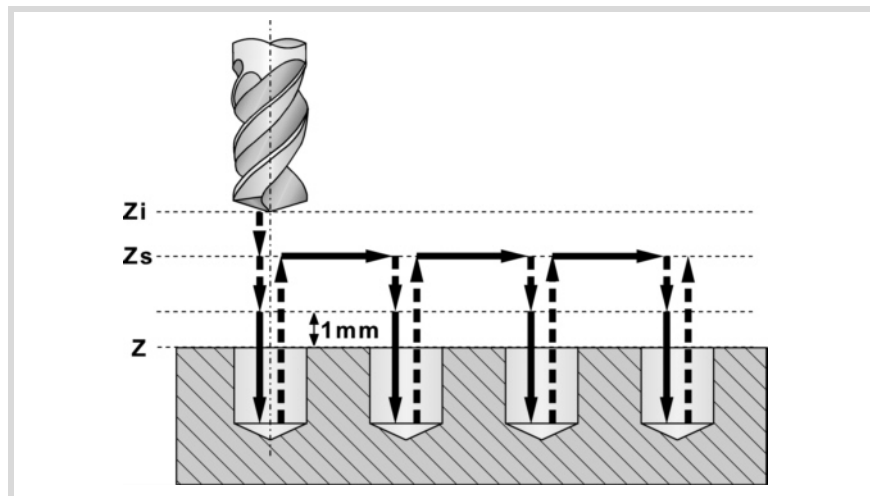
4

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

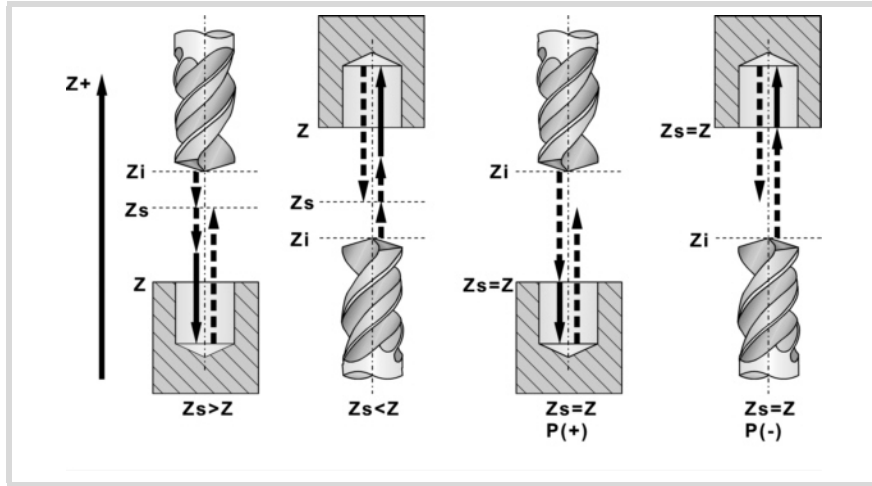
FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

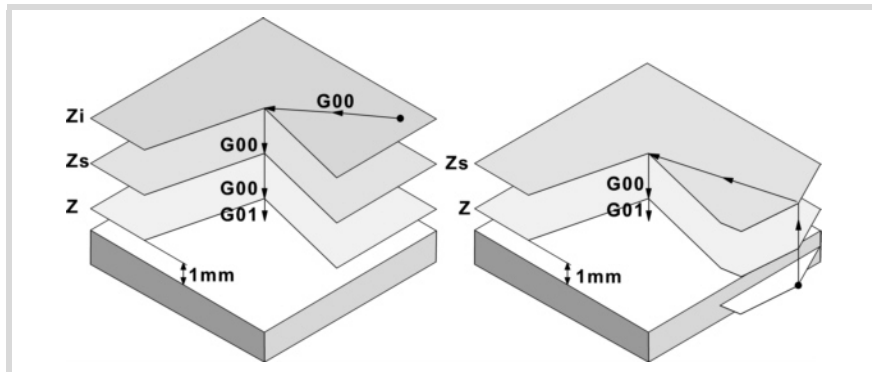
(parameter P). If $Z=Z_s$ and $P>0$ machining in the negative direction of the longitudinal axis (Z-), if $Z=Z_s$ and $P<0$ machining in the positive direction (Z+).



Movments in the work planes.

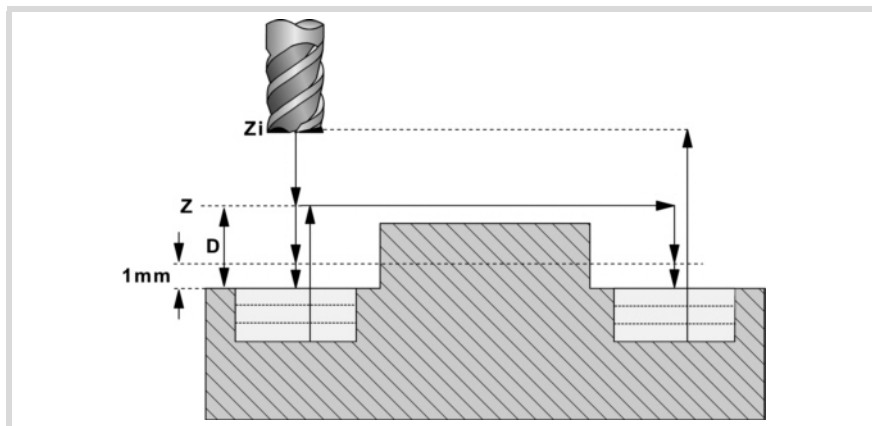
When beginning to execute the cycle, the tool moves in rapid (G0) from the starting plane (Zi) to the safety plane (Zs).

- 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.



4.

WORK PLANES AND SHIFTING OF THE MACHINING



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

4.0.1 Value applied when the value of a parameter is 0

Penetration step I=0:

When programming I=0, it assumes as step the cutting length assigned to the tool in the tool table.

An error will be issued if the table value is also 0.

Penetration feedrate Fz=0:

When programming Fz=0, the roughing and finishing penetration takes place at half the milling feedrate "F" selected for each operation.

Penetration angles $\beta=0$ and $\theta=0$:

In both cases, when programming 0, it takes the value assigned to the table in the tool table.

If the table value is also 0, it penetrates vertically, without inclination, 90° angle.

Finishing passes or number of penetrations N=0:

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

In pockets and bosses (except in 2D and 3D pockets), if the table value is also 0, it checks the roughing and finishing tools. If it is the same, the wall finishing is carried out with tangential entry and exit at each penetration after the roughing operation.

An error will be issued if they are different.

4.

4.

WORK PLANES AND SHIFTING OF THE MACHINING



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.](#)".

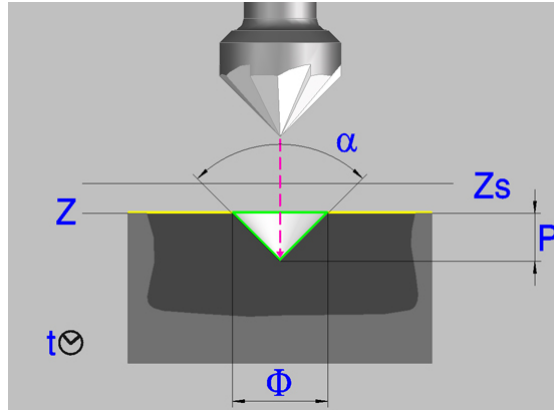
- Points in line.
- Points in arc.
- Points in rectangle.
- Points in grid.
- Random points (several points defined by the user).

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.

5.

CANNED CYCLES OF THE EDITOR: Z AXIS MACHINING.
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 $Z=Z_s$ 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.

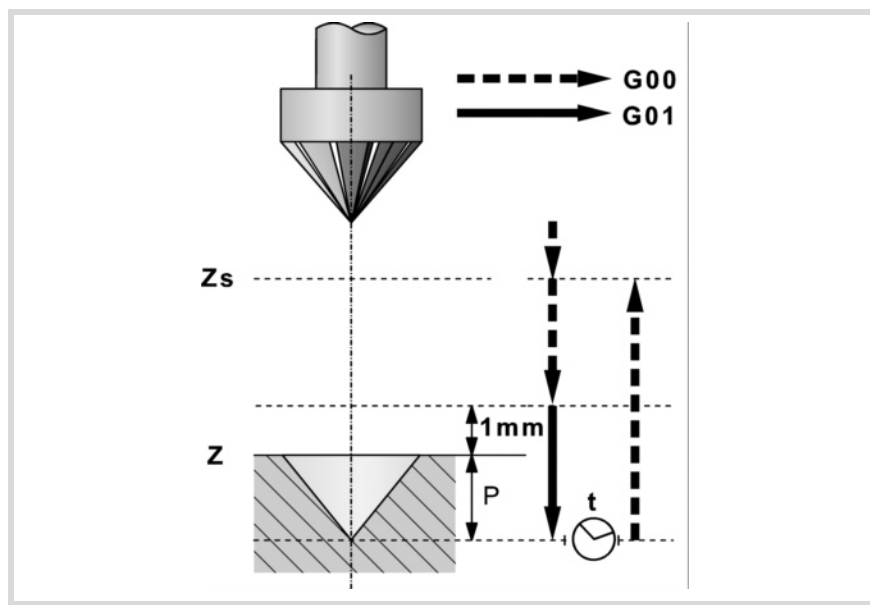
Next tool.



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.

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.

CANNED CYCLES OF THE EDITOR. Z AXIS MACHINING.
Center punching.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

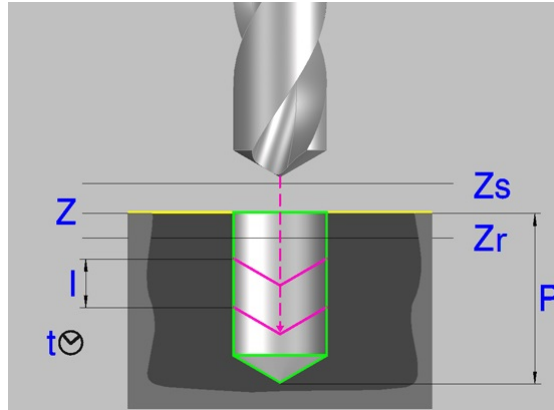
REF: 2102

5.2 Drilling.

5.

CANNED CYCLES OF THE EDITOR: Z AXIS MACHINING.

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.



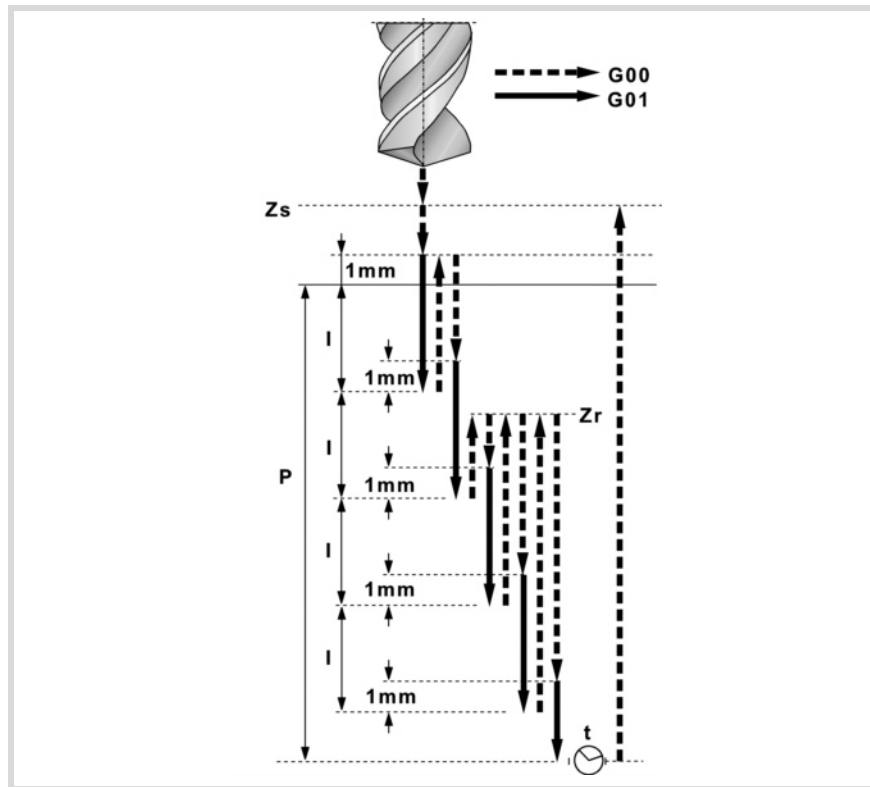
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

Drilling.

CANNED CYCLES OF THE EDITOR. Z AXIS MACHINING.

FAGOR 

FAGOR AUTOMATION

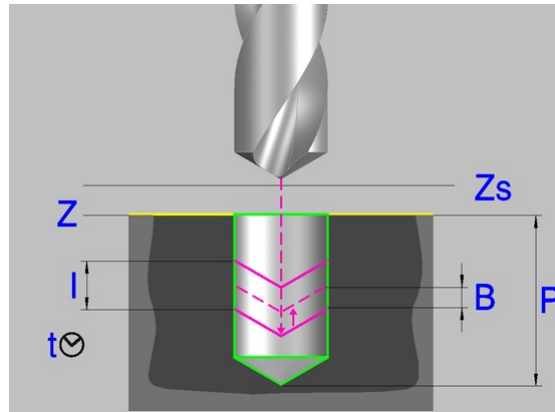
CNC 8058
CNC 8060
CNC 8065

REF: 2102

5.3 Deep hole drilling.

5.

CANNED CYCLES OF THE EDITOR: Z AXIS MACHINING.
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.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

CANNED CYCLES OF THE EDITOR. Z AXIS MACHINING.

Deep hole drilling.



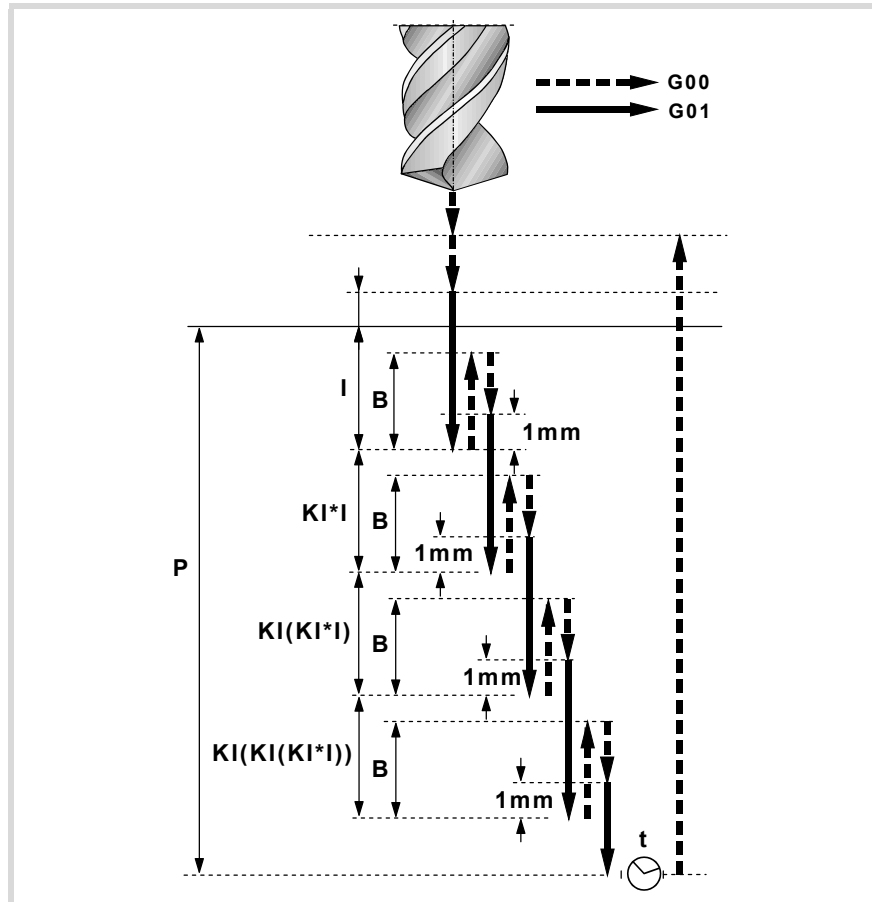
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

CANNED CYCLES OF THE EDITOR: Z AXIS MACHINING.
Deep hole drilling.



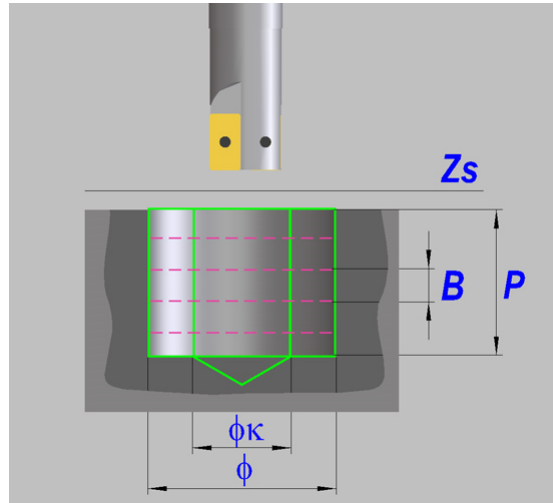
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.



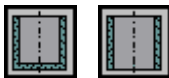
Geometric parameters:

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.

5.

CANNED CYCLES OF THE EDITOR. Z AXIS MACHINING.
Bore milling.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

CANNED CYCLES OF THE EDITOR. Z AXIS MACHINING. Bore milling.



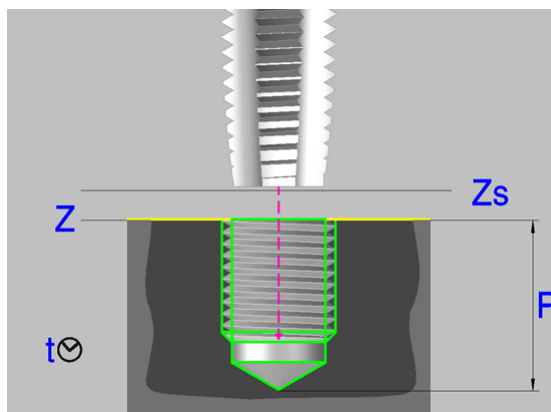
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

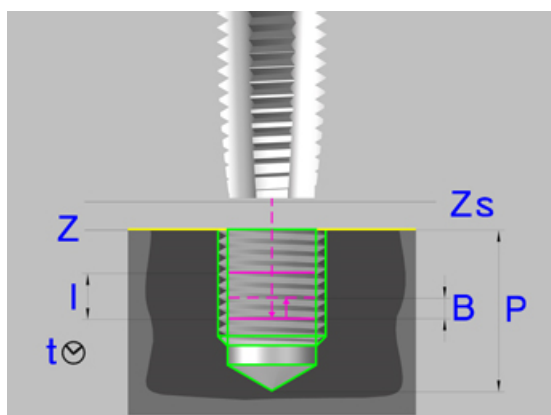
REF: 2102

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.

5.

CANNED CYCLES OF THE EDITOR. Z AXIS MACHINING.
Tapping.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

5.

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.



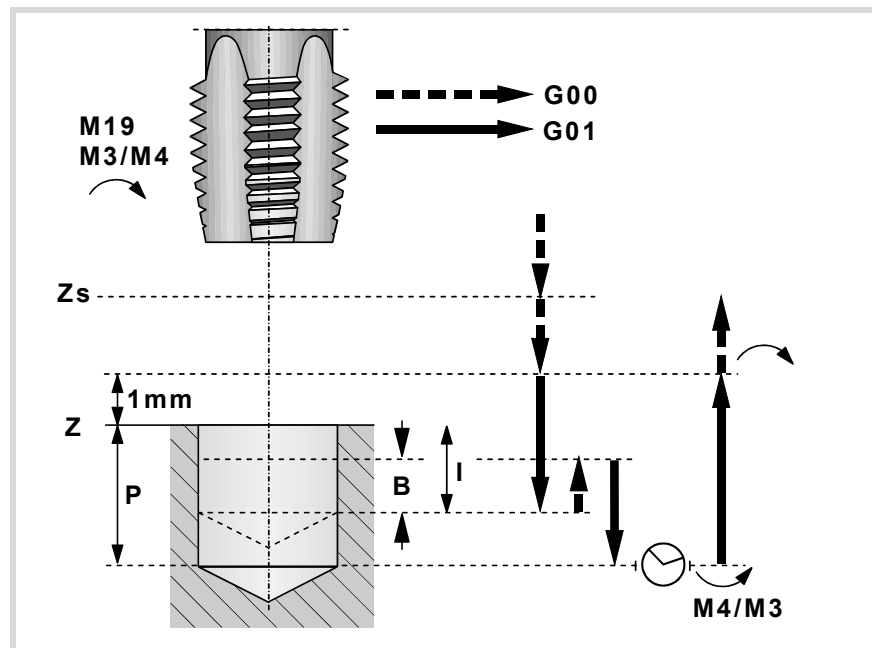
Activating or deactivating the preparation of the next tool.



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.

Tapping.

CANNED CYCLES OF THE EDITOR. Z AXIS MACHINING.

FAGOR 

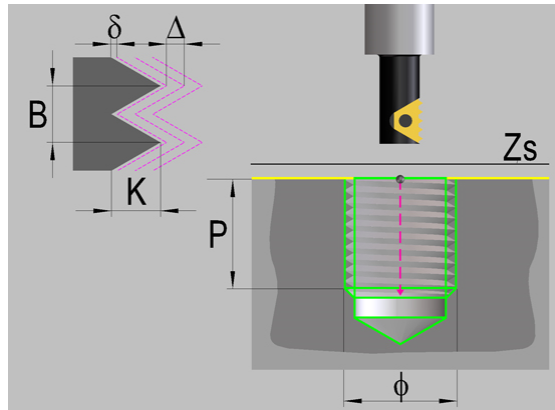
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

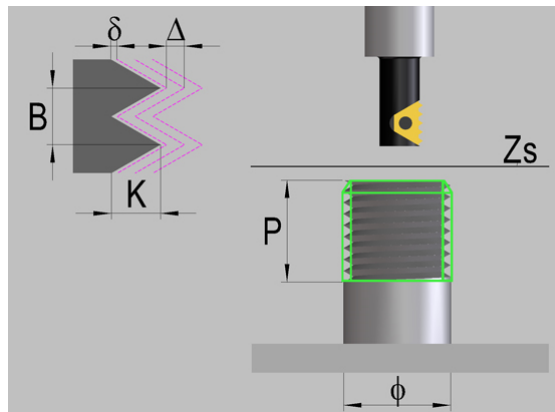
REF: 2102

5.6 Thread milling.

Inside thread milling.



Outside 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.

5.

CANNED CYCLES OF THE EDITOR: Z AXIS MACHINING.
Thread milling.

Canned cycles (·M· model).

| | |
|----------|---|
| 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.

CANNED CYCLES OF THE EDITOR. Z AXIS MACHINING.
Thread milling.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

CANNED CYCLES OF THE EDITOR: Z AXIS MACHINING.
Thread milling.

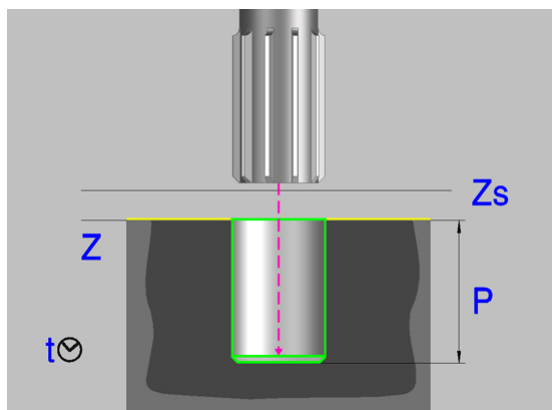


FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

5.

CANNED CYCLES OF THE EDITOR. Z AXIS MACHINING.
Reaming.

FAGOR 

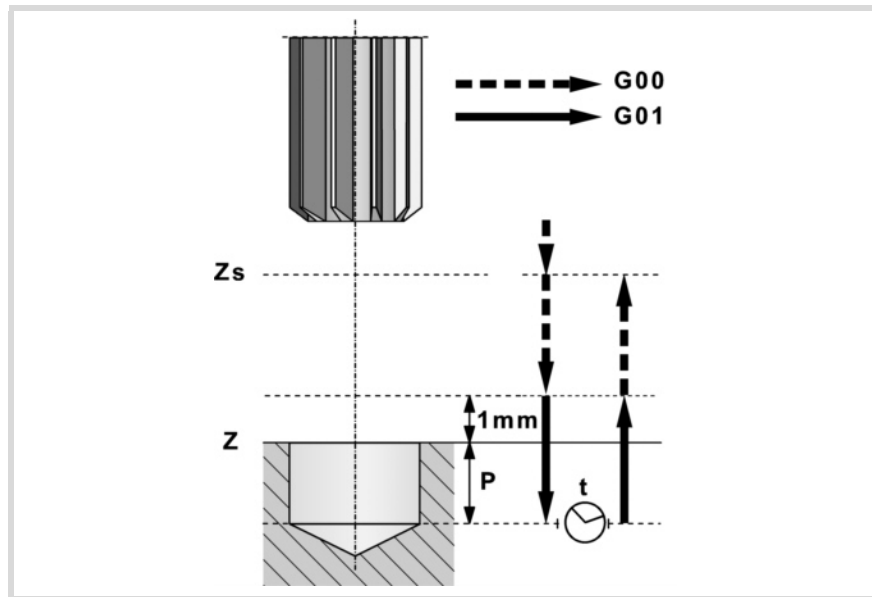
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

CANNED CYCLES OF THE EDITOR. Z AXIS MACHINING. Reaming.

