CNC 8065

Canned cycles (·M· model)

(Ref: 1209)
MACHINE SAFETY
It is up to the machine manufacturer to make sure that the safety of the machine is enabled in order to prevent personal injury and damage to the CNC or to the products connected to it. On start-up and while validating CNC parameters, it checks the status of the following safety elements. If any of them is disabled, the CNC shows a warning message.

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

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

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

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

COMPUTER VIRUSES
FAGOR AUTOMATION guarantees that the software installed contains no computer viruses. It is up to the user to keep the unit virus free in order to guarantee its proper operation.

Computer viruses at the CNC may cause it to malfunction. An antivirus software is highly recommended if the CNC is connected directly to another PC, it is part of a computer network or floppy disks or other computer media is used to transmit data.

FAGOR AUTOMATION shall not be held responsible for any personal injuries or physical damage caused or suffered by the CNC due to a computer virus in the system.

If a computer virus is found in the system, the unit will no longer be under warranty.
INDEX

CHAPTER 1 MILLING CANNED CYCLES.

1.1 General concepts ........................................ 23
1.1.1 Definition, Influence zone and cancellation of a canned cycle ............. 25
1.1.2 Starting plane and reference plane .................................................. 26
1.1.3 Programming the canned cycles ....................................................... 27
1.1.4 Programming a canned cycle in different planes .......................... 28
1.1.5 Modifying the parameters of a canned cycle ................................. 30
1.2 G81. Drilling canned cycle ............................................................. 32
1.2.1 Programming example ................................................................. 33
1.3 G82. Drilling canned cycle with variable peck ..................................... 34
1.3.1 Programming example ................................................................. 37
1.4 G83. Deep-hole drilling canned cycle with constant peck .................... 38
1.4.1 Programming example ................................................................. 40
1.5 G84. Tapping canned cycle .............................................................. 41
1.5.1 Programming example ................................................................. 43
1.6 G85. Reaming canned cycle ............................................................ 44
1.6.1 Programming example ................................................................. 45
1.7 G86. Boring canned cycle ............................................................... 46
1.7.1 Programming example ................................................................. 48
1.8 G87. Rectangular pocket canned cycle ............................................ 49
1.8.1 Programming example ................................................................. 52
1.9 G88. Circular pocket canned cycle .................................................. 53
1.9.1 Programming example ................................................................. 56
1.10 G210. Bore milling canned cycle .................................................... 57
1.11 G211. Inside thread milling cycle .................................................... 59
1.12 G212. Outside thread milling cycle .................................................. 62

CHAPTER 2 MULTIPLE MACHINING

2.1 G160. Multiple machining in straight line .......................................... 66
2.1.1 Programming example ................................................................. 68
2.2 G161. Multiple machining in rectangular pattern ............................... 69
2.2.1 Programming example ................................................................. 71
2.3 G162. Multiple machining in grid pattern .......................................... 72
2.3.1 Programming example ................................................................. 74
2.4 G163. Multiple machining in a full circle .......................................... 75
2.4.1 Programming example ................................................................. 77
2.5 G164. Multiple machining in arc pattern .......................................... 78
2.5.1 Programming example ................................................................. 80
2.6 G165. Multiple machining in a chord pattern ..