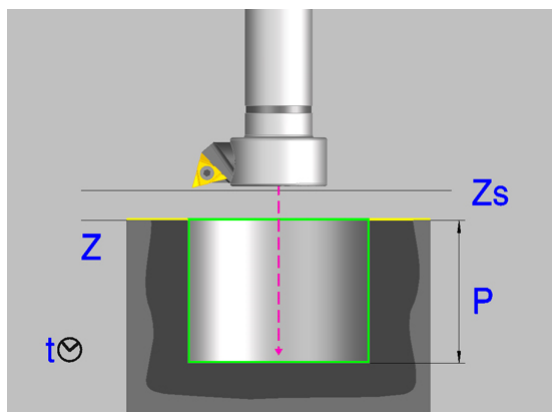


FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

Boring.

CANNED CYCLES OF THE EDITOR. Z AXIS MACHINING.

FAGOR 

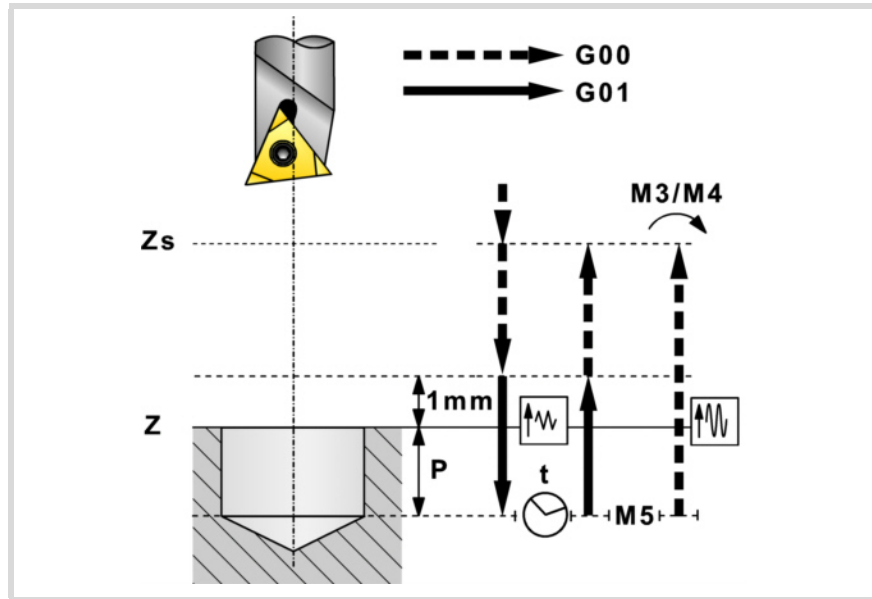
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

CANNED CYCLES OF THE EDITOR: Z AXIS MACHINING. Boring.

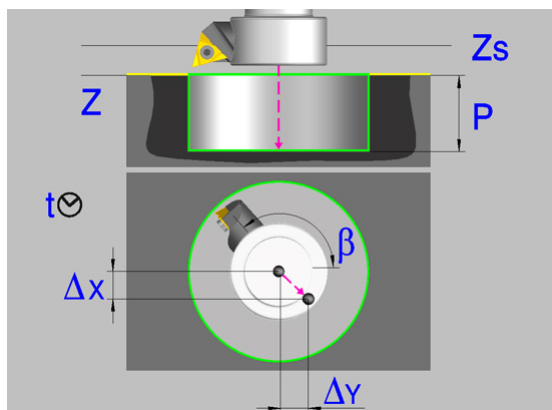


FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

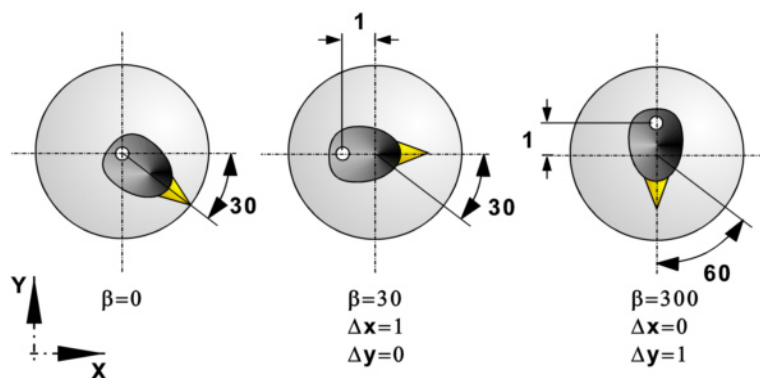
5.9 Boring with spindle orientation.



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.
- $\Delta x, \Delta y$ 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 -30° 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.

5.

CANNED CYCLES OF THE EDITOR. Z AXIS MACHINING.
Boring with spindle orientation.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

CANNED CYCLES OF THE EDITOR. Z AXIS MACHINING.
Boring with spindle orientation.



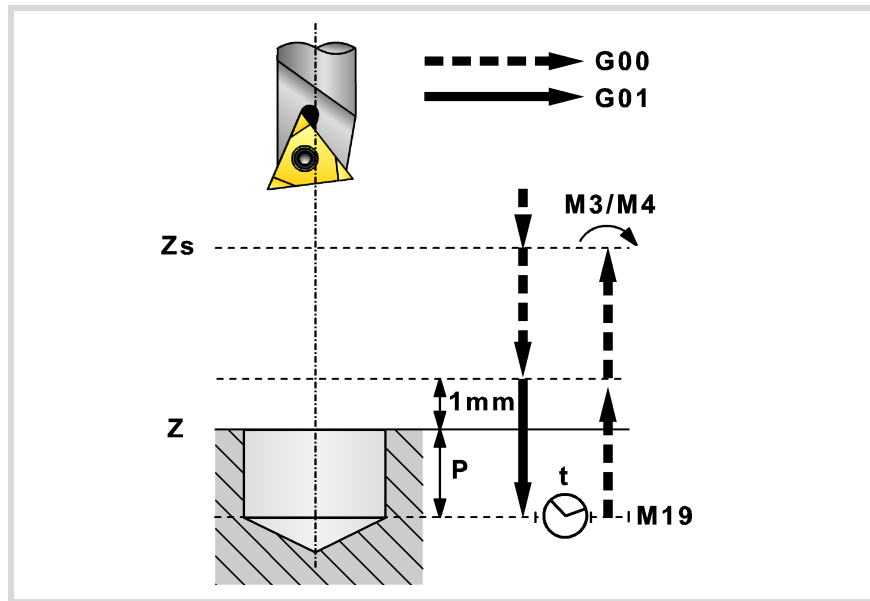
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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 " $\Delta x, \Delta y$ ".
- 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.

5.

CANNED CYCLES OF THE EDITOR: Z AXIS MACHINING.

Boring with spindle orientation.



FAGOR AUTOMATION

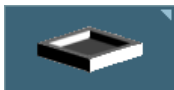
CNC 8058
CNC 8060
CNC 8065

REF: 2102

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-empted circular pocket.
- Rectangular boss.
- Circular boss.



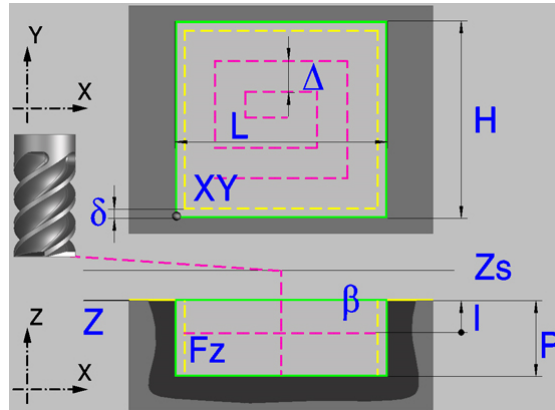
Multiple machining may be added to pocket cycles so the cycle is repeated in several points. The available multiple machining operations are the following. See chapter "[9 Canned cycles of the editor. Multiple positioning.](#)".

- 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.

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).



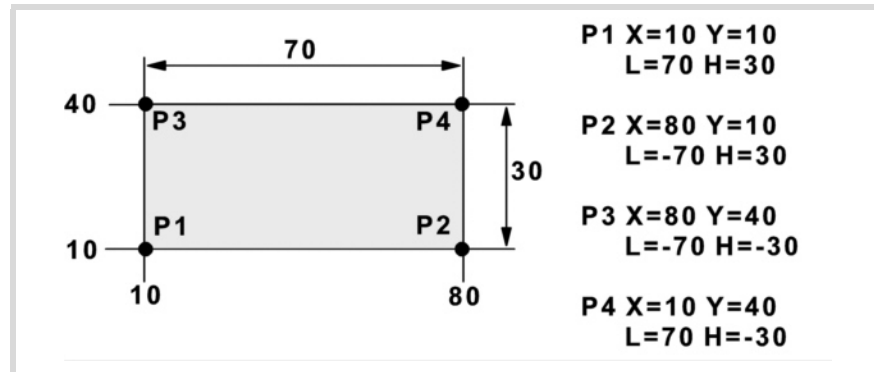
Starting point at one corner of the pocket.



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.

6.

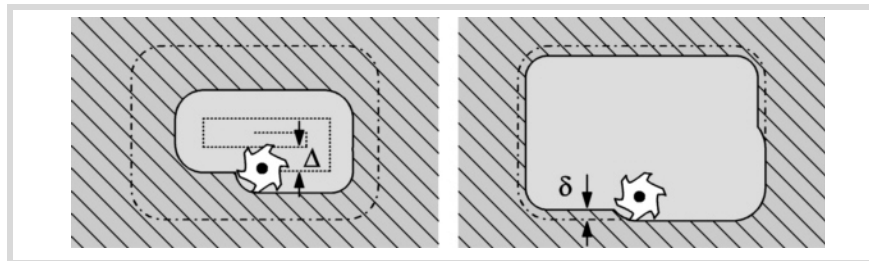
CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES.
Simple rectangular pocket.



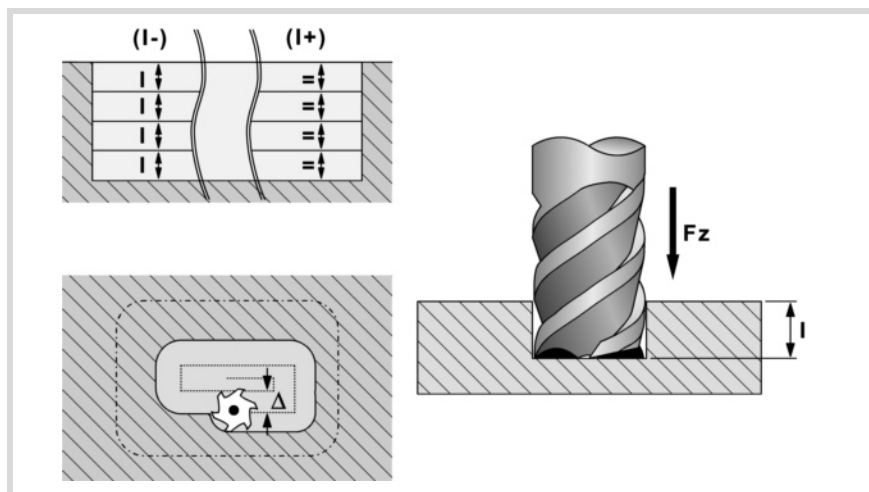
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102



- 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.

6.

CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES.

Simple rectangular pocket.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES.

Simple rectangular pocket.



FAGOR AUTOMATION

CNC 8058

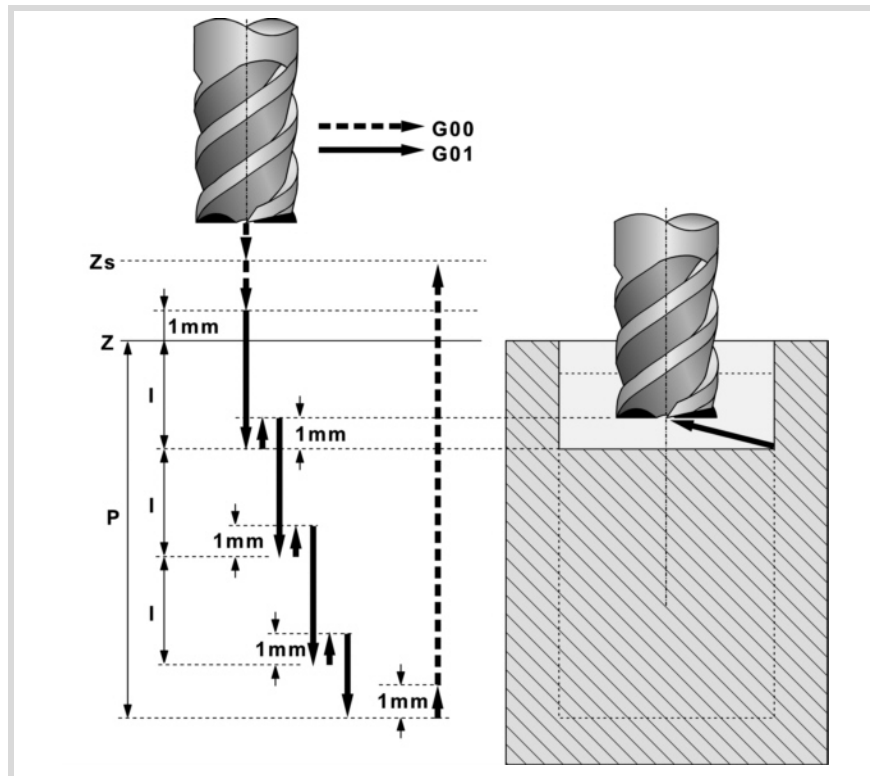
CNC 8060

CNC 8065

REF: 2102

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.

CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES.
Simple rectangular pocket.

FAGOR 

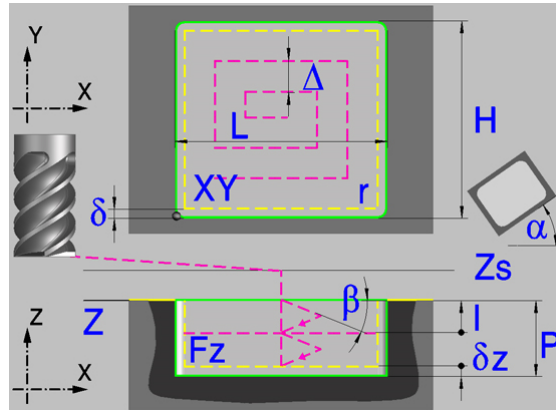
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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).

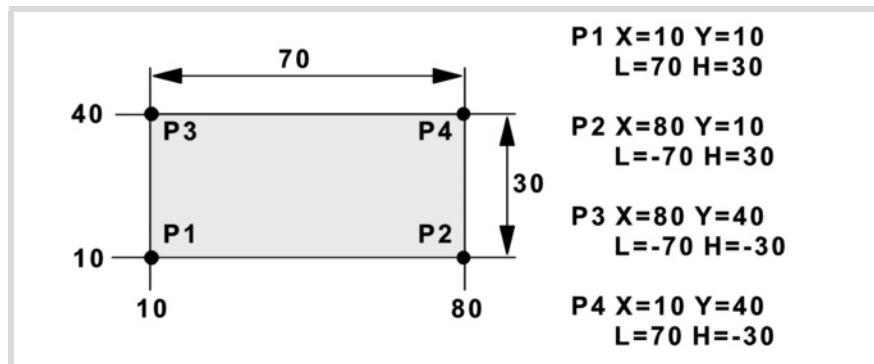


Starting point at one corner of the pocket.



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.

6.

CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES.
Rectangular pocket with rounding.



FAGOR AUTOMATION

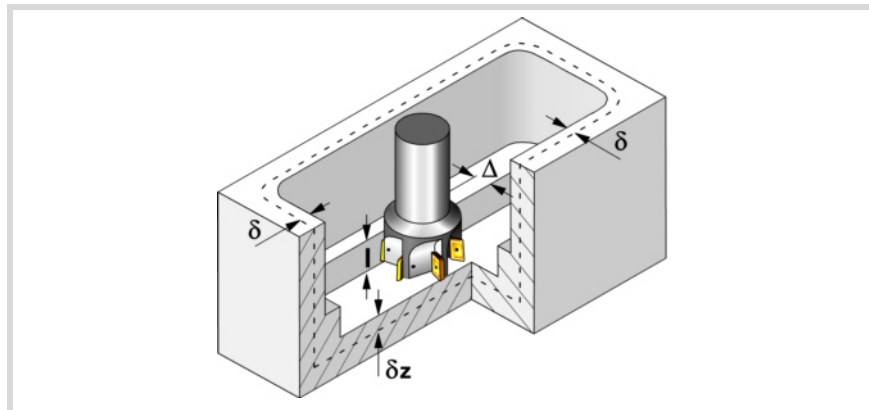
CNC 8058
CNC 8060
CNC 8065

REF: 2102

Roughing parameters:

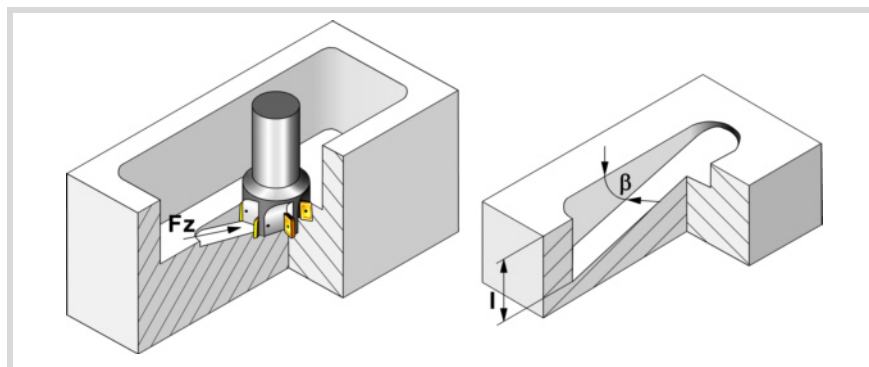
The roughing operation empties the pocket leaving the following finishing stocks. Both stocks are defined as finishing parameters.

- 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.

6.

CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES.

Rectangular pocket with rounding.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

6.

CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES.
Rectangular pocket with rounding.

Spindle turning direction (icon).



Clockwise.



Counterclockwise.

Machining direction (icon).



Clockwise.



Counterclockwise.

Programming of M functions.



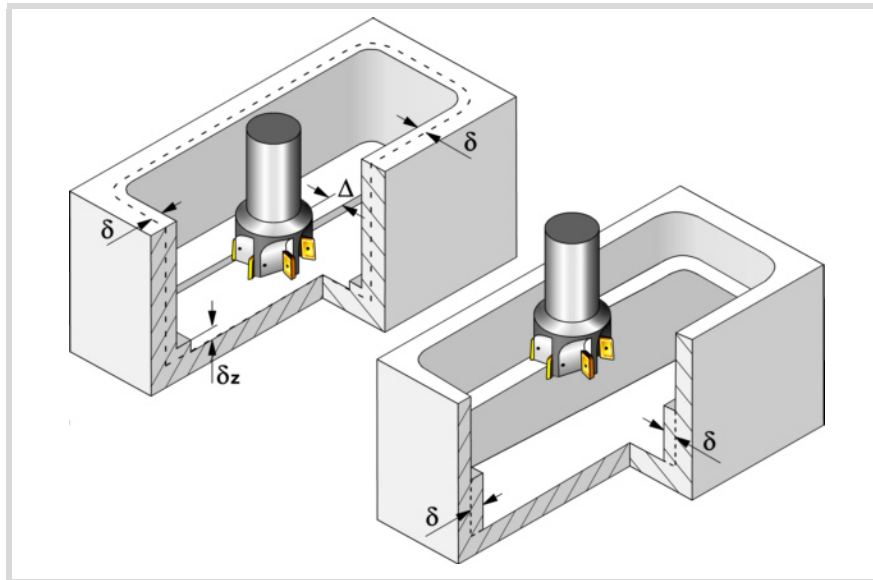
Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.



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, the 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.



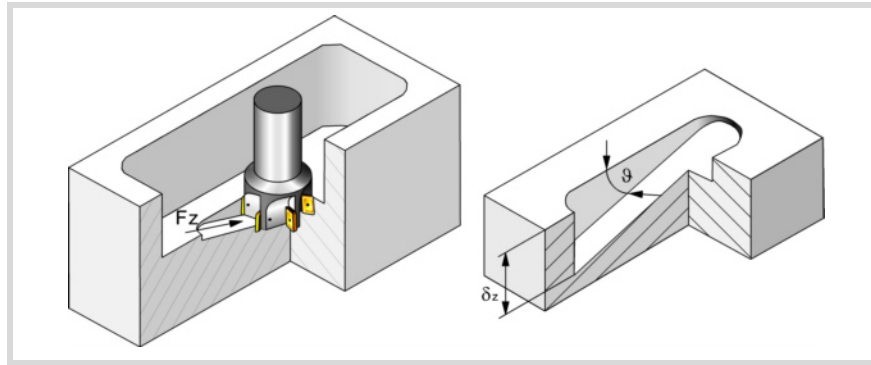
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

0 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.

6.

CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES.

Rectangular pocket with rounding.



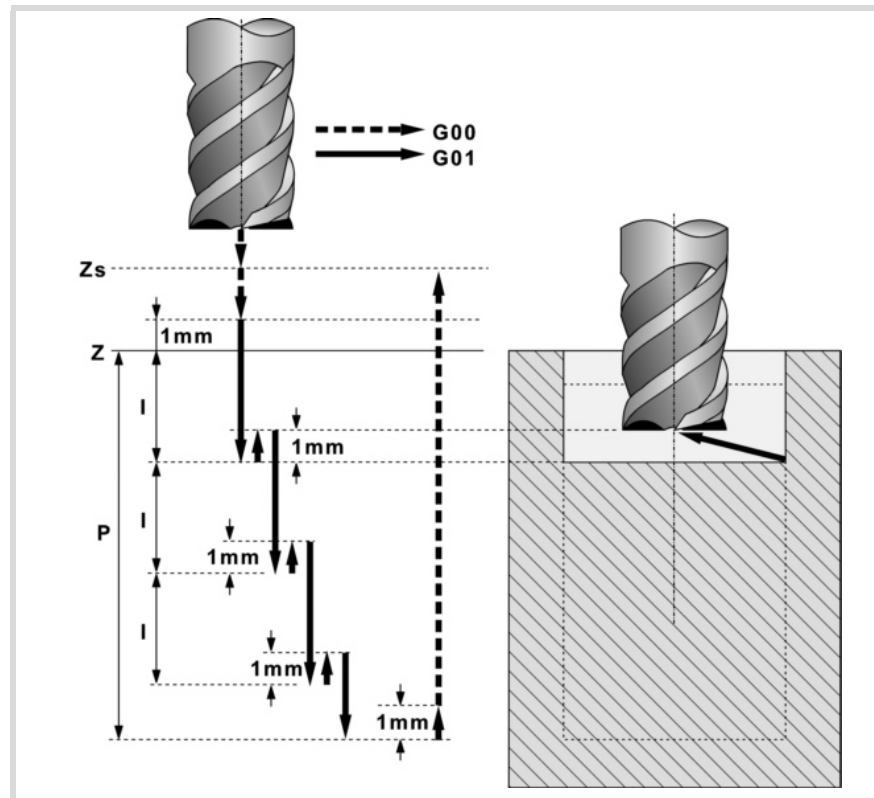
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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 " l " 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.

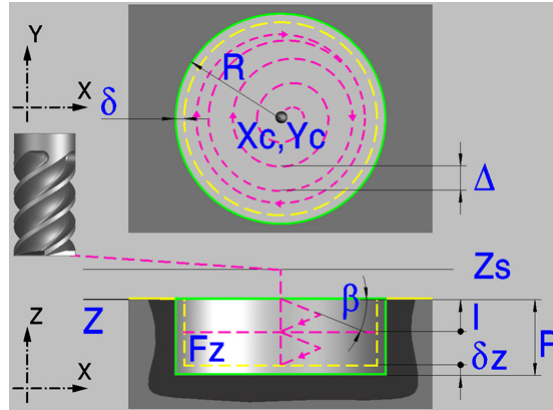
CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES.
Rectangular pocket with rounding.

FAGOR 
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

6.3 Circular pocket.



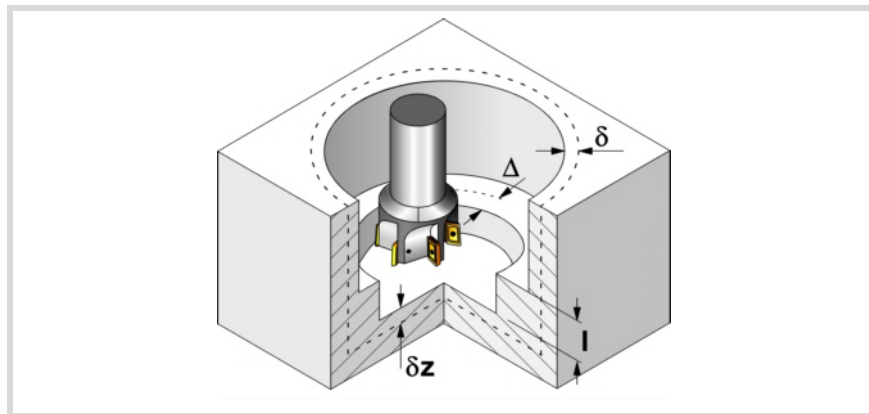
Geometric parameters:

- 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.

- delta Finishing stock on the side walls.
- delta z Finishing stock at the bottom of the pocket.



The roughing operation defining parameters are:

- Delta 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 (+), 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 (-), 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.

6.

CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES. Circular pocket.

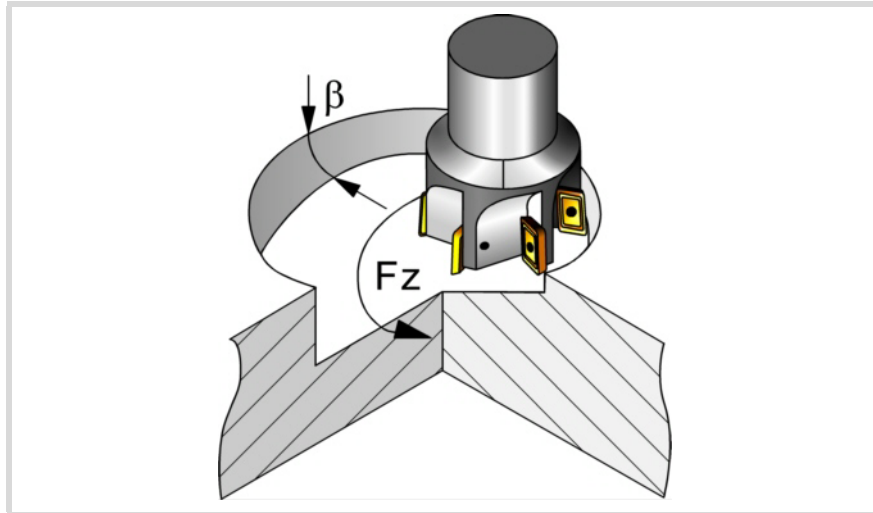


FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

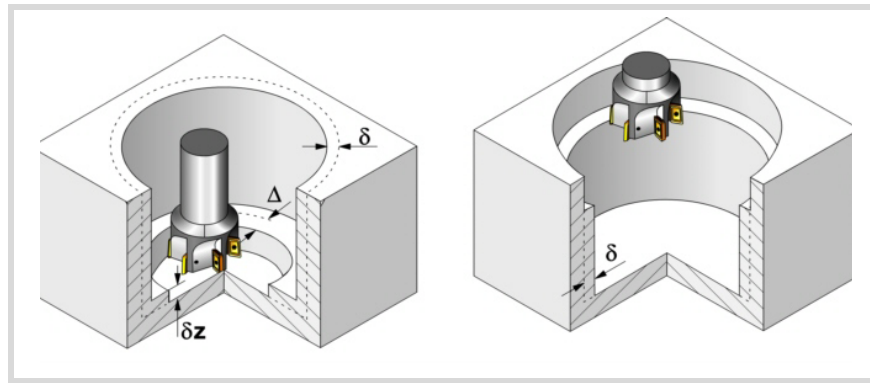
6.

CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES.
Circular pocket.

FAGOR 
FAGOR AUTOMATION

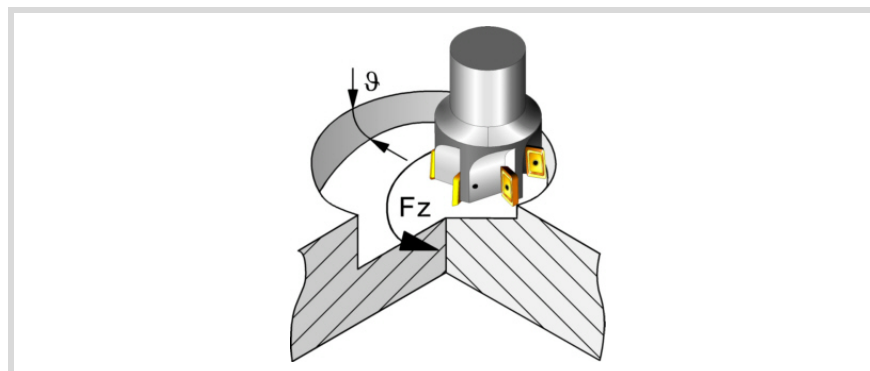
CNC 8058
CNC 8060
CNC 8065

REF: 2102



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.

6.

Circular pocket.

CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

6.

CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES.
Circular pocket.

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.



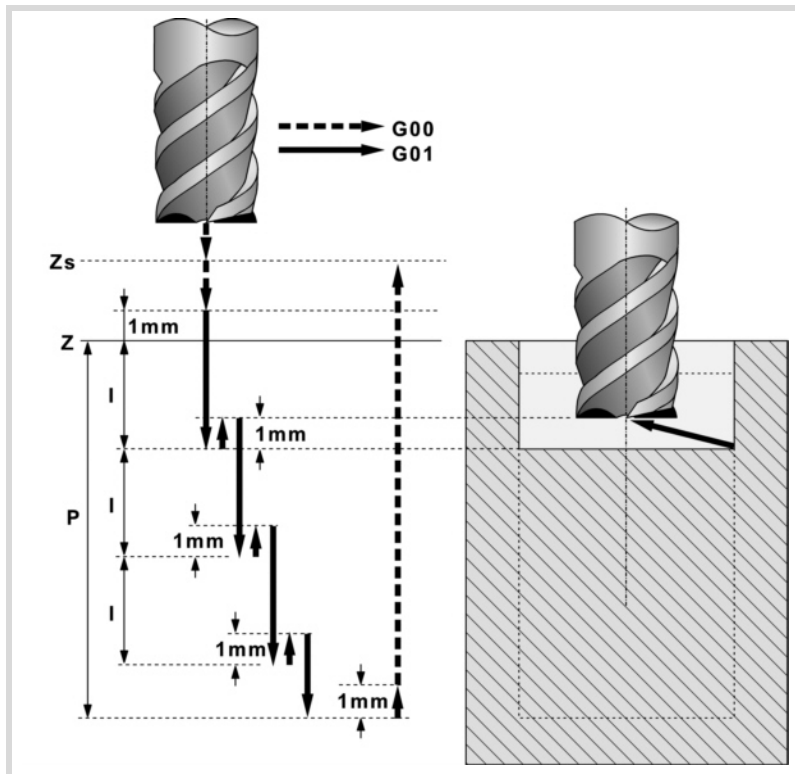
Activating or deactivating the preparation of the next tool.



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

6.3.1 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.

CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES.
Circular pocket.

FAGOR 

FAGOR AUTOMATION

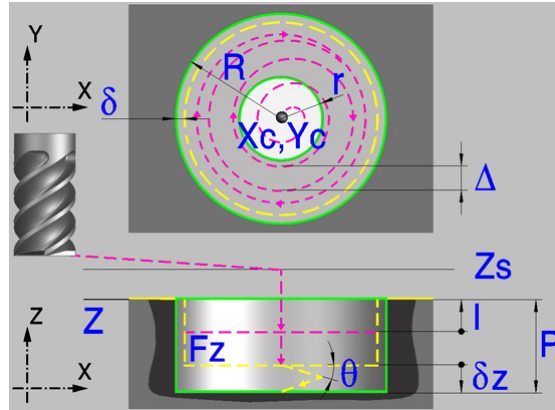
CNC 8058
CNC 8060
CNC 8065

REF: 2102

6.4 Pre-empted circular pocket.

6.

CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES.
Pre-empted circular pocket.



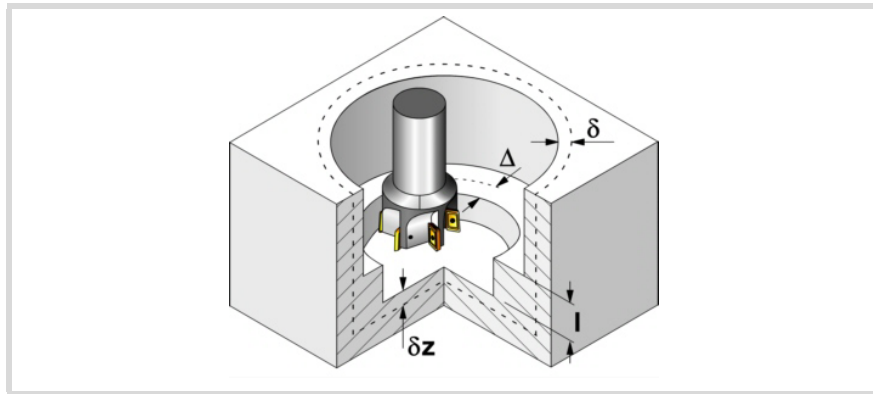
Geometric parameters:

- X_c, Y_c Center of the pocket.
 R Pocket radius.
 r Pre-empting radius.
 Z Part surface coordinate.
 Z_s 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.



FAGOR AUTOMATION

CNC 8058
 CNC 8060
 CNC 8065

REF: 2102

Canned cycles (·M· model).

- 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.

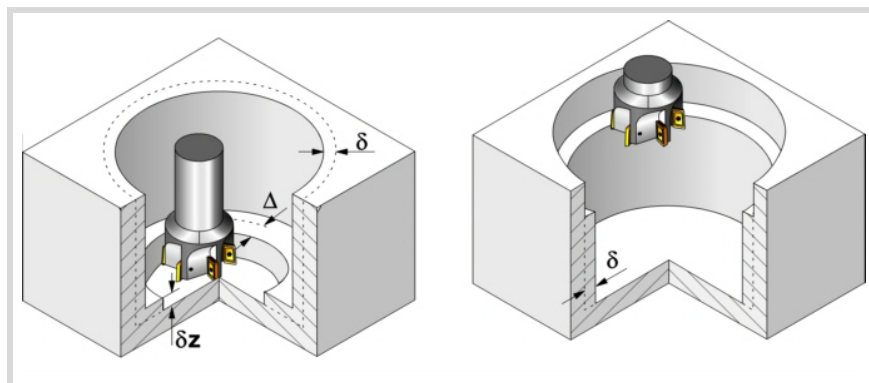


Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.

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.

6.

CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES.

Pre-empted circular pocket.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

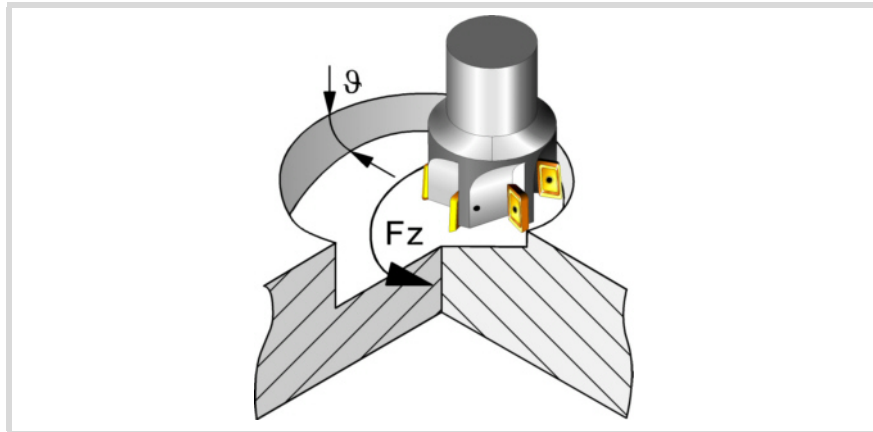
REF: 2102

6.

CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES.
Pre-empted circular pocket.

0 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.

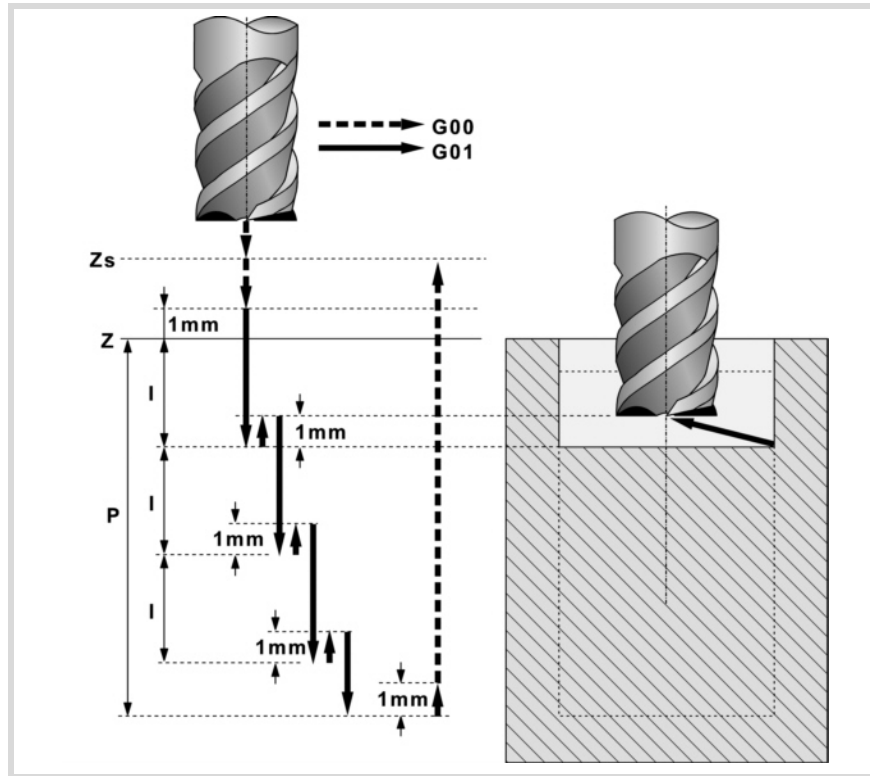


Activating or deactivating the preparation of the next tool.

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

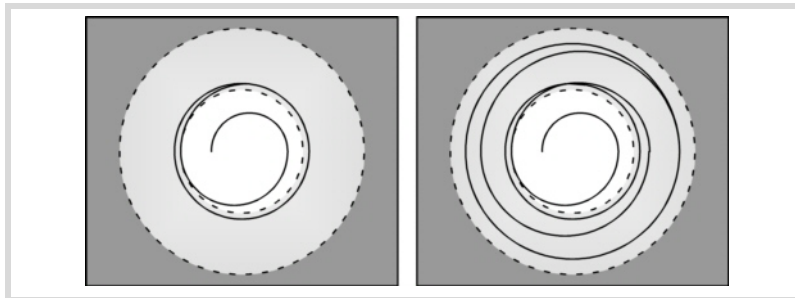
6.4.1 Basic operation.

- 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-empted 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).

6.

CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES.
Pre-empted circular pocket.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

Pre-empted circular pocket.



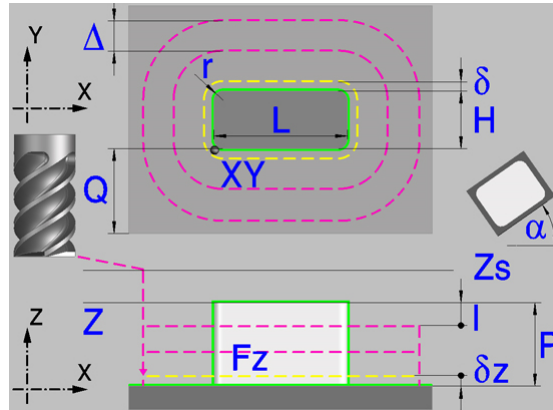
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

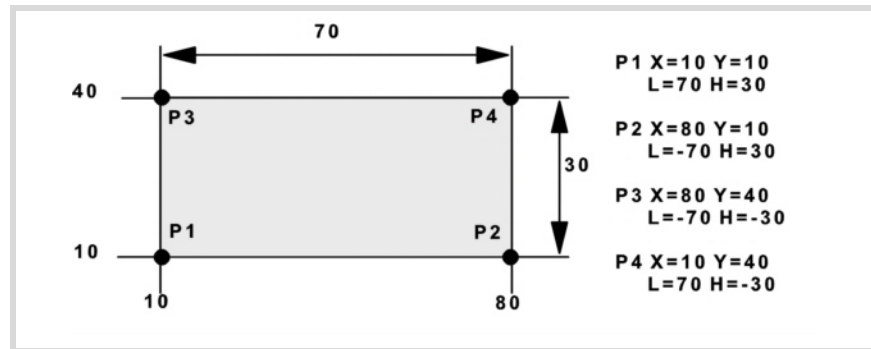
REF: 2102

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.
- alpha 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.

6.

CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES.

Rectangular boss.

FAGOR 

FAGOR AUTOMATION

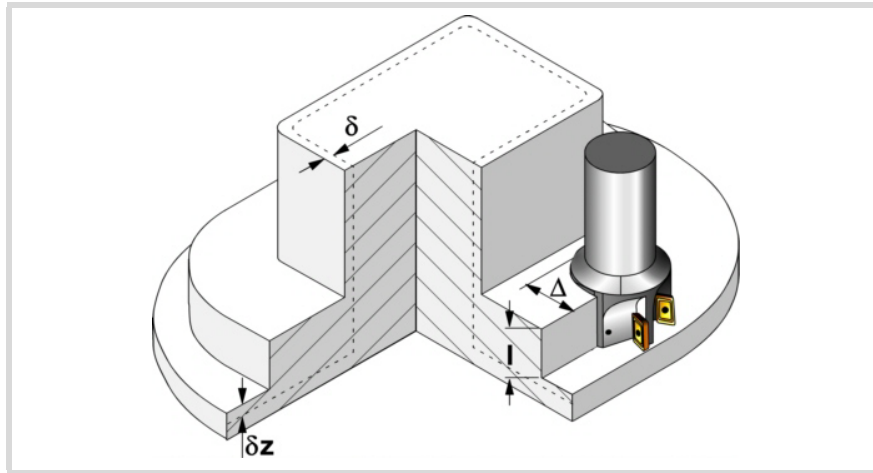
CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

6.

CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES.
 Rectangular boss.

FAGOR 
 FAGOR AUTOMATION

CNC 8058
 CNC 8060
 CNC 8065

REF: 2102

Programming of M functions.

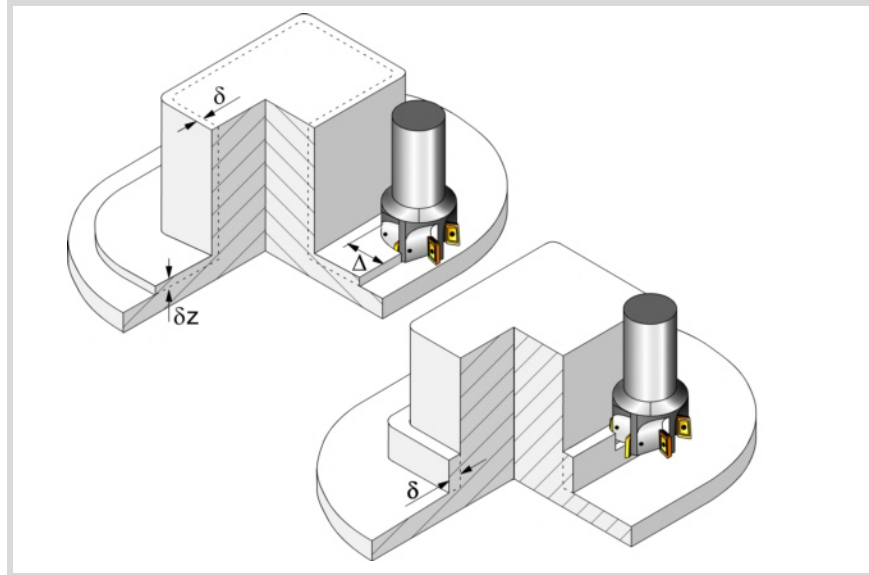


Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.

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.

CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES.

Rectangular boss.



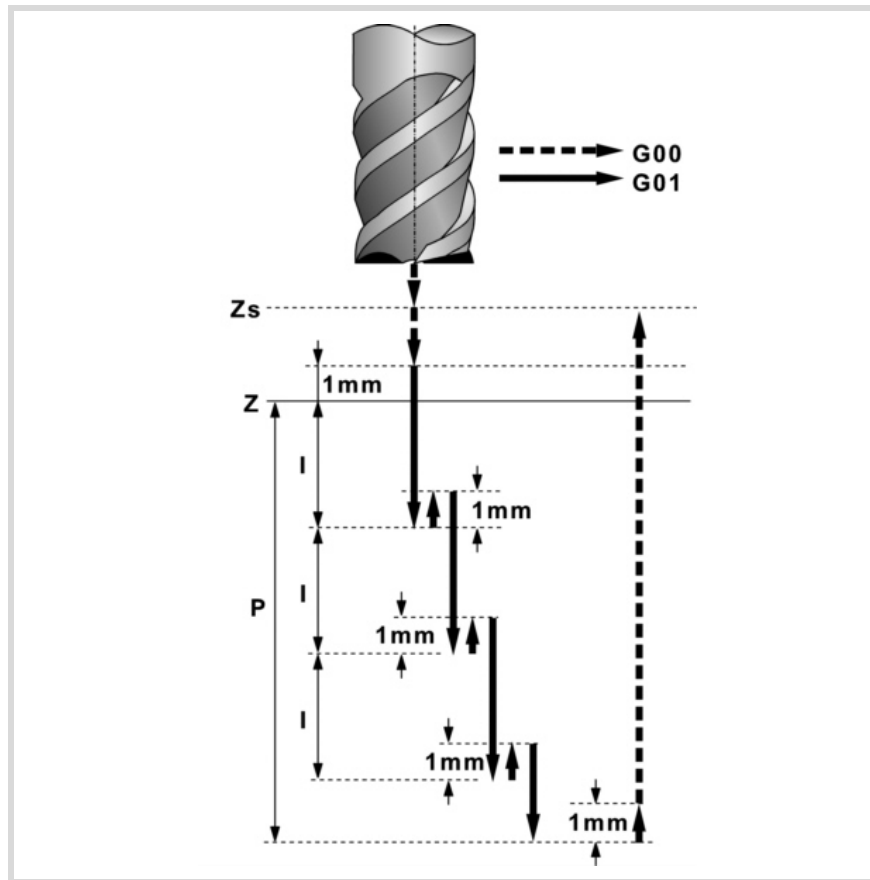
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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 "l" 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.
Rectangular boss.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

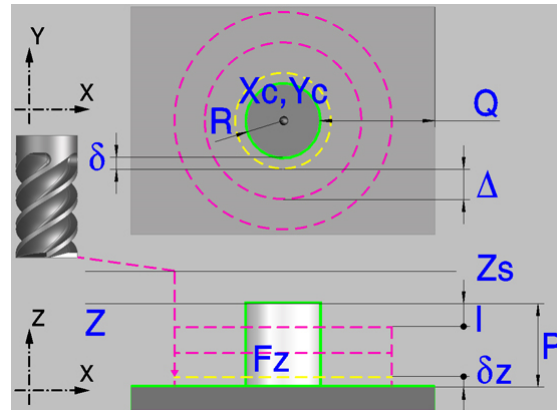
REF: 2102

6.6 Circular boss.

6.

CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES.

Circular boss.

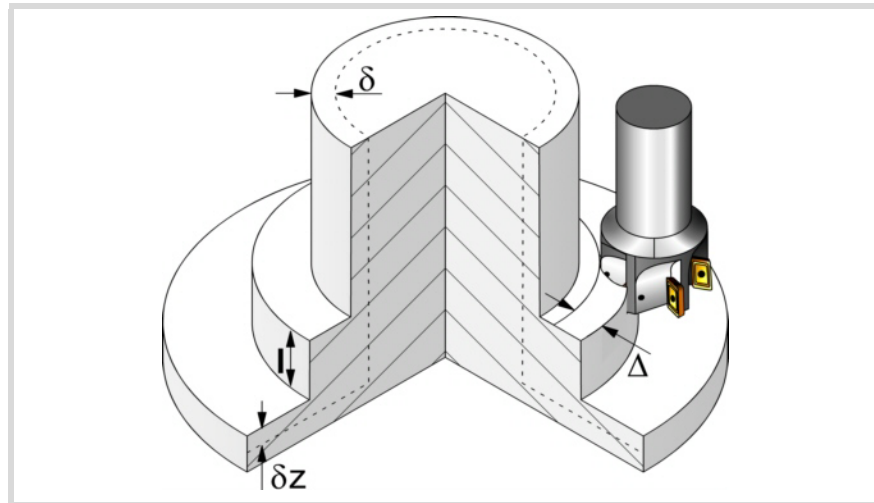
**Geometric parameters:**

- X_c, Y_c Center of the boss.
- R Boss radius.
- Z Part surface coordinate.
- Z_s 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.

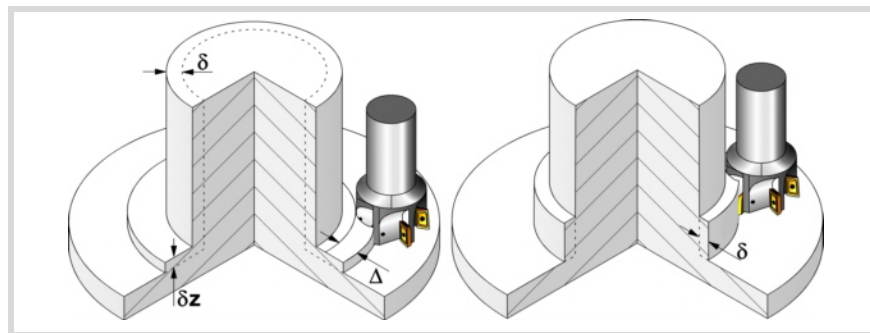


Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.

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.

6.

- 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.

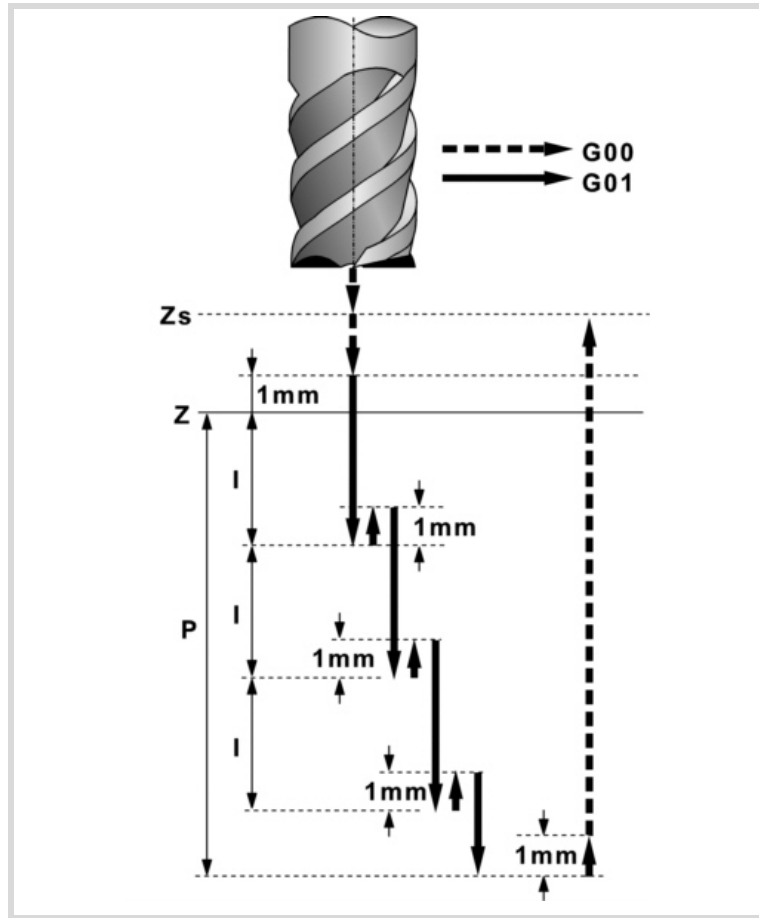


Activating or deactivating the preparation of the next tool.

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 "l" 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 AUTOMATION

CNC 8058

CNC 8060

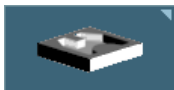
CNC 8065

REF: 2102

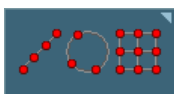
CANNED CYCLES OF THE EDITOR. 2D/3D PROFILE POCKETS.

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.](#)".

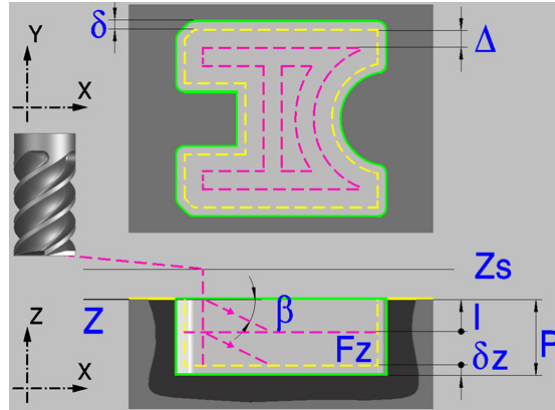
- Points in line.
- Points in arc.
- Points in rectangle.
- Points in grid.
- Random points (several points defined by the user).

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.

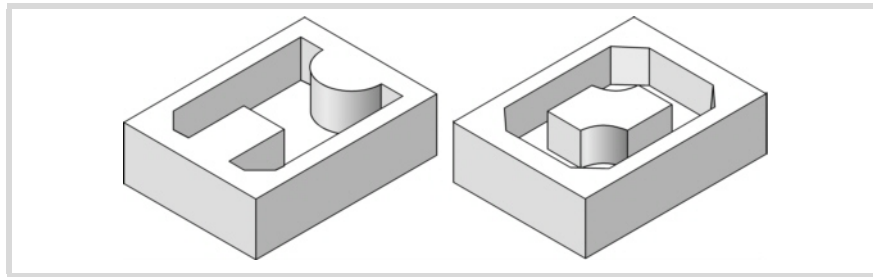
7.1 2D profile pocket.

7.

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



FAGOR AUTOMATION

CNC 8058
 CNC 8060
 CNC 8065

REF: 2102

Machining strategy (icon).



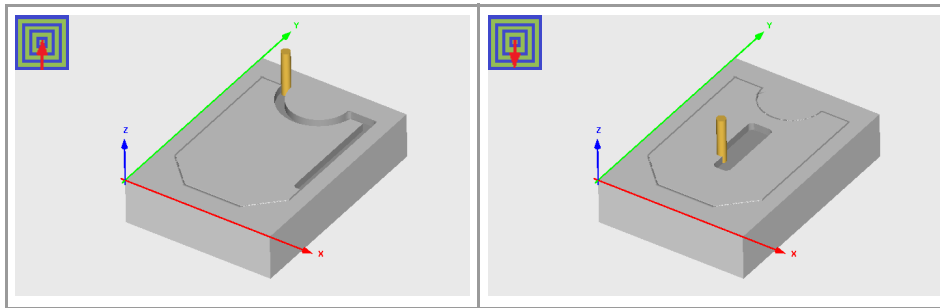
Outward-inward concentric machining. The wall of the pocket is machined in the first pass. Depending on the geometry, this option may be faster, but the wear on the tool is greater.

- 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.

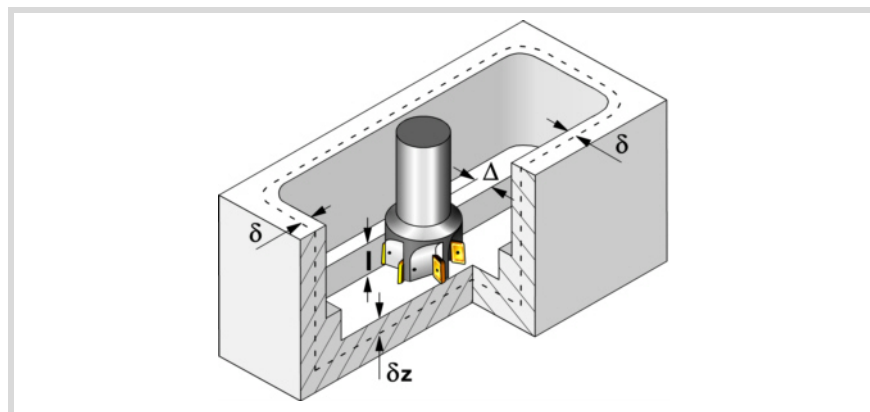
- The starting point does not depend on the initial point of the outer contour and can vary if either the tool radius or the machining pass is changed (milling width). In a pocket without islands, the initial point is found near the center of the pocket. In a pocket with islands, where different areas 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.

7.

I Penetration step.

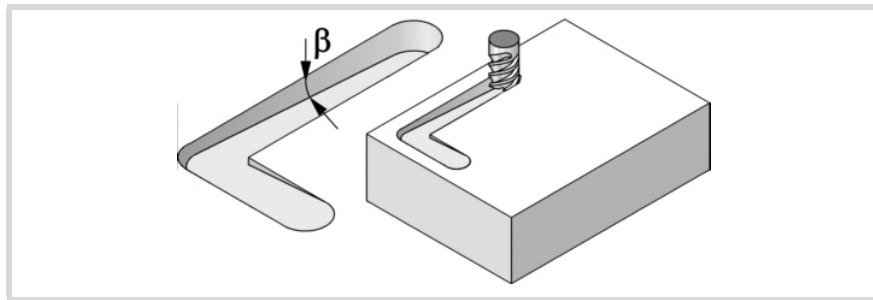
- If programmed with a positive sign (I+), the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If programmed with a negative sign (I-), the pocket is machined with the given pass (step) except the last pass that machines the 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.

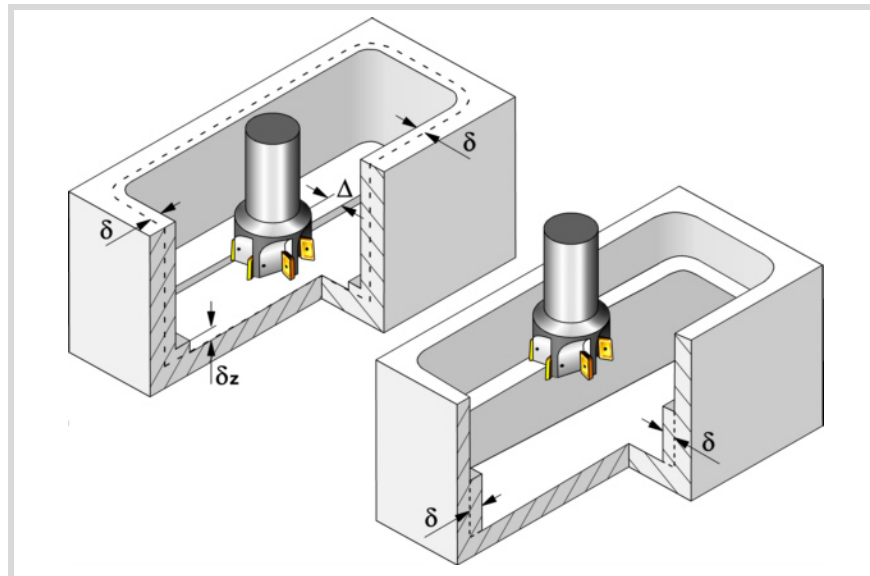


Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.

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

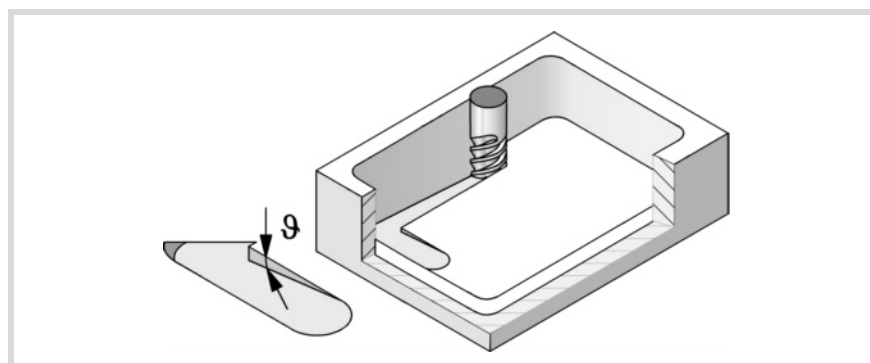
Finishing parameters:

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.

7.

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

2D profile pocket.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

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



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

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

2D profile pocket.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

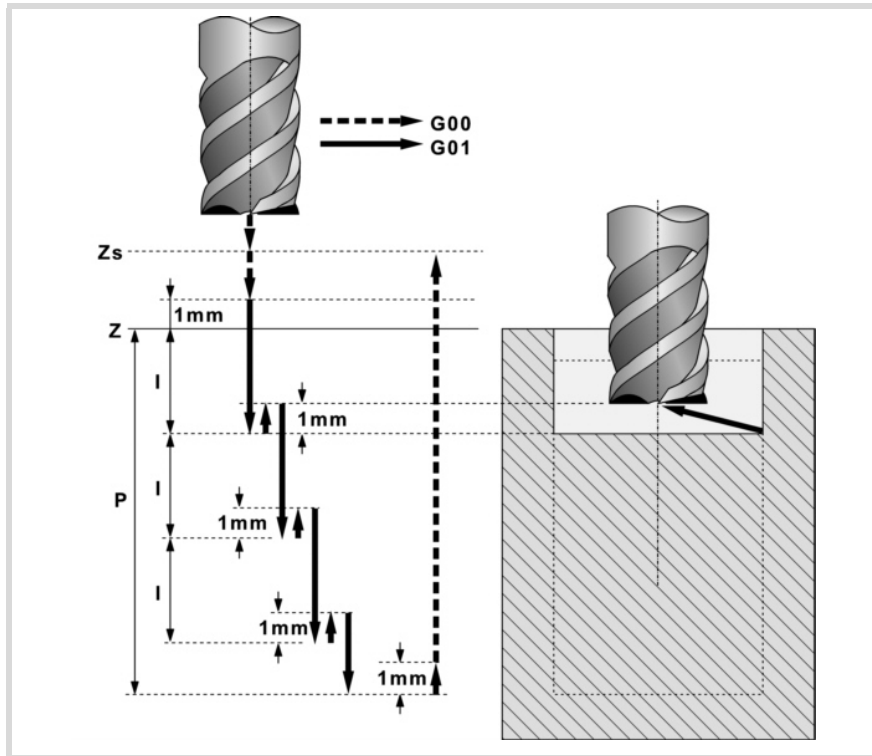


The CNC selects the optimal drilling point, as close as possible to the initial point of the outer contour.



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 "l" 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.

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



CNC 8058
CNC 8060
CNC 8065

REF: 2102

Canned cycles (·M· model).

- 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).

7.

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

2D profile pocket.

FAGOR 

FAGOR AUTOMATION

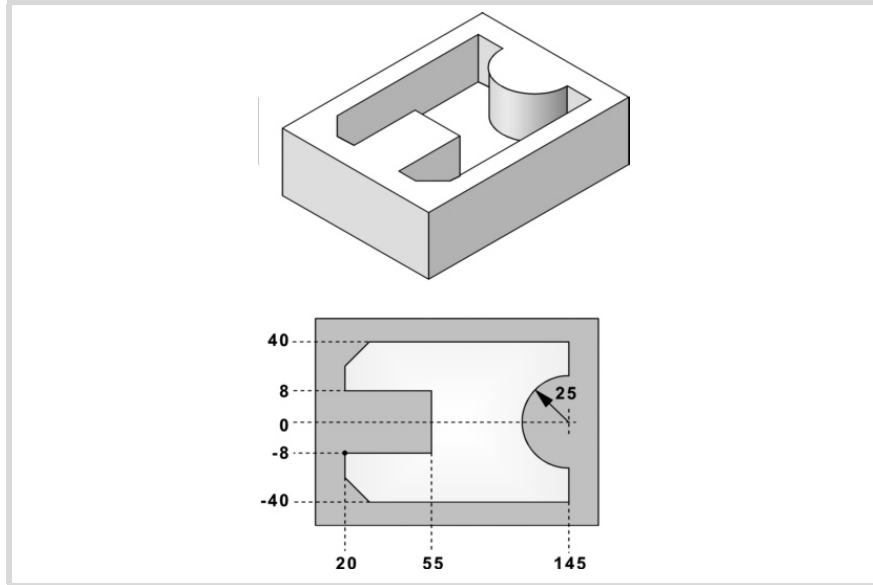
CNC 8058
CNC 8060
CNC 8065

REF: 2102

7.1.3 Examples of how to define 2D profiles

7.

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



| | | |
|--------------|-----------|----------|
| 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] |

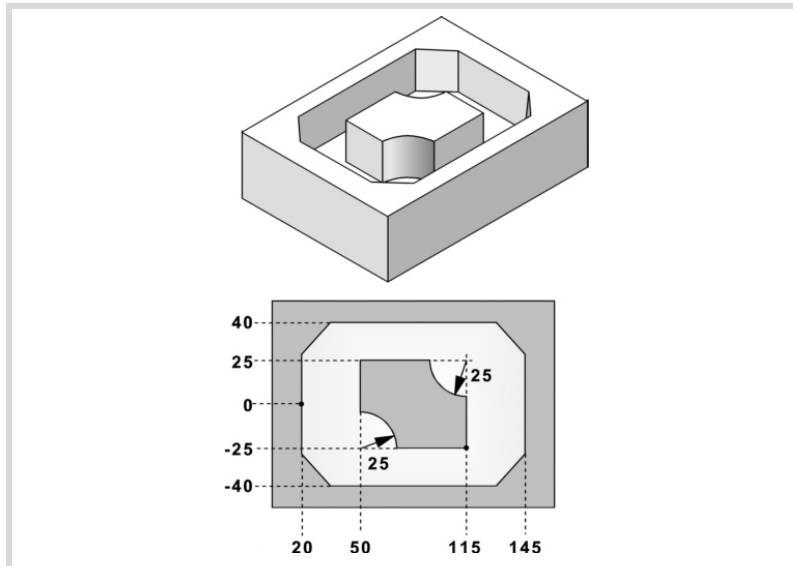
End:

Save profile



CNC 8058
CNC 8060
CNC 8065

REF: 2102



| | | |
|--------------|-----------|----------|
| Profile P.XY | FAGOR 102 | [RECALL] |
|--------------|-----------|----------|

Configuration:

| | |
|-----------------|------------------|
| Abcissa 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] |

7.

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

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.

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

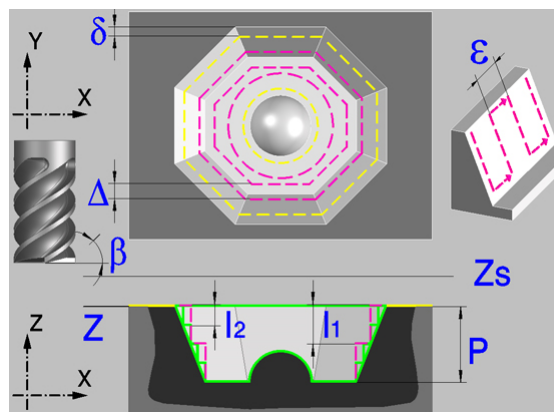


FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

7.2 3D profile pocket with islands.

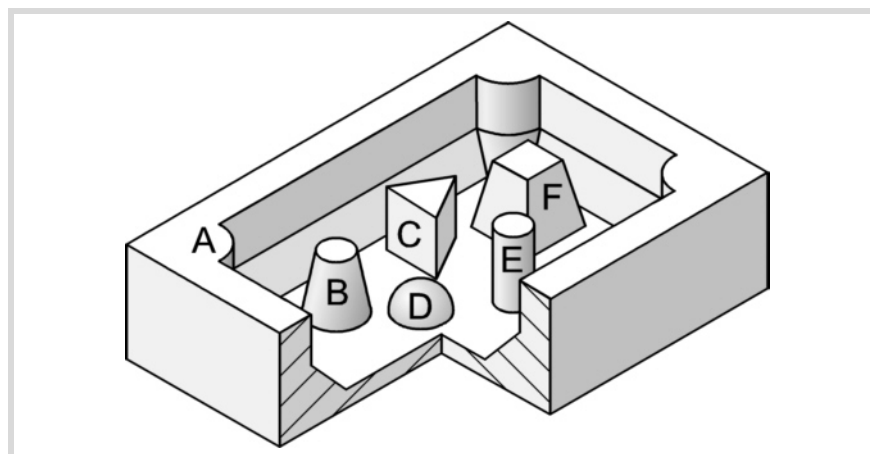


A pocket consists of an outside contour and a number of inside contours called islands.

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).

7.

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

3D profile pocket with islands.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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

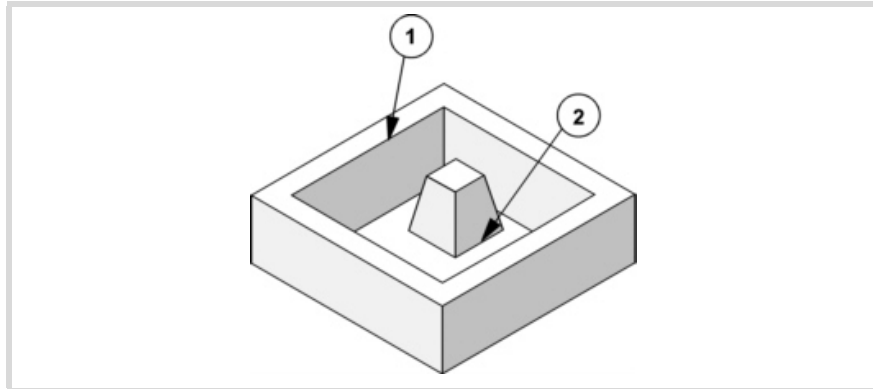
It must indicate all the contours.

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.

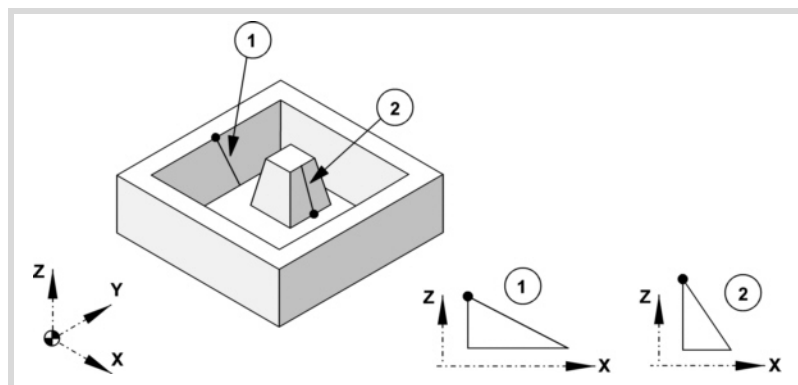
They corresponds to the first 4 contours defined in the surface profile, the number indicates the order.

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 need not be programmed.

The figure shows three programming examples.

7.

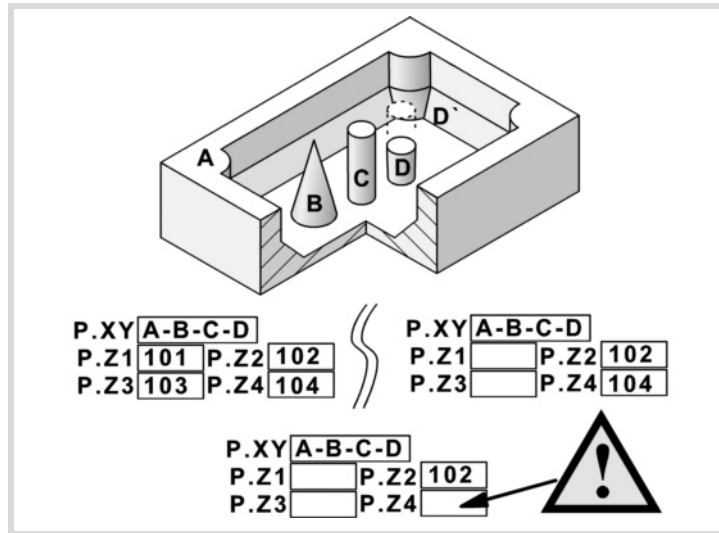
CANNED CYCLES OF THE EDITOR. 2D/3D PROFILE POCKETS.
3D profile pocket with islands.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102



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').

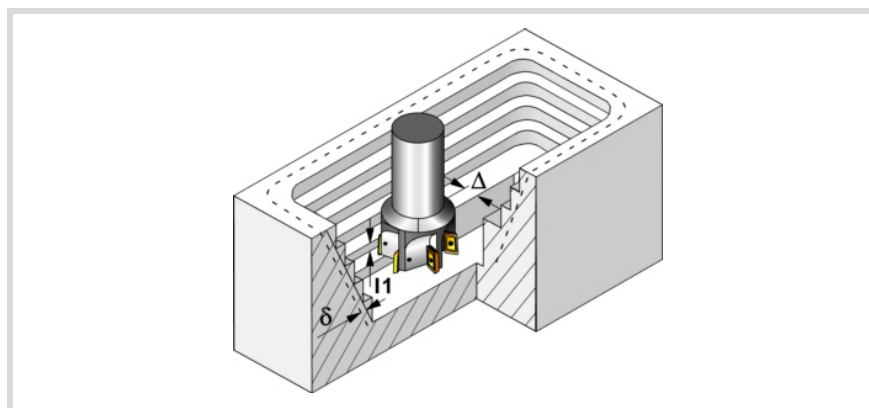
Z Part surface coordinate.

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.

7.

CANNED CYCLES OF THE EDITOR. 2D/3D PROFILE POCKETS. 3D profile pocket with islands.



FAGOR AUTOMATION

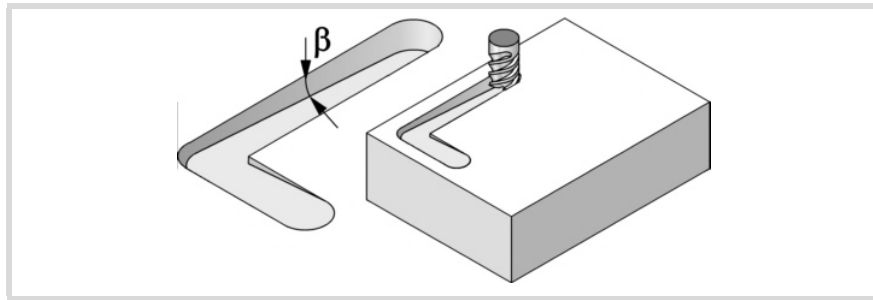
CNC 8058
CNC 8060
CNC 8065

REF: 2102

7.

CANNED CYCLES OF THE EDITOR. 2D/3D PROFILE POCKETS.
3D profile pocket with islands.

- 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.



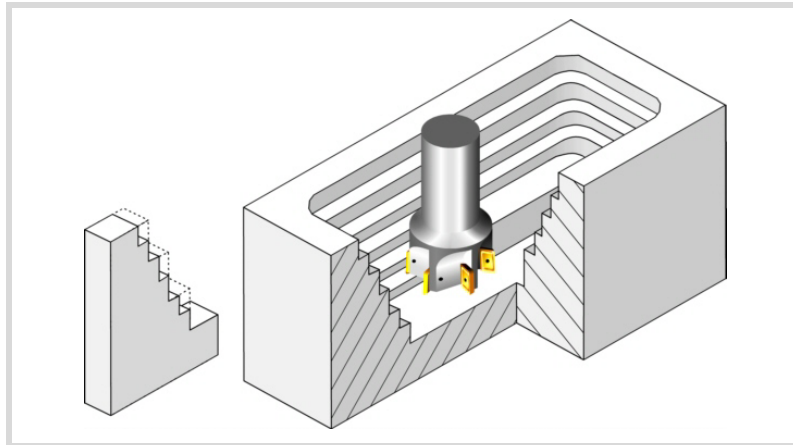
Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.



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.

CANNED CYCLES OF THE EDITOR. 2D/3D PROFILE POCKETS.
3D profile pocket with islands.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

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

3D profile pocket with islands.

FAGOR 

FAGOR AUTOMATION

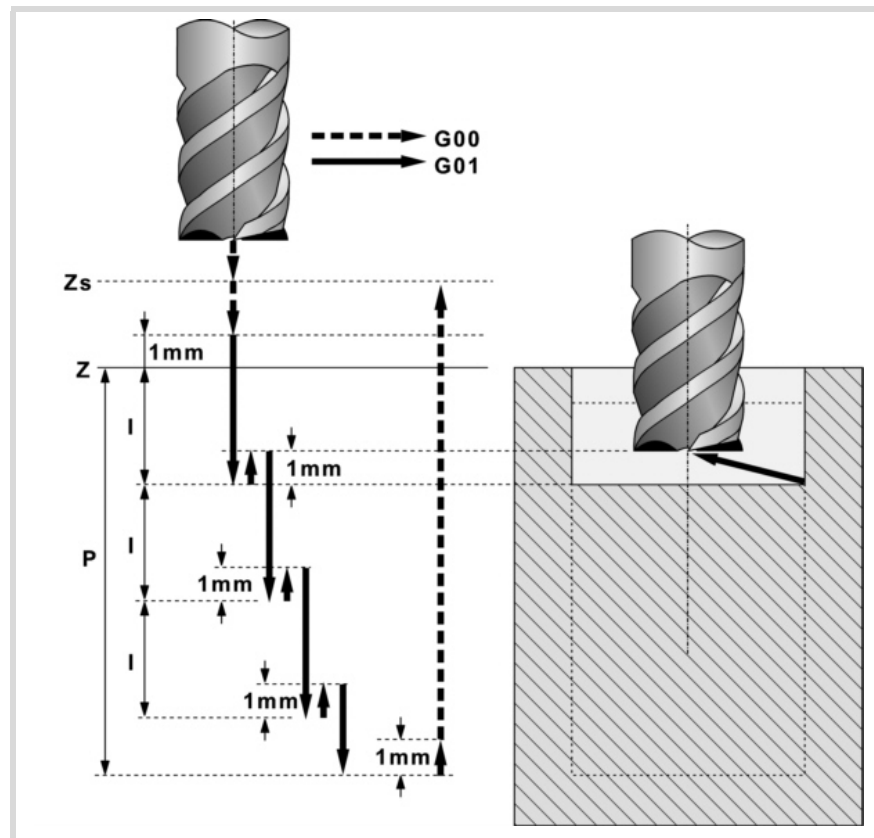
CNC 8058
CNC 8060
CNC 8065

REF: 2102

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 "l1" 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 "l2" 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.

CANNED CYCLES OF THE EDITOR. 2D/3D PROFILE POCKETS.
3D profile pocket with islands.

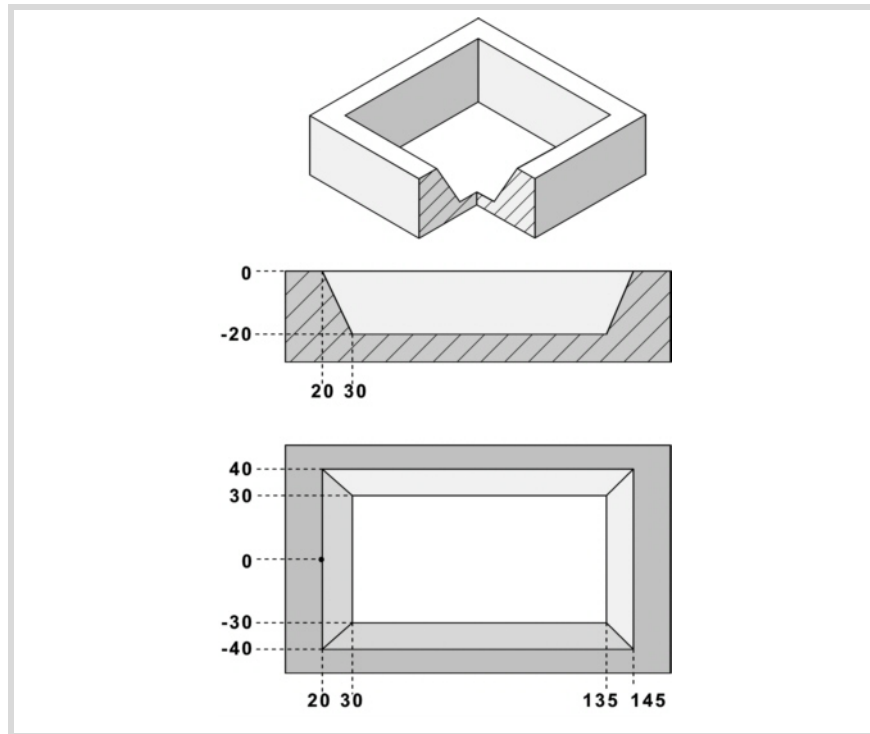


FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

7.2.3 Examples of how to define 3D profiles



| | |
|-------------|---------|
| Pocket P.3D | FAGOR-A |
|-------------|---------|

| | | |
|--------------|-----------|----------|
| Profile P.XY | FAGOR 110 | [RECALL] |
|--------------|-----------|----------|

Configuration:

| | |
|-----------------|------------------|
| Abcissa 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 |
|--------------|-----------|--------|

7.

CANNED CYCLES OF THE EDITOR. 2D/3D PROFILE POCKETS.
3D profile pocket with islands.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

Configuration:

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

Profile (depth profile):

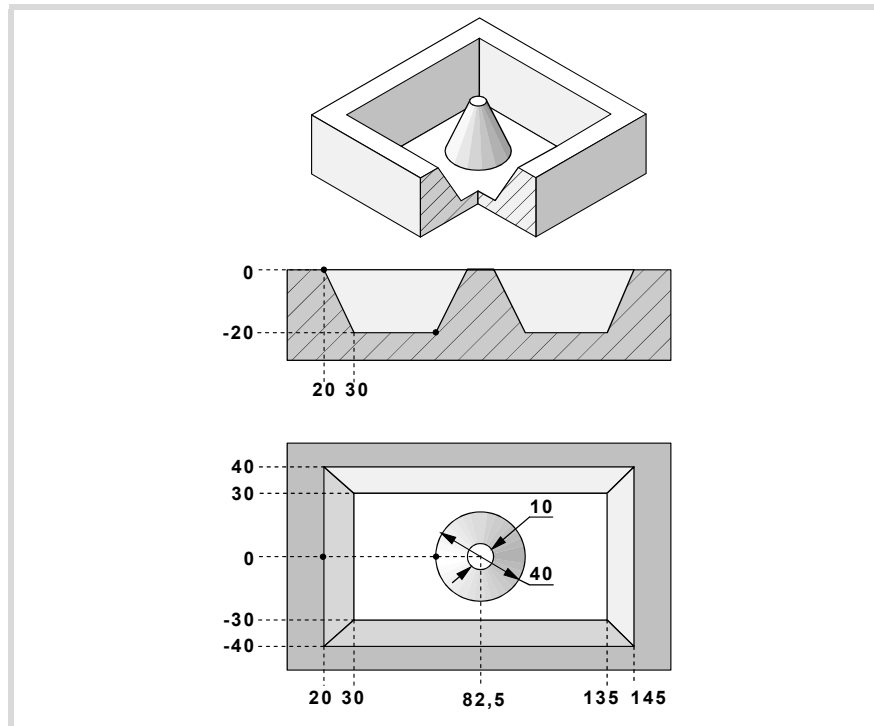
| | | | |
|----------------|------|------|----------|
| Starting point | X 20 | Z0 | Validate |
| Straight | X 30 | Z-20 | Validate |

End:

Save profile

7.

CANNED CYCLES OF THE EDITOR. 2D/3D PROFILE POCKETS.
3D profile pocket with islands.



| | |
|-------------|---------|
| 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 | Y 0 | 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 | Z 0 | 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 | Z 0 | Validate |

End:

Save profile

7.

CANNED CYCLES OF THE EDITOR. 2D/3D PROFILE POCKETS.
3D profile pocket with islands.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

7.

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

3D profile pocket with islands.



FAGOR AUTOMATION

CNC 8058

CNC 8060

CNC 8065

REF: 2102

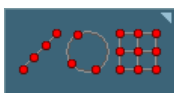
CANNED CYCLES OF THE EDITOR. ROUGHING.

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.



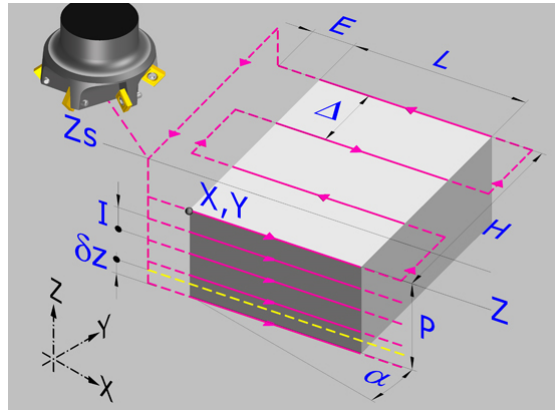
Multiple machining may be added to pocket cycles so the cycle is repeated in several points. The available multiple machining operations are the following. See chapter "[9 Canned cycles of the editor. Multiple positioning.](#)".

- 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.

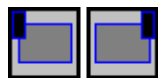


Spiral machining along the abscissa axis.

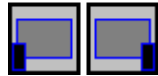


Spiral machining along the ordinate axis.

Corner where the surface milling begins (icon).



Any of the 4 corners may be selected.

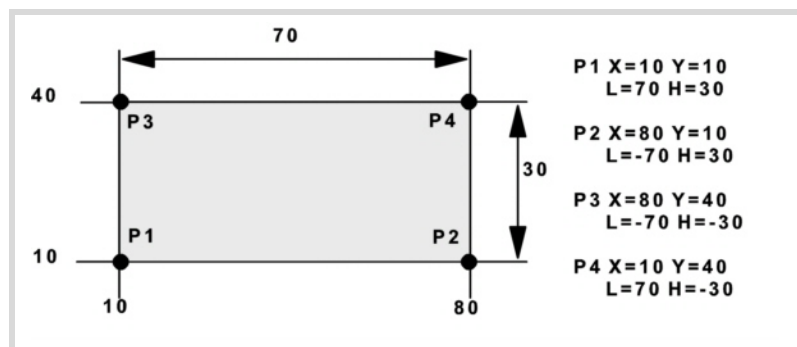


X Y Corner to begin machining.

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.



8.

CANNED CYCLES OF THE EDITOR. ROUGHING. Surface milling.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

Canned cycles (·M· model).

| | |
|----------|---|
| 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. <ul style="list-style-type: none">• If programmed with a positive sign (+), 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 (-), 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. |

8.

CANNED CYCLES OF THE EDITOR: ROUGHING.
Surface milling.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

CANNED CYCLES OF THE EDITOR. ROUGHING.
Surface milling.



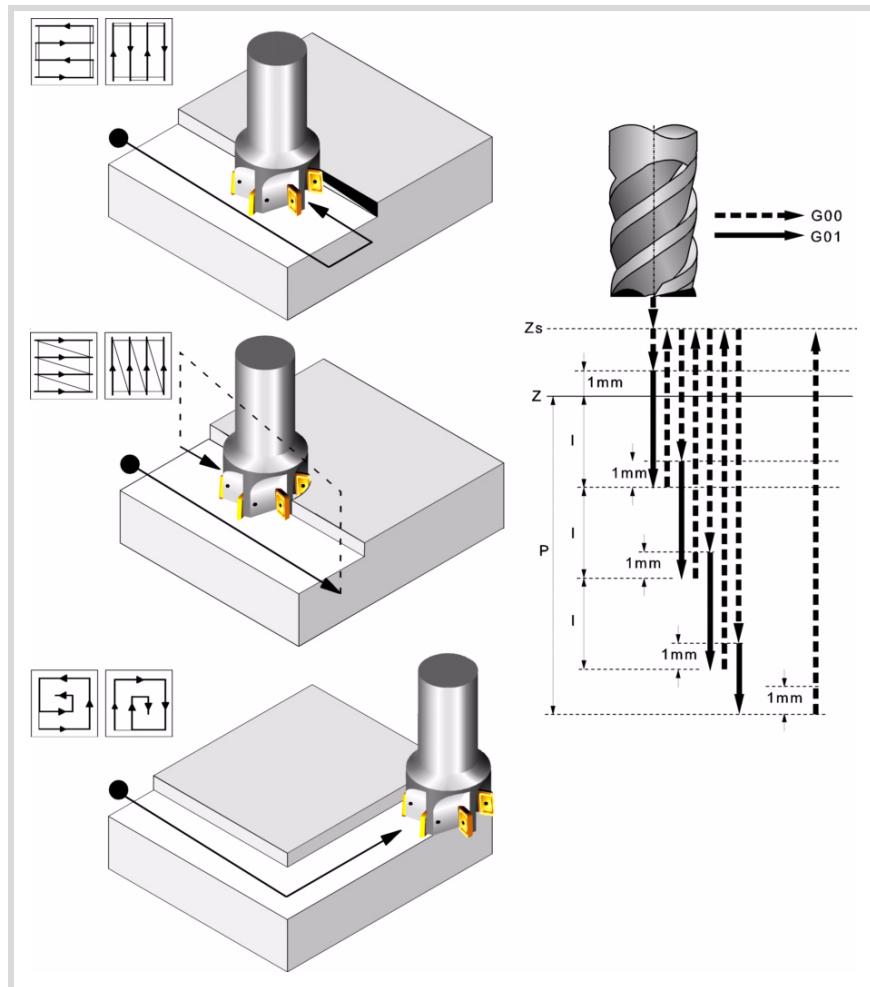
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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 "l" 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.

CANNED CYCLES OF THE EDITOR. ROUGHING.
Surface milling.

FAGOR 

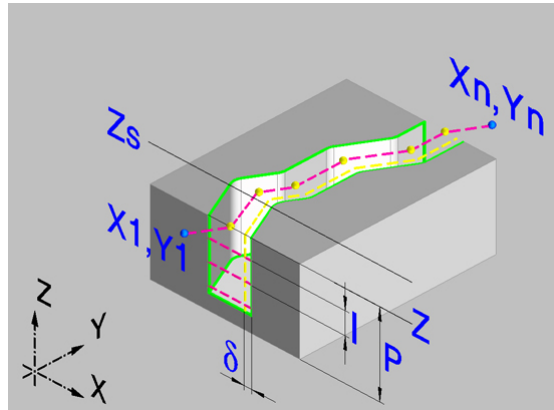
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

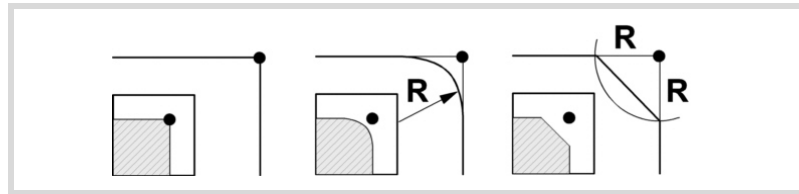


Geometric parameters:

- 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.

8.

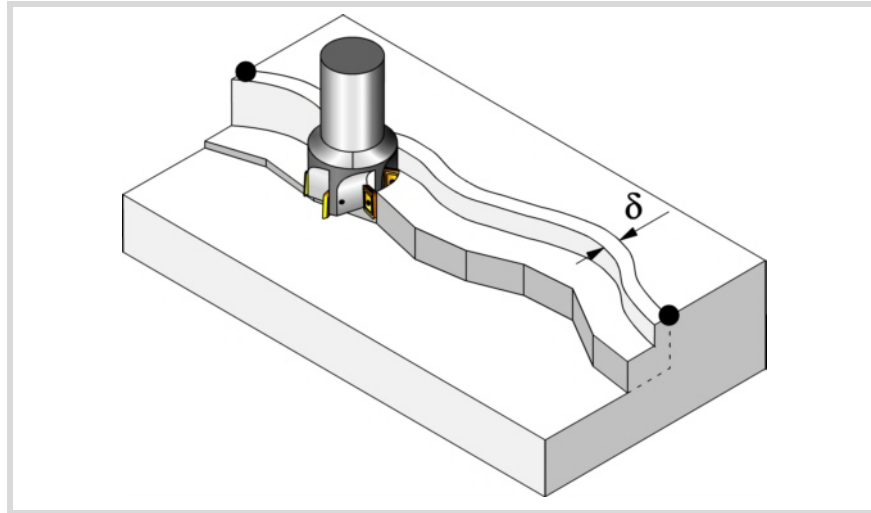
CANNED CYCLES OF THE EDITOR. ROUGHING.
Point-to-point profile milling.



FAGOR AUTOMATION

CNC 8058
 CNC 8060
 CNC 8065

REF: 2102



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 (+), 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 (-), 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.



CANNED CYCLES OF THE EDITOR. ROUGHING.

Point-to-point profile milling.



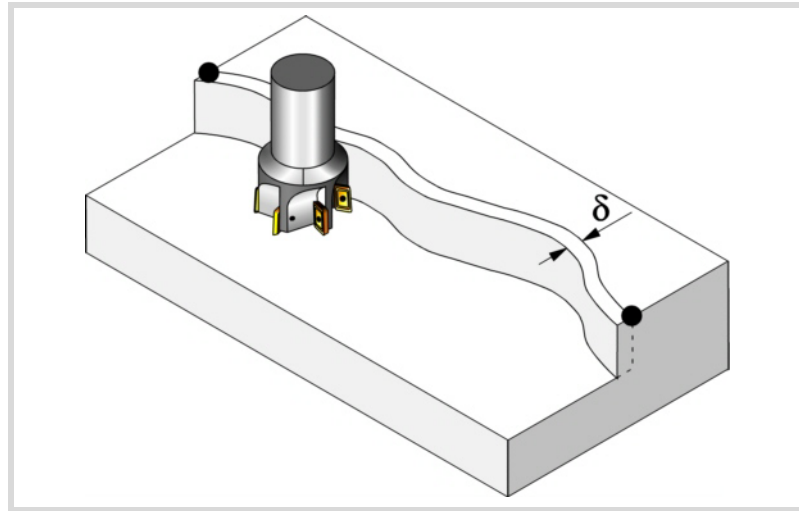
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

CANNED CYCLES OF THE EDITOR. ROUGHING.
Point-to-point profile milling.

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).



CANNED CYCLES OF THE EDITOR. ROUGHING.

Point-to-point profile milling.



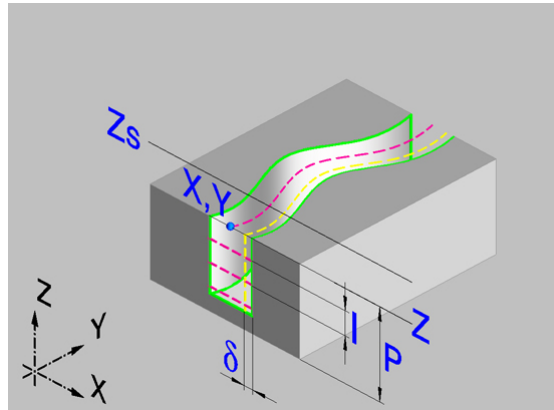
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

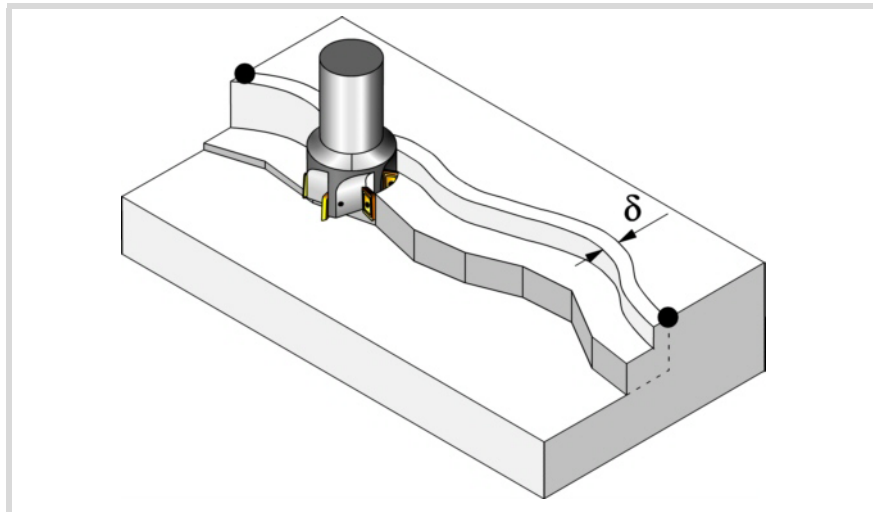
Z Part surface coordinate.

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.

8.

CANNED CYCLES OF THE EDITOR. ROUGHING.
Free profile milling.

FAGOR 
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

I Penetration step.

- If programmed with a positive sign (+), 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 (-), 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.



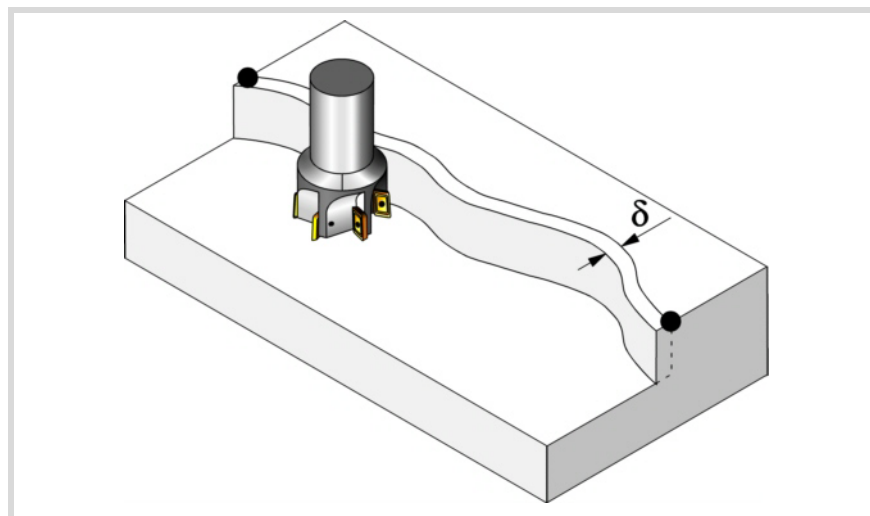
Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.



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).



CANNED CYCLES OF THE EDITOR. ROUGHING.

Free profile milling.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

CANNED CYCLES OF THE EDITOR. ROUGHING.
Free profile milling.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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 "l" 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).



CANNED CYCLES OF THE EDITOR. ROUGHING.

Free profile milling.



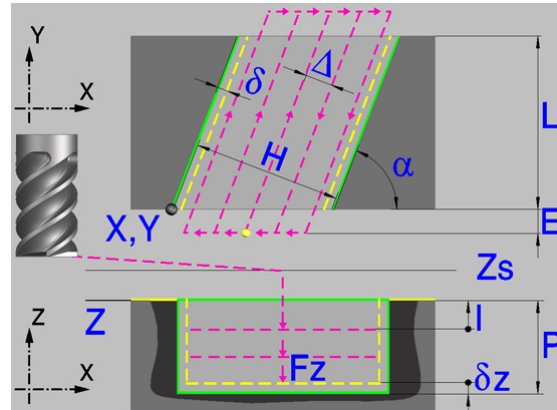
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

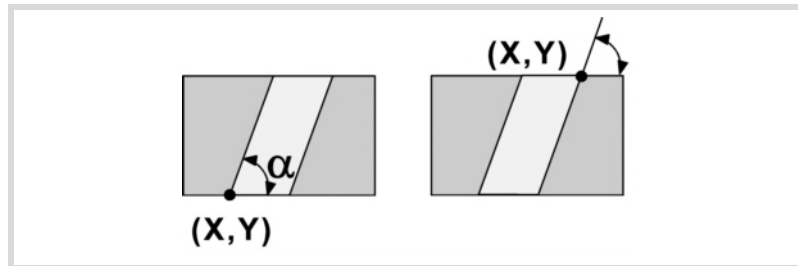
The sign indicates the orientation referred to the XY point.

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.

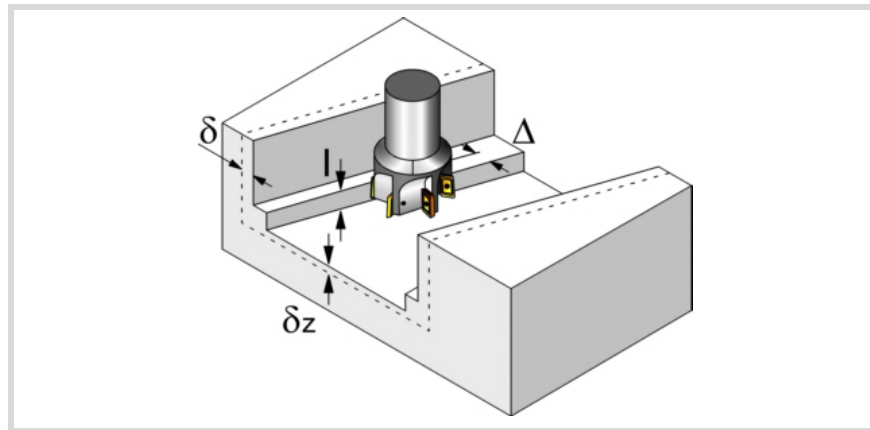
8.

CANNED CYCLES OF THE EDITOR. ROUGHING.
Slot milling.

FAGOR 
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102



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 (+), 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 (-), 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.

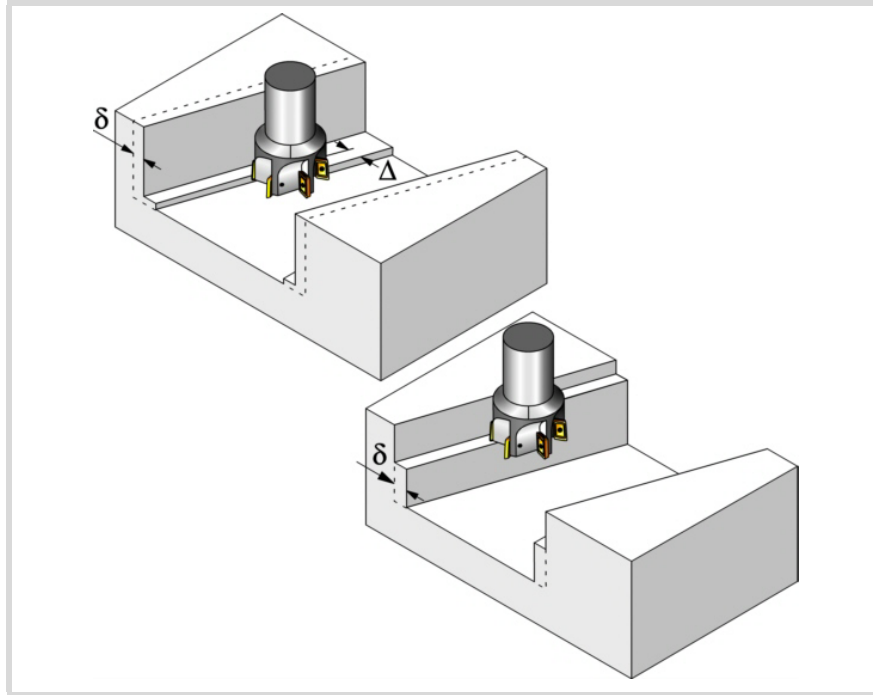


Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.

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.

8.

CANNED CYCLES OF THE EDITOR. ROUGHING. Slot milling.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

Canned cycles (·M· model).

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.

CANNED CYCLES OF THE EDITOR. ROUGHING.

Slot milling.

FAGOR 

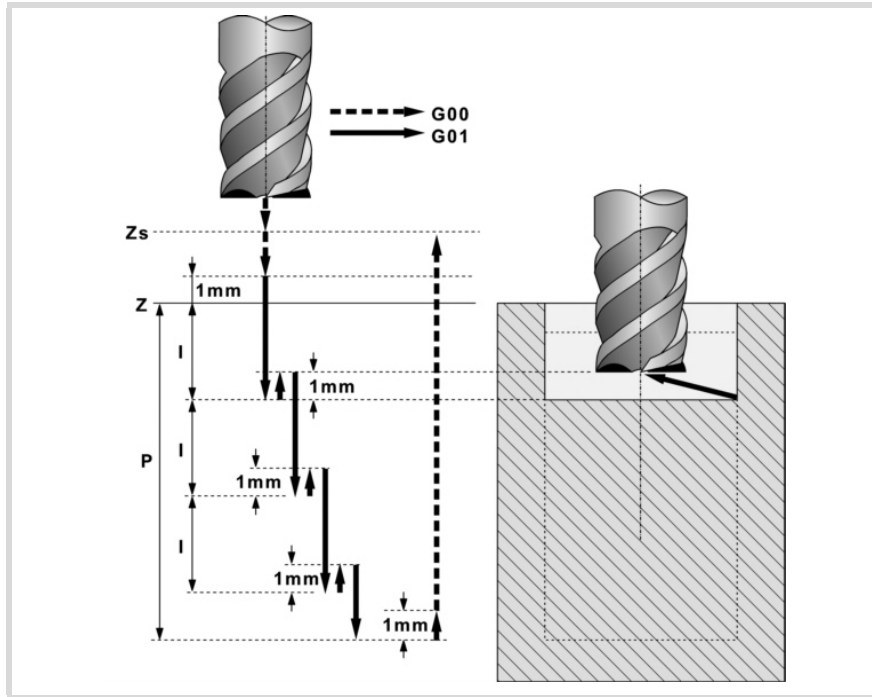
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

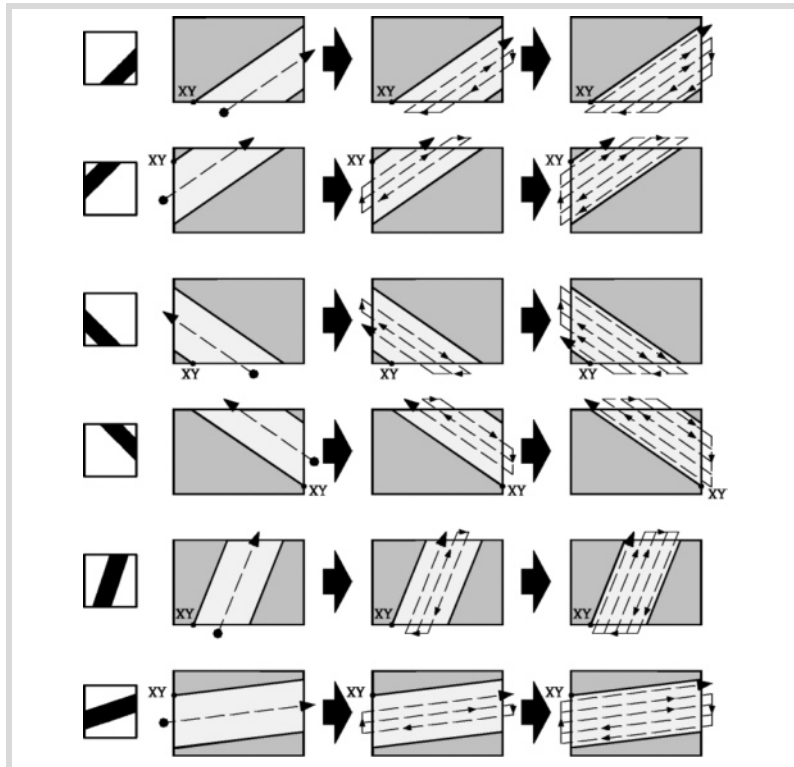
REF: 2102

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.

8.

CANNED CYCLES OF THE EDITOR. ROUGHING. Slot milling.

Canned cycles (·M· model).

- 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

CNC 8058
CNC 8060
CNC 8065

REF: 2102

8.

CANNED CYCLES OF THE EDITOR. ROUGHING.

Slot milling.



FAGOR AUTOMATION

CNC 8058

CNC 8060

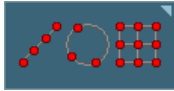
CNC 8065

REF: 2102

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.

FAGOR 

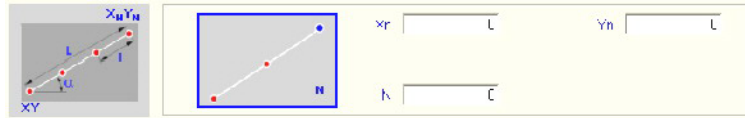
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

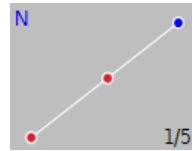
REF: 2102

9.1 Multiple machining in straight line.

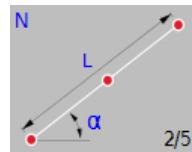
There are 5 different ways to define the machining operation. Use the icon to select the desired option.



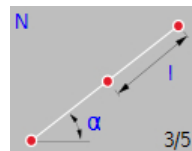
Positioning options.



End point and number of machining operations.



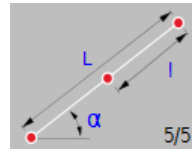
Length, angle and number of machining operations.



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.

X_n, Y_n 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.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.
Multiple machining in straight line.



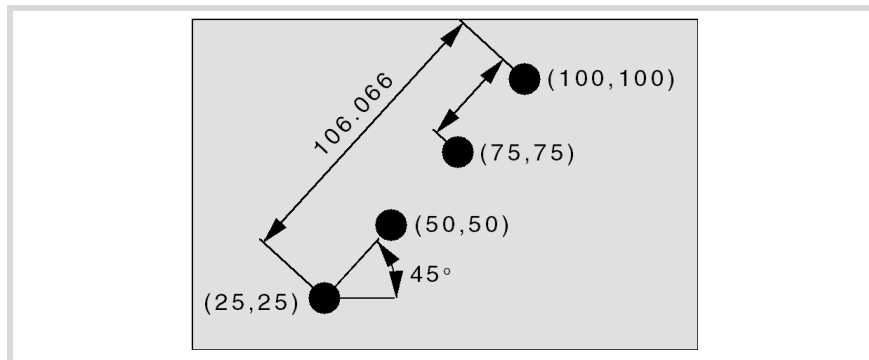
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.
Multiple machining in straight line.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

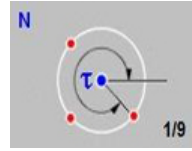
REF: 2102

9.2 Multiple machining in arc pattern.

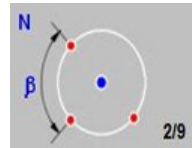
There are 9 ways to define the machining operation. Use the icon to select the desired option.



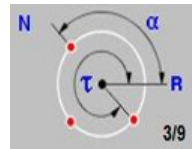
Positioning options.



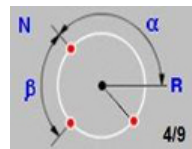
Arc center coordinates, number of machining operations and final angle of the arc.



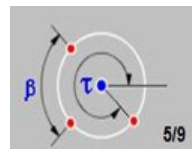
Arc center coordinates, number of machining operations and angular distance between two machining operations.



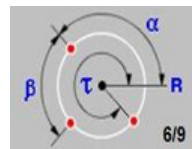
Arc radius, number of machining operations and starting and final angles of the arc.



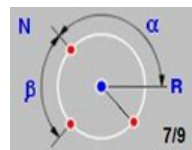
Arc radius, number of machining operations, starting angle of the arc and angular distance between two machining operations.



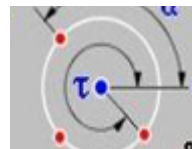
Arc center coordinates, final angle of the arc and angular distance between two machining operations.



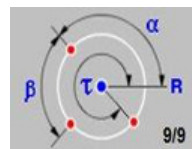
Arc radius, angular distance between two machining operations and starting and final angles of the arc.



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.

9.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.
Multiple machining in arc pattern.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

Canned cycles (·M· model).

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.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.

Multiple machining in arc pattern.

FAGOR 

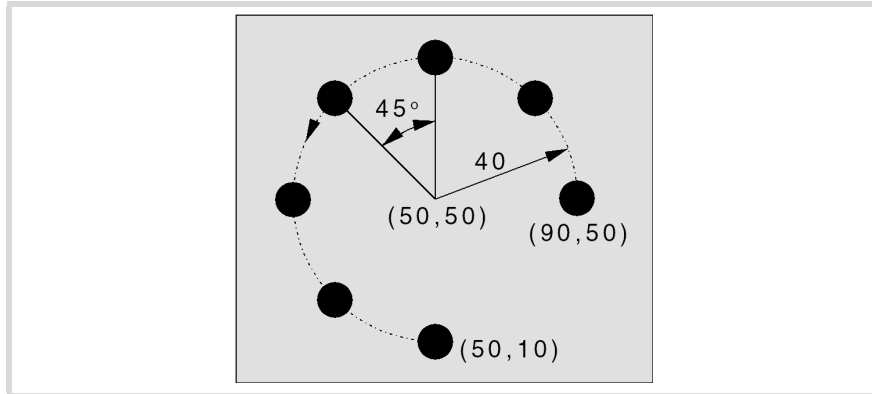
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.
Multiple machining in arc pattern.



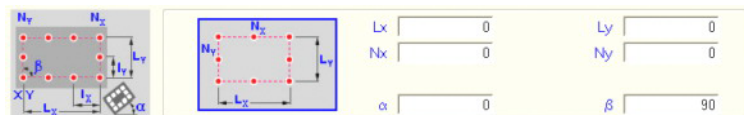
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

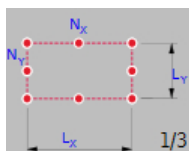
REF: 2102

9.3 Multiple machining in rectangular pattern.

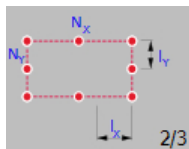
There are 3 ways to define the machining operation. Use the icon to select the desired option.



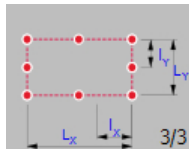
Positioning options.



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.
- lx, ly 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 (lx, ly).

9.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.
Multiple machining in rectangular pattern.



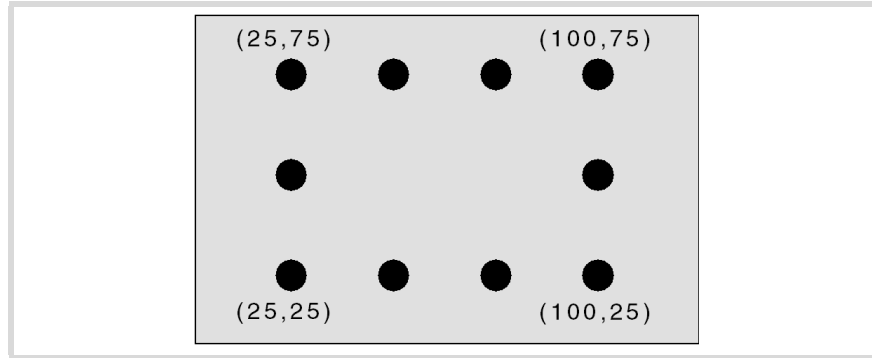
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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. | lx 25, ly 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. | lx 25, ly 25 |
| | Angle with respect to the abscissa axis. | α 0 |
| | Angle between paths. | β 90 |

9.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.
Multiple machining in rectangular pattern.



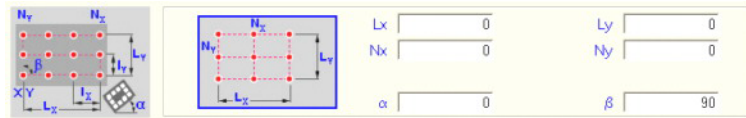
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

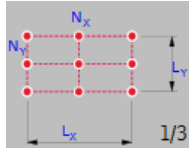
REF: 2102

9.4 Multiple machining in grid pattern.

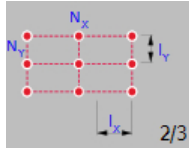
There are 3 ways to define the machining operation. Use the icon to select the desired option.



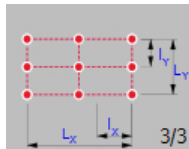
Positioning options.



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.
- lx, ly 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 (lx, ly).

9.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.
Multiple machining in grid pattern.

FAGOR 

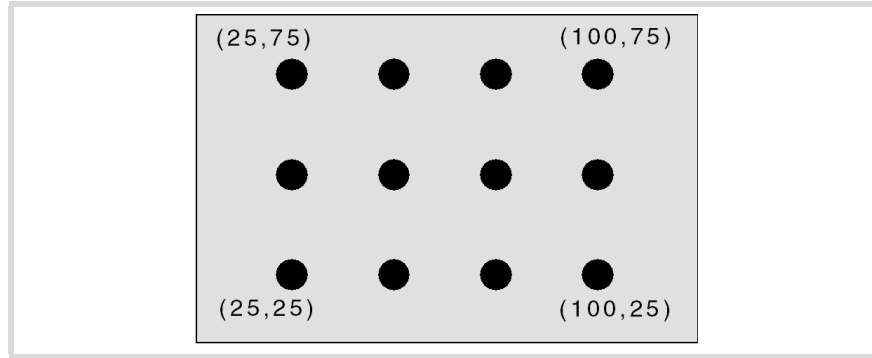
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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. | lx 25, ly 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. | lx 25, ly 25 |
| | Angle with respect to the abscissa axis. | α 0 |
| | Angle between paths. | β 90 |

9.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.
Multiple machining in grid pattern.

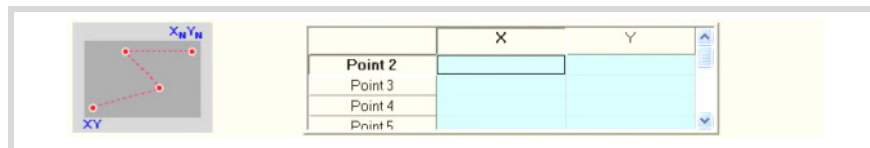


FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

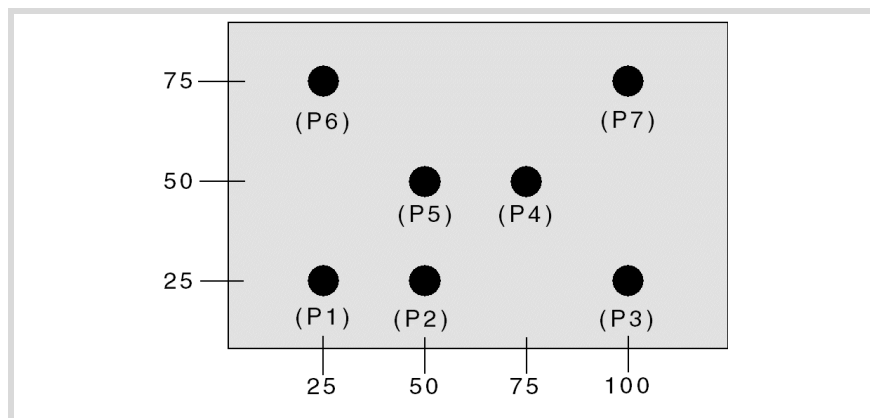
9.5 Random multiple machining.



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 |

9.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.
Random multiple machining.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

9.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.

Random multiple machining.



FAGOR AUTOMATION

CNC 8058

CNC 8060

CNC 8065

REF: 2102

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.



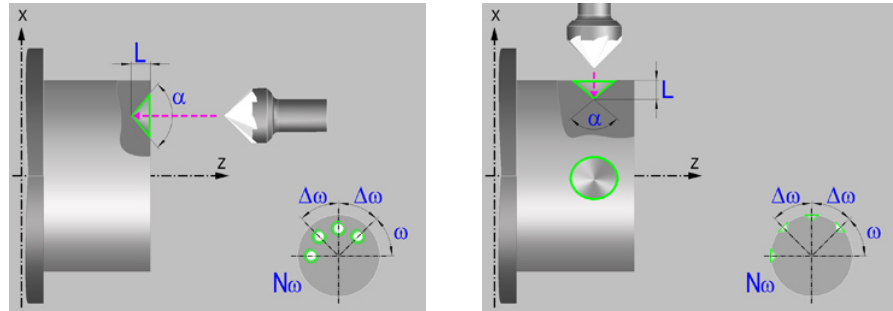
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

10.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Multiple center-punching.

FAGOR 
FAGOR AUTOMATION

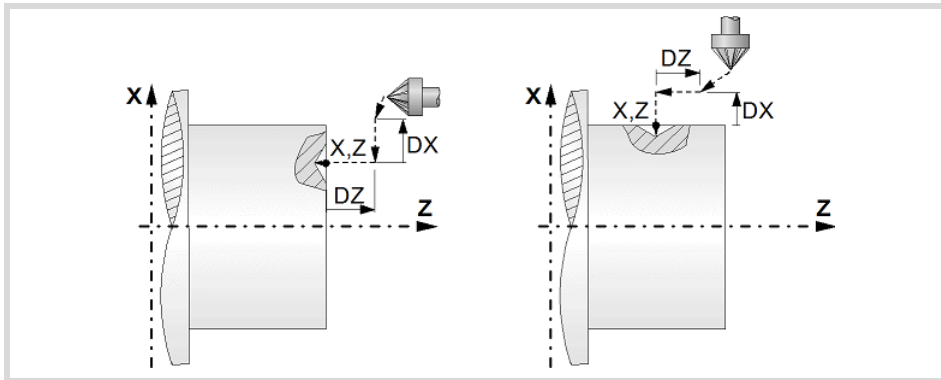
CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

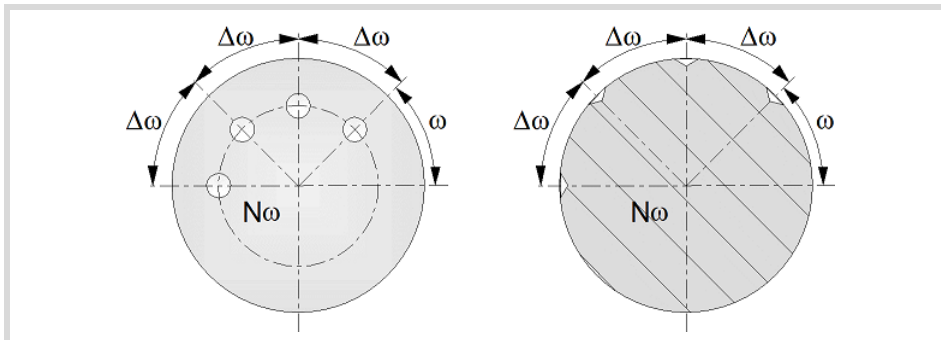
ϕ Center-punching diameter.

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.



10.

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.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.

Multiple center-punching.

FAGOR 

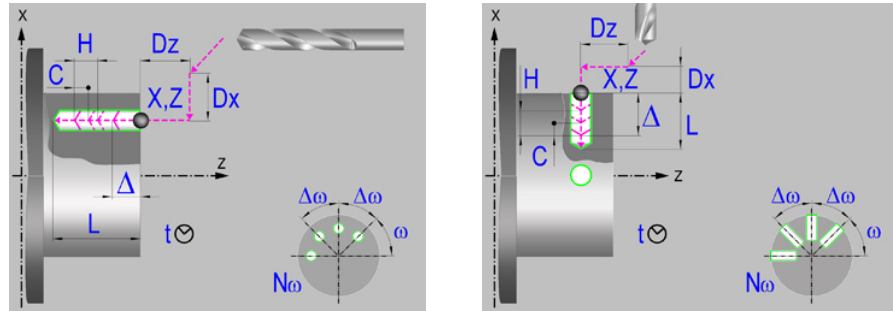
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

10.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Multiple drilling.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

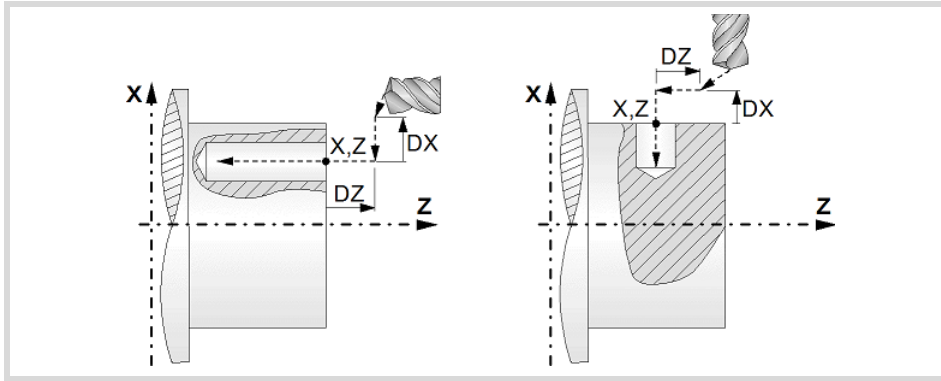
REF: 2102

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.

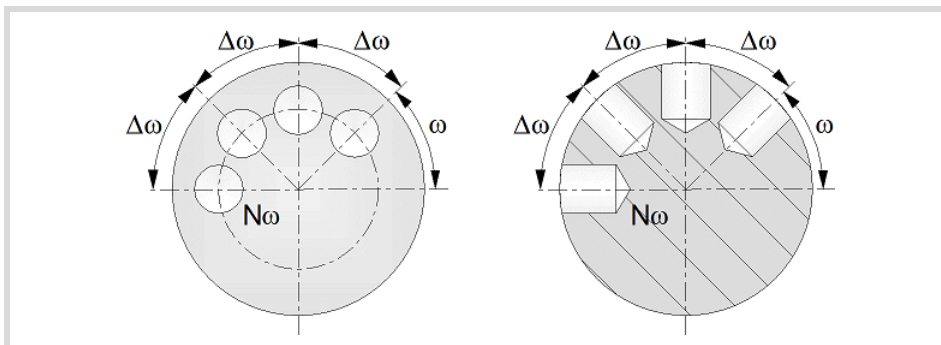


Programming of the drilling operation.

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.
- $\Delta\omega$ 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.

10.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Multiple drilling.

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

- Δ Maximum drilling peck (step).
- $K\Delta$ 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 " $K\Delta$ D", the third one " $K\Delta$ ($K\Delta$ 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 $K\Delta$.

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.

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.

10.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.

Multiple drilling.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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 Δ defines the drilling peck and $K\Delta$ the reduction factor for this peck. The first drilling peck will be " Δ ", the second one " $K\Delta$ ", the third one " $K\Delta (K\Delta D)$ " 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 $K\Delta$.

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. Then, 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.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Multiple drilling.



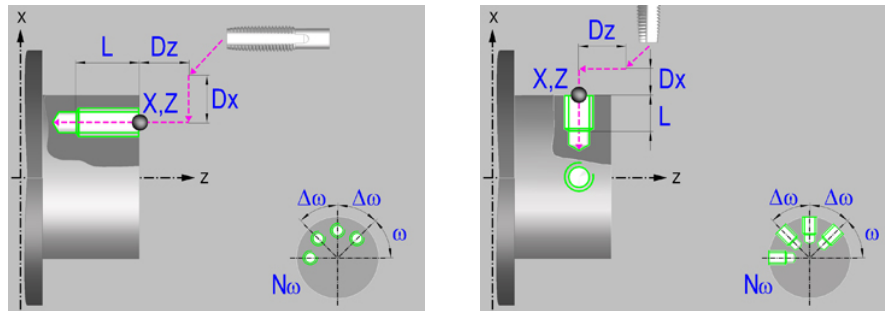
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

10.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Multiple tapping.

FAGOR

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

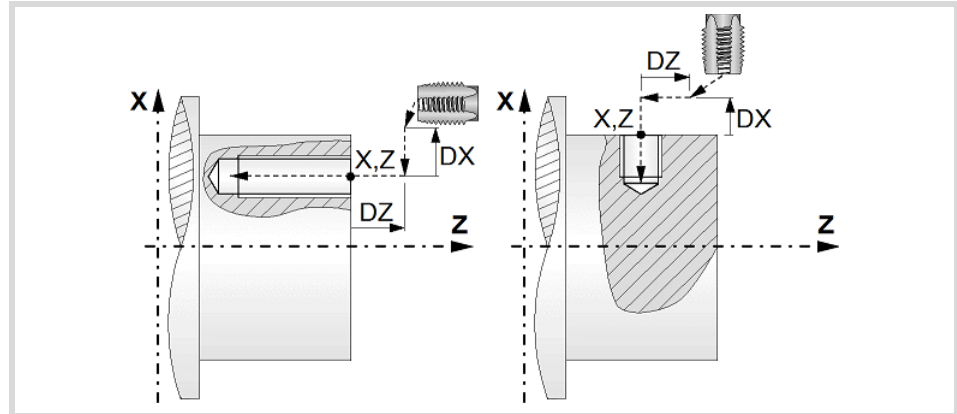
REF: 2102

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.

D_x, D_z 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.



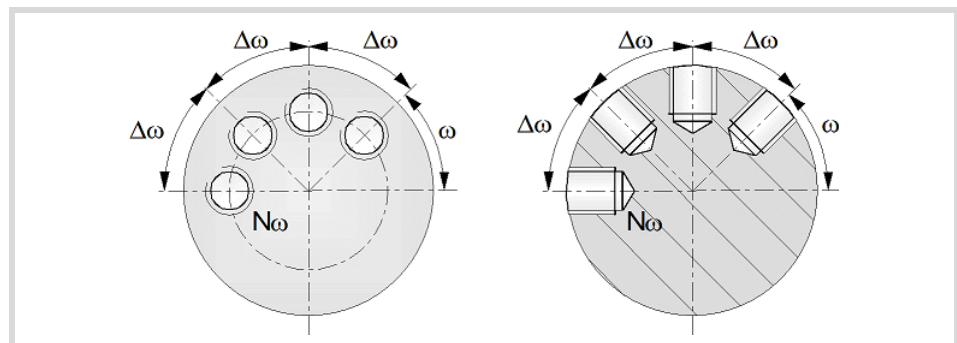
Rigid tapping. The spindle must have a servo-motor-drive system and an encoder.

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



10.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Multiple tapping.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Multiple tapping.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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 (I_o).
- 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.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Multiple tapping.



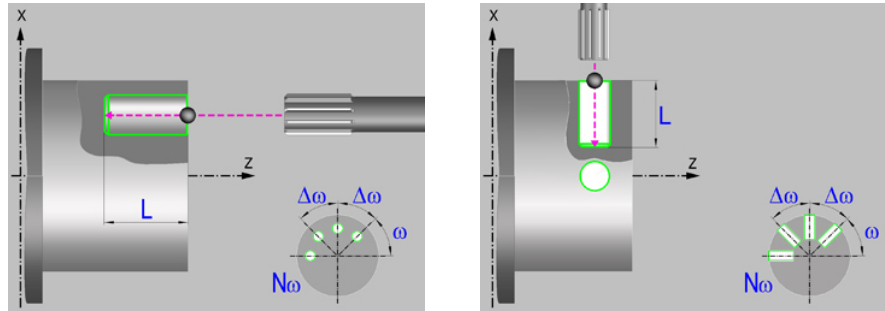
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

10.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Multiple reaming.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

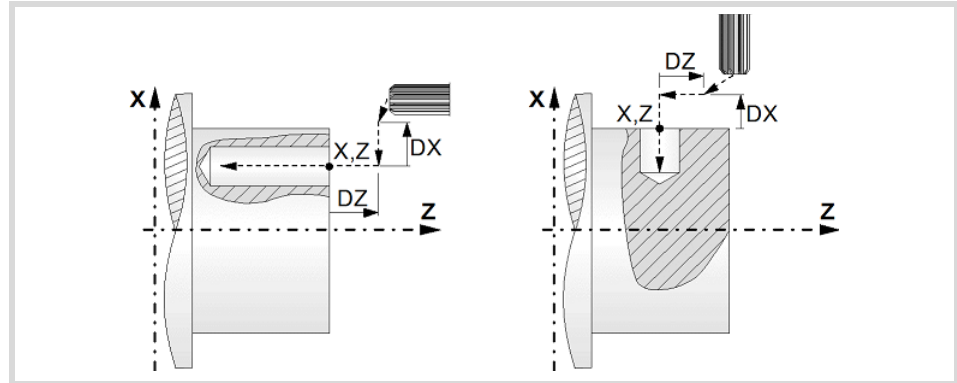
REF: 2102

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.

D_x, D_z Safety distance referred to the starting point.

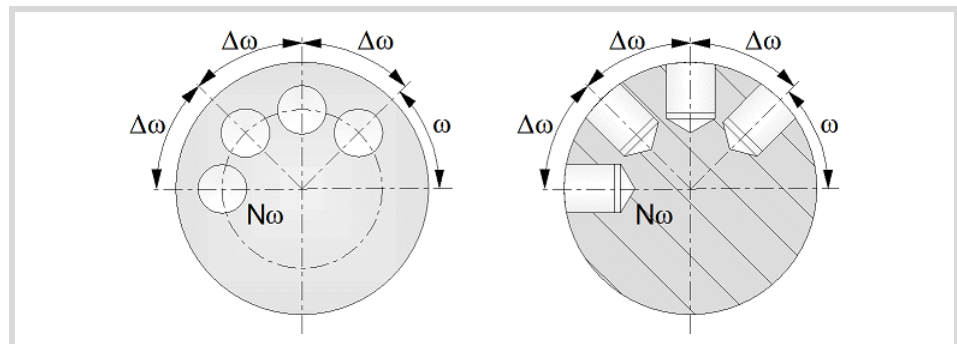
- Machining on the face of the part. The value of the safety distance on X is always defined in radius and it may be either positive or negative. The value of the safety distance in Z is always assumed with positive values.
- Machining on the side of the part. The value of the safety distance on X is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

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Multiple reaming.



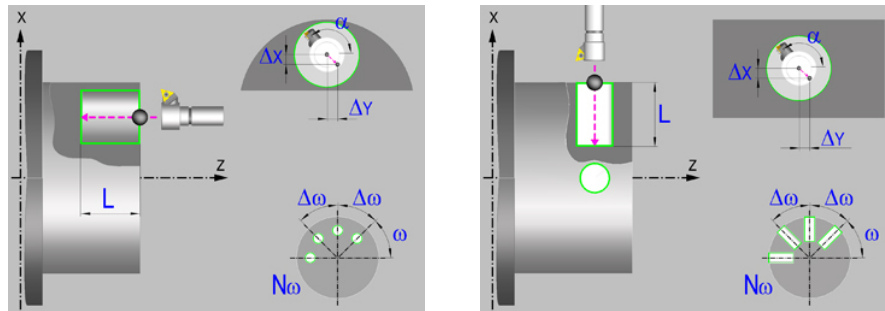
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

10.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Multiple boring.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

Coordinates of the end point.

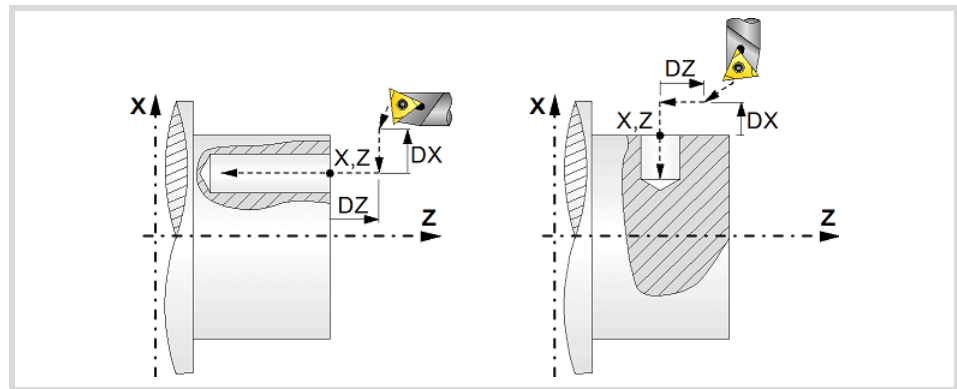
Xf, Yf, Zf Coordinate to withdrawal the tool after the cycle has completed; if not programmed, there is withdrawal. If the cycle is machining in multiple mode, the tool will only be withdrawn after the last cycle.

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.

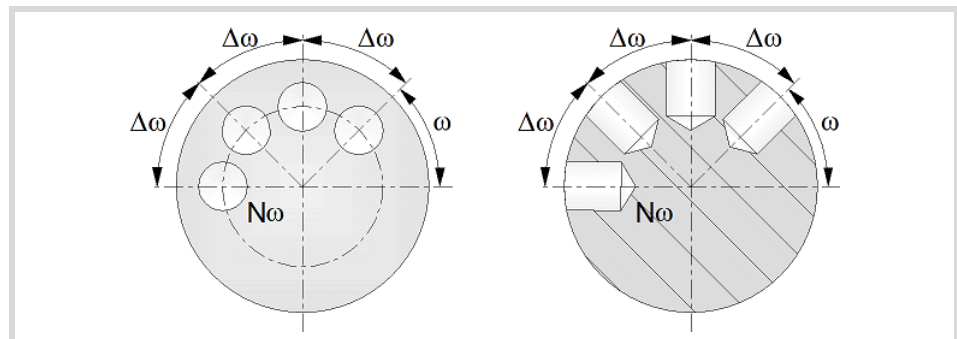
- Machining on the face of the part. The value of the safety distance on X is always defined in radius and it may be either positive or negative. The value of the safety distance in Z is always assumed with positive values.
- Machining on the side of the part. The value of the safety distance on X is 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.
- $\Delta\omega$ Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.
- $N\omega$ Total number of operations.



10.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Multiple boring.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

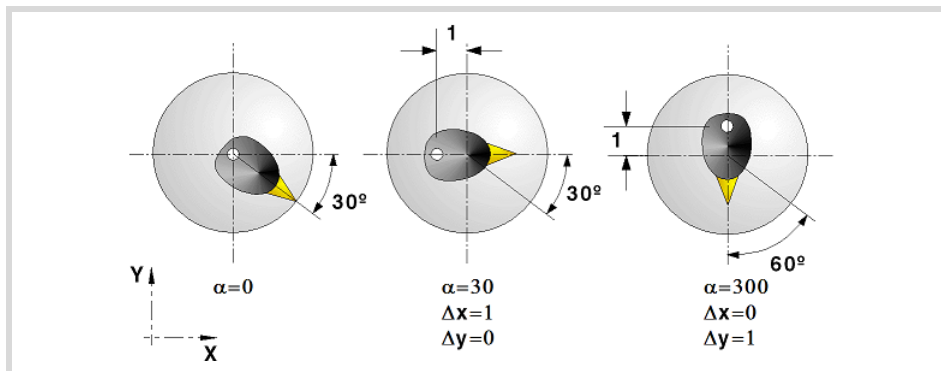


Rapid withdrawal with spindle orientation and the tool is not touching the surface. This type of withdrawal is not available for XZ positioning, nor for C axis positioning on the cylindrical side of the part.

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.

10.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Multiple boring.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

10.

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.

10.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Multiple boring.



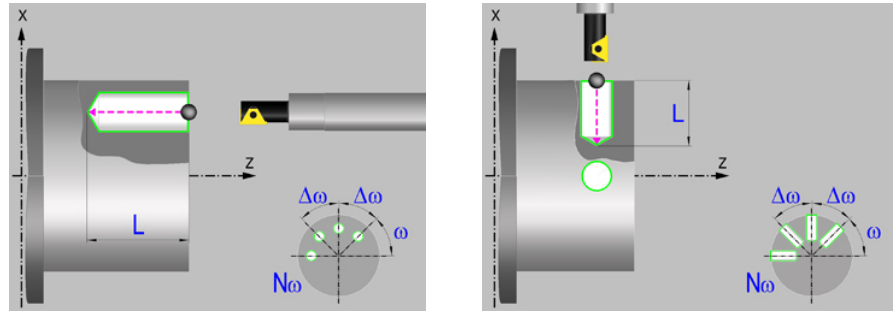
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

10.

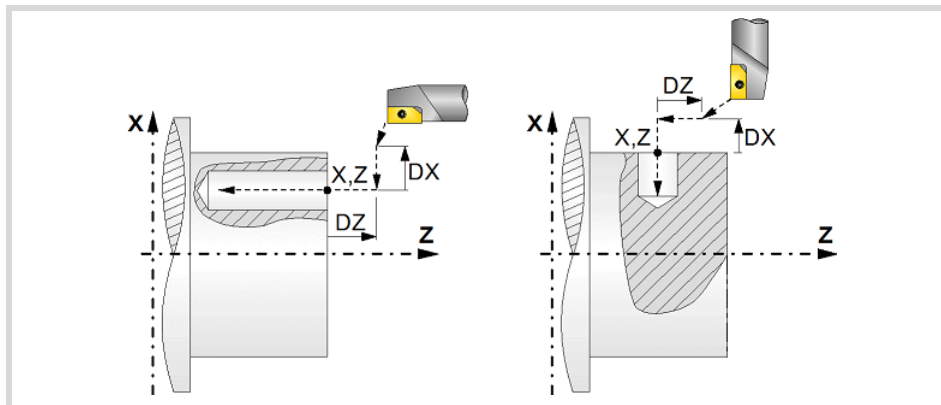
CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Multiple bore milling.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102



10.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Multiple bore milling.

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.

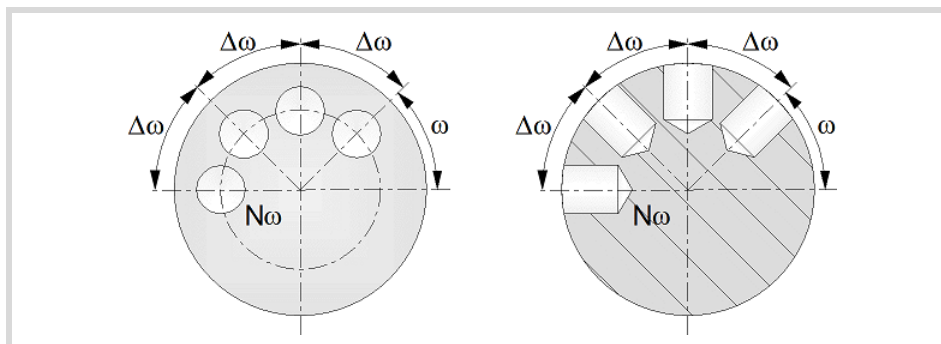
When starting from a hole that was machined earlier, parameter **Øk** defines the diameter of that hole; if not programmed or programmed with a 0 value, the cycle assumes that there is no previous hole.

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.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

10.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Multiple bore milling.

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.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.

Multiple bore milling.

FAGOR 

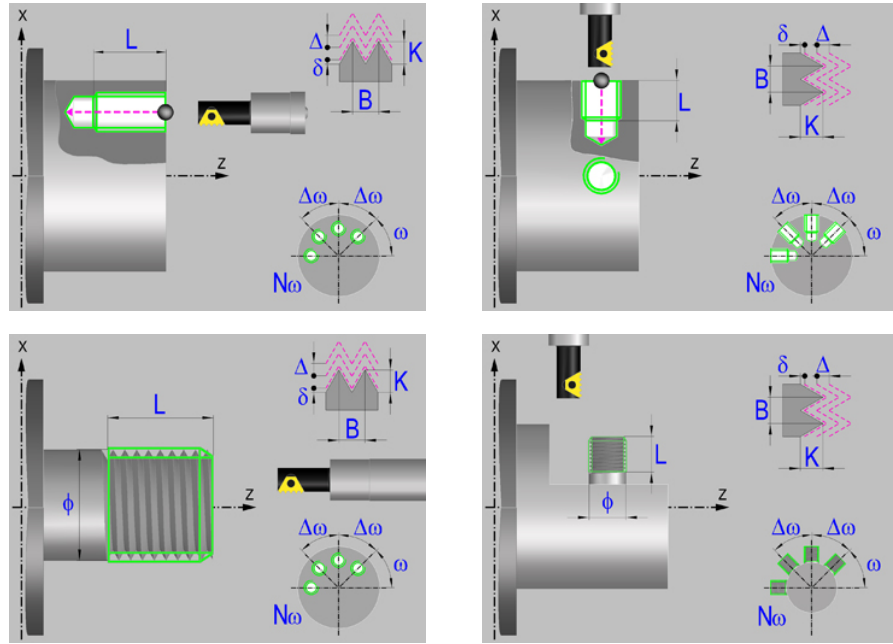
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

10.7 Multiple thread milling.

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



10.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Multiple thread milling.

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.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

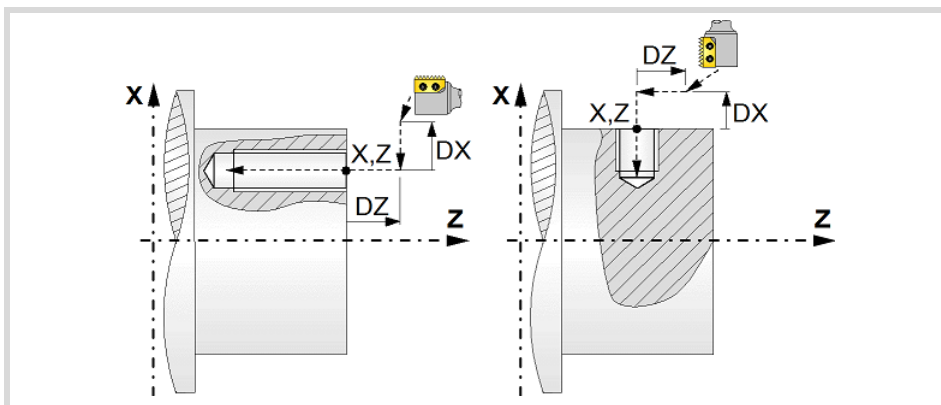
REF: 2102

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.

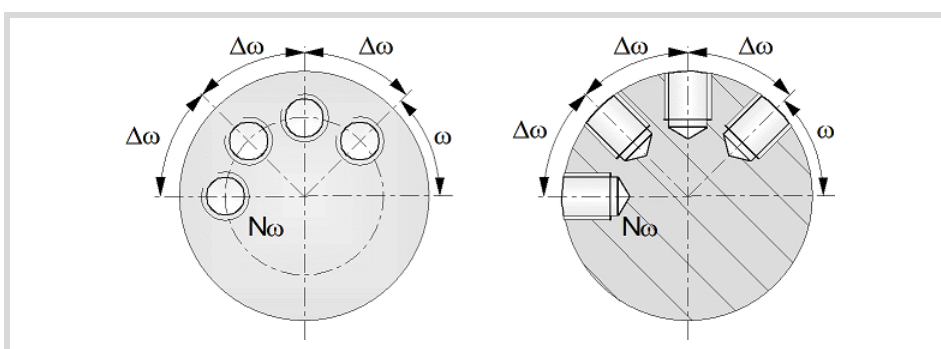
Ø Nominal thread diameter.

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.

10.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Multiple thread milling.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Multiple thread milling.



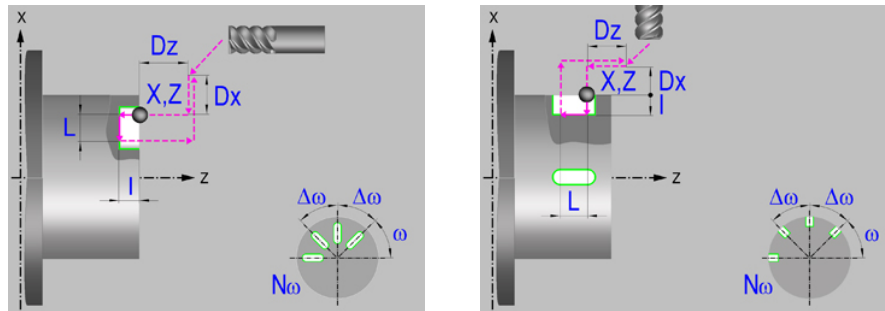
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

10.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Multiple slot milling.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

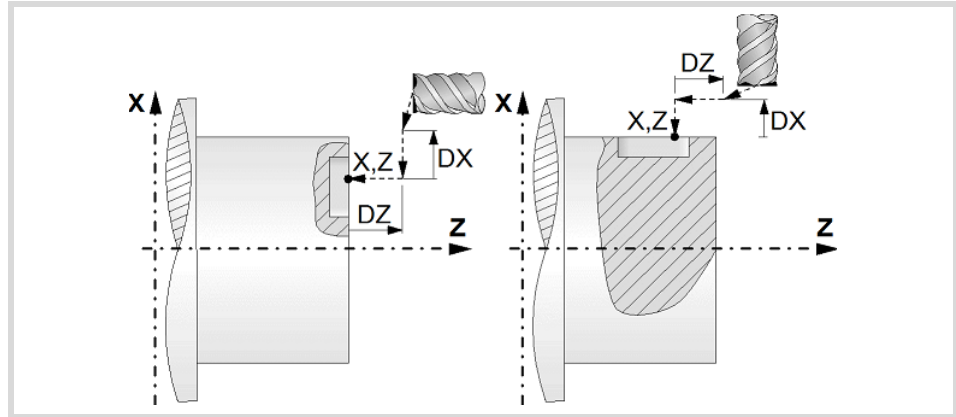
REF: 2102

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.

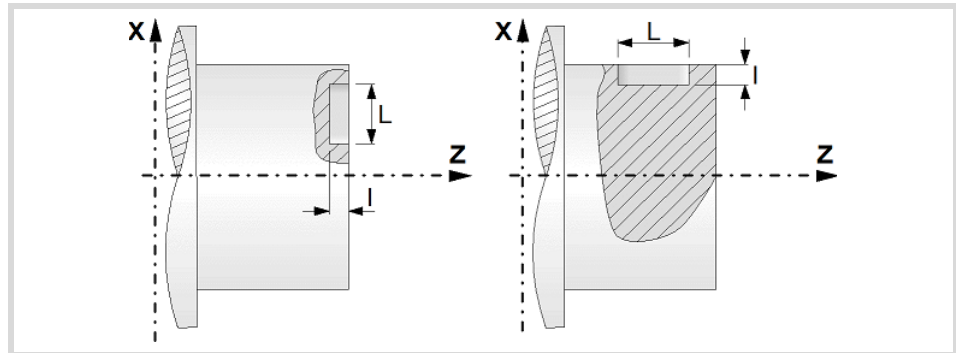
D_x, D_z Safety distance referred to the starting point.

- 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.



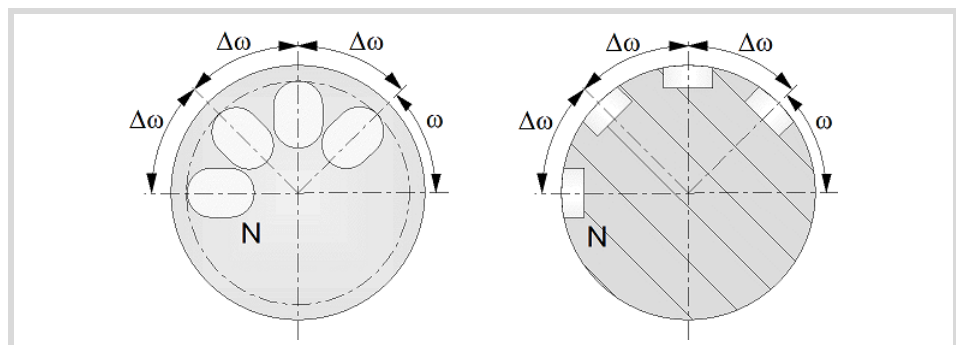
Programming of the drilling operation.

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



10.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Multiple slot milling.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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 cycle 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.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Multiple slot milling.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Multiple slot milling.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

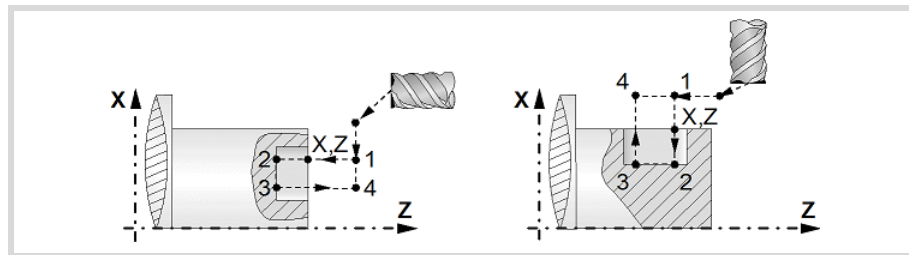
10.8.1 Basic operation.

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 D_x and D_z 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 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

10.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.

Multiple slot milling.



FAGOR AUTOMATION

CNC 8058

CNC 8060

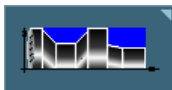
CNC 8065

REF: 2102

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-empted 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-empted 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.



FAGOR AUTOMATION

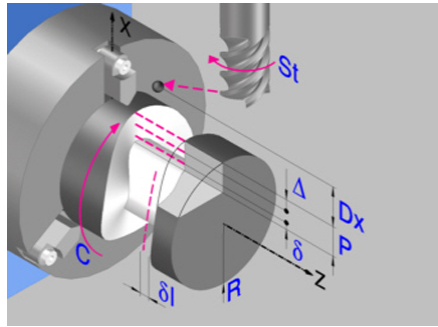
CNC 8058
CNC 8060
CNC 8065

REF: 2102

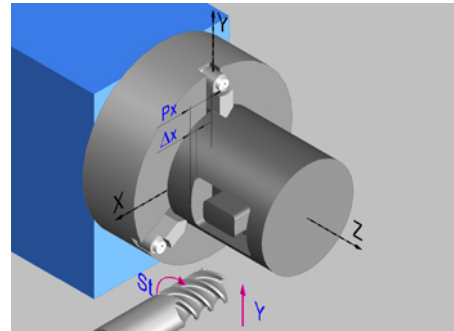
11.1 ZC/YZ plane profiling cycle.

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
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.

X_i, Y_i, Z_i 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.

X_f, Y_f, Z_f 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.

D_x X safety distance referred to the starting point. This parameter is always defined in radius.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

11.

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.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
ZC/YZ plane profiling cycle.

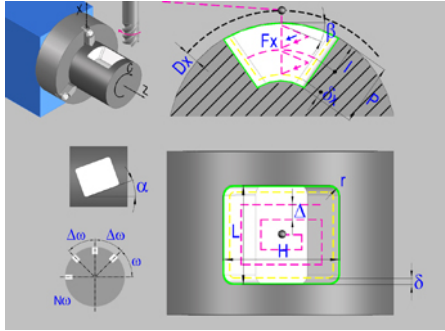


FAGOR AUTOMATION

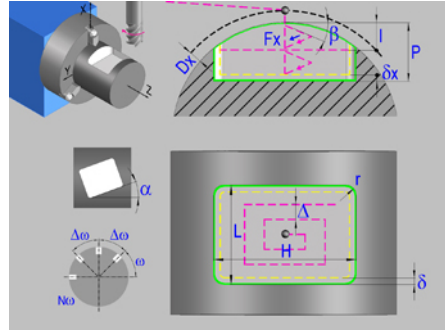
CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

X_c, Y_c, Z_c Coordinates of the starting point. The CNC will show the associated coordinates for the selected plane.

A_c, B_c, Z_c

Coordinates of the starting point.

X_i, Y_i, Z_i 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.

X_f, Y_f, Z_f 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.

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Rectangular pocket cycle in the ZC/YZ plane.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

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

$N\omega$ Total number of operations.

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.

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Rectangular pocket cycle in the ZC/MZ plane.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.

Rectangular pocket cycle in the ZC/NZ plane.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

11.

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.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.

Rectangular pocket cycle in the ZC/NZ plane.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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 passes 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.
Rectangular pocket cycle in the ZC/MZ plane.

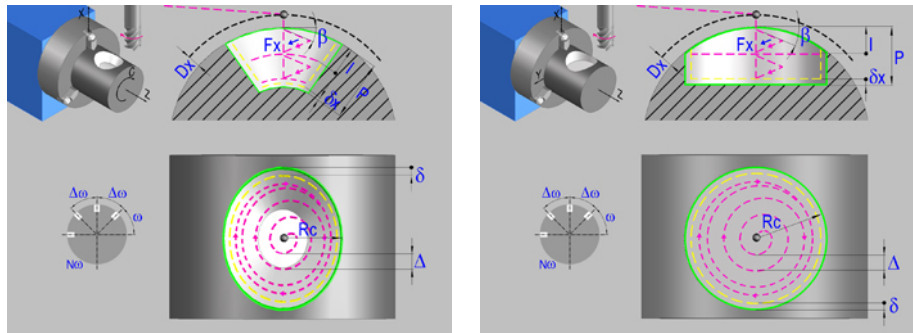


FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

X_c, Y_c, Z_c Coordinates of the center of the pocket. The CNC will show the associated coordinates for the selected plane.

A_c, B_c, Z_c

Coordinates of the starting point.

X_i, Y_i, Z_i 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.

X_f, Y_f, Z_f 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.

R_c 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.

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.

Circular pocket cycle in the ZC/YZ plane.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Circular pocket cycle in the ZC/MZ plane.

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

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).

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Circular pocket cycle in the ZC/MZ plane.

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 passes 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.

Circular pocket cycle in the ZC/NZ plane.

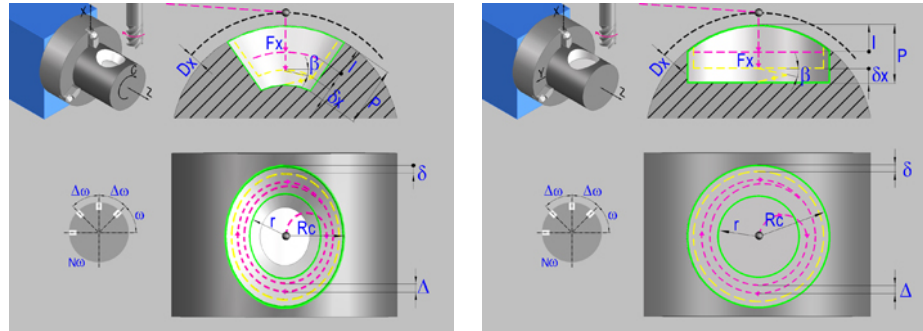
FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

11.4 Circular pre-empted pocket cycle in the ZC/YZ plane.



Circular pre-empted pocket in the ZC plane.

Circular pre-empted pocket in the YZ plane.

Geometric parameters.

Work plane.



ZC plane.



YZ plane.

Coordinates of the center of the pocket.

X_c, Y_c, Z_c Coordinates of the center of the pocket. The CNC will show the associated coordinates for the selected plane.

A_c, B_c, Z_c

Coordinates of the starting point.

X_i, Y_i, Z_i 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.

X_f, Y_f, Z_f 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.

R_c 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.

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Circular pre-empted pocket cycle in the ZC/YZ plane.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

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

$N\omega$ Total number of operations.

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-empted area, it goes in straight in G0, ignoring the penetration angle. If the tool does not fit in the pre-empted area, it goes in at the programmed penetration angle.

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.

Circular pre-empted pocket cycle in the ZC/YZ plane.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Circular pre-empted pocket cycle in the ZC/MZ plane.

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 passes 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.

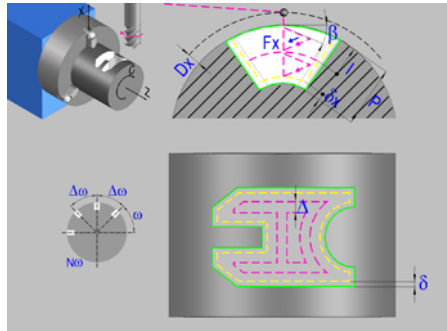
If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- 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.

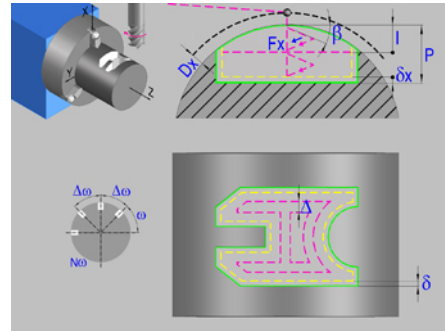
11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Circular pre-empted pocket cycle in the ZC/MZ plane.

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.

X_i, Y_i, Z_i 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.

X_f, Y_f, Z_f 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.

D_x 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.

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
2D profile pocket cycle in the ZC/YZ plane.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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

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.

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
2D profile pocket cycle in the ZC/MZ plane.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

Penetration feedrate.

Fx Penetration feedrate.

Machining strategy (icon).



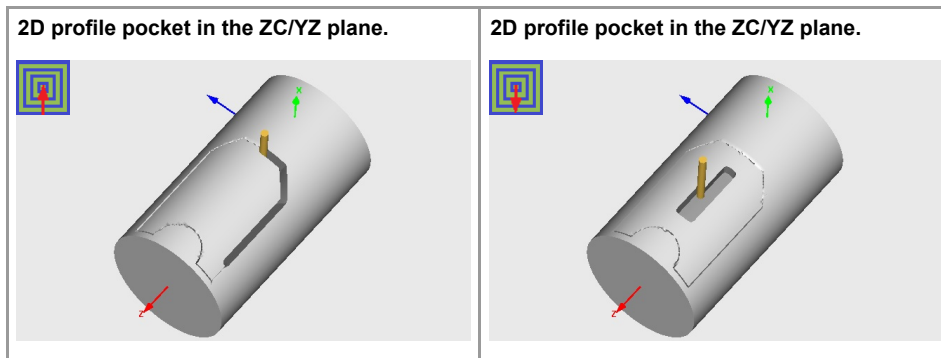
Outward-inward concentric machining. The wall of the pocket is machined in the first pass. Depending on the geometry, this option may be faster, but the wear on the tool is greater.

- 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).



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.

- l 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.

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
2D profile pocket cycle in the ZC/MZ plane.

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 passes 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.

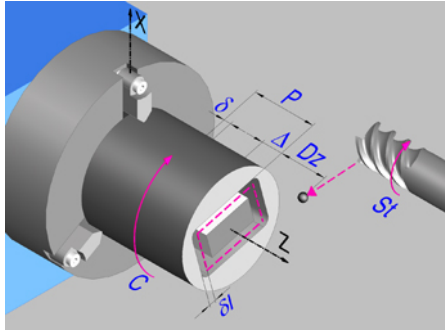
If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- 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.

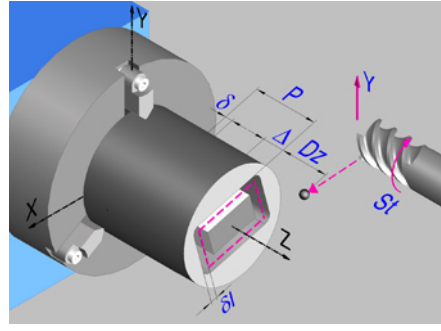
11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
2D profile pocket cycle in the ZC/MZ plane.

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.

X_i, Y_i, Z_i 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.

X_f, Y_f, Z_f 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.

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
XC/XY plane profiling cycle.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
XC/XY plane profiling cycle.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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).

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
XC/XY plane profiling cycle.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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 cycle 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.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
XC/XY plane profiling cycle.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.

XC/XY plane profiling cycle.

FAGOR 

FAGOR AUTOMATION

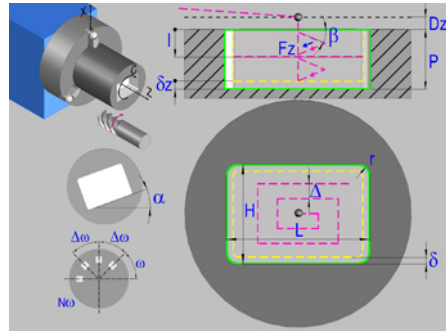
CNC 8058
CNC 8060
CNC 8065

REF: 2102

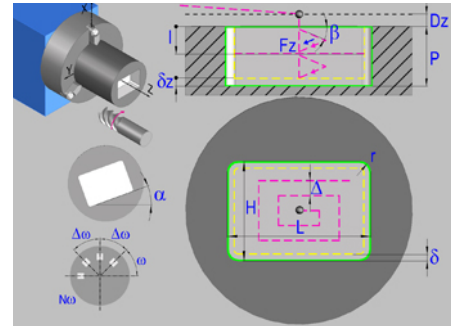
11.7 Rectangular pocket cycle in the XC/XY plane.

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Rectangular pocket cycle in the XC/XY plane.



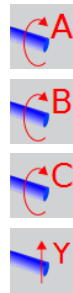
Rectangular pocket in the XC plane.



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

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.

A, B, Z

Coordinates of the starting point.

X_i, Y_i, Z_i

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.

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

N ω 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.

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.

Rectangular pocket cycle in the XC/XY plane.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Rectangular pocket cycle in the XCi/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.



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.

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.

Rectangular pocket cycle in the XC/XY plane.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

- 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.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.

Rectangular pocket cycle in the XC/XY plane.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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 passes 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.

Rectangular pocket cycle in the Xc/Y plane.

FAGOR 

FAGOR AUTOMATION

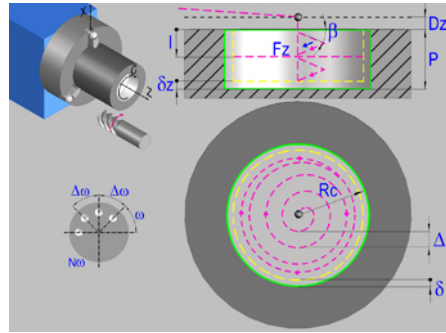
CNC 8058
CNC 8060
CNC 8065

REF: 2102

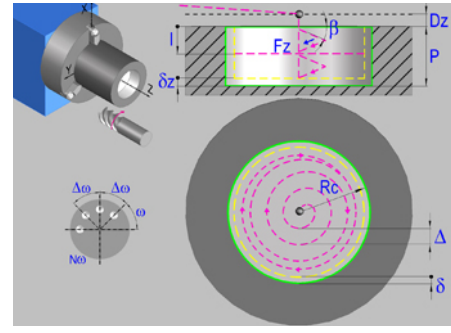
11.8 Circular pocket cycle in the XC/XY plane.

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
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.

X_c, Y_c, Z_c 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.

A_c, B_c, Z_c

Coordinates of the starting point.

X_i, Y_i, Z_i 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.

X_f, Y_f, Z_f 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.

R_c 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.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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.

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

$N\omega$ Total number of operations.

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.

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.

Circular pocket cycle in the XC/XY plane.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Circular pocket cycle in the XCi/Y plane.

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 cycle 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 passes 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.

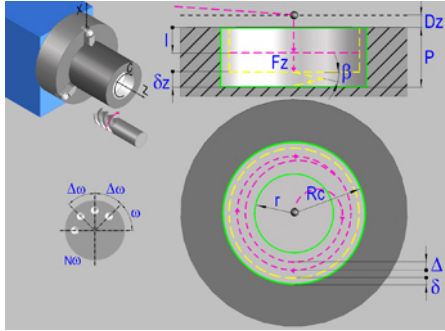
If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- 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.

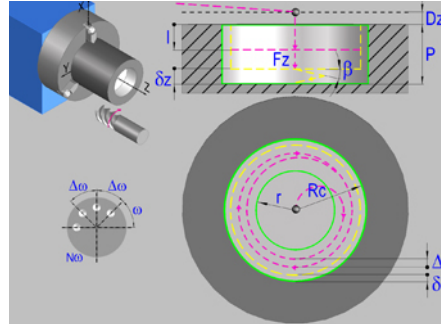
11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Circular pocket cycle in the XC/XY plane.

11.9 Circular pre-empted pocket in the XC/XY plane.



Circular pre-empted pocket in the XC plane.



Circular pre-empted 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 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.
Ac, Bc, Zc

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.

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Circular pre-empted pocket in the XC/XY plane.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Circular pre-empted pocket in the XC/XY plane.

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

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-empted area, it goes in straight in G0, ignoring the penetration angle. If the tool does not fit in the pre-empted 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.

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.

Circular pre-empted pocket in the XC/XY plane.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
Circular pre-empted pocket in the XC/XY plane.



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.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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 passes 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.

Circular pre-empted pocket in the XC/XY plane.

FAGOR 

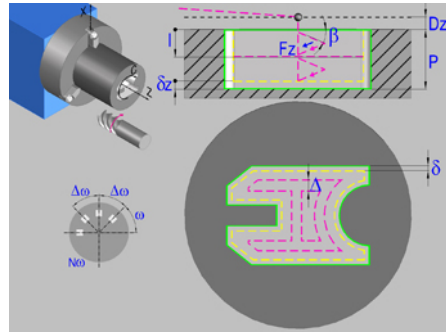
FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

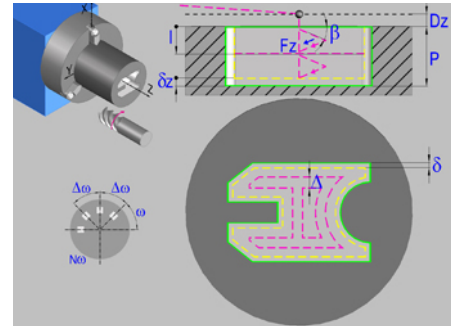
REF: 2102

11.10 2D profile pocket cycle in the XC/XY plane.

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
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.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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

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.

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
2D profile pocket cycle in the XC/XY plane.

Machining strategy (icon).

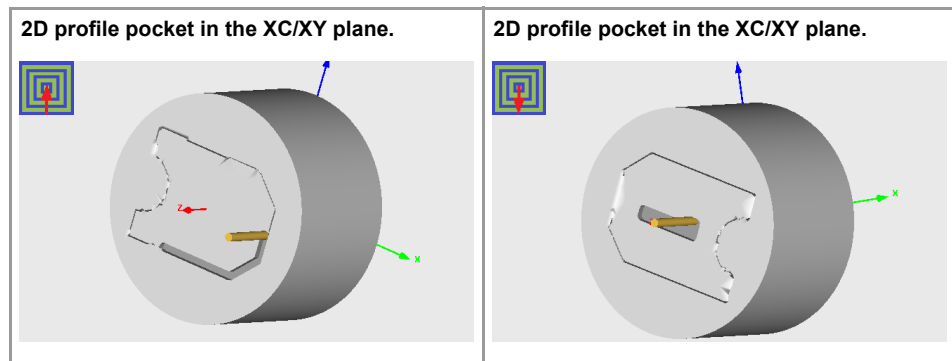
Outward-inward concentric machining. The wall of the pocket is machined in the first pass. Depending on the geometry, this option may be faster, but the wear on the tool is greater.

- 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).

**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.

l 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.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

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).

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.
2D profile pocket cycle in the XCi/XY plane.

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 passes 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/Y plane.

FAGOR 

FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.

2D profile pocket cycle in the XC/XY plane.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

The user and the OEM can add their own canned cycles (user cycles) using the FGUIM application that comes installed on the CNC. The text of the softkey to access these cycles is defined by the creator of the canned cycle. User cycles work in a similar way as Fagor canned cycles.

- It is possible to browse through the cycles
- Pressing the [ESC] key returns to the editor without inserting the cycle.
- Pressing the [INS] inserts the cycle into the program.

User cycles in ISO mode.

The softkey to access the user cycles is on the menu of the EDISIMU mode, and they can even hide the softkeys for the Fagor cycles. In this mode, the cycles help programming, they can only be inserted into the program, the editor does not allow simulating or executing a cycle from its definition screen.

User cycles in conversational mode.

In conversational mode, the user cycle accessing softkey is on the cycles menu. In this mode, the user cycles may be inserted into the program and may be simulated and executed from their definition screen.

Defining user cycles.

The application offers a guided way to define a new component and its softkey menu without having to be familiar with script languages. Follow these steps to create a new canned cycle:

- 1 Access the FGUIM application, installed on the control.
- 2 Create a new user component which will contain the user cycles. Each page of the component will be a canned cycle and will have its own horizontal menu with the softkeys for each cycle.
- 3 Create a softkey for calling the new component with the user cycles, this new component will be integrated into the menu of the editor of the EDISIMU mode.

12.

USER CYCLES.



FAGOR AUTOMATION

CNC 8058

CNC 8060

CNC 8065

REF: 2102

User notes:

Horizontal lines for user notes, with a pencil icon pointing to the first line.



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

User notes:



FAGOR AUTOMATION

CNC 8058
CNC 8060
CNC 8065

REF: 2102

Lined area for user notes, consisting of approximately 27 horizontal lines.



FAGOR AUTOMATION

Fagor Automation S. Coop.

Bº San Andrés, 19 - Apdo. 144
E-20500 Arrasate-Mondragón, Spain

Tel: +34 943 719 200

+34 943 039 800

Fax: +34 943 791 712

E-mail: info@fagorautomation.es

www.fagorautomation.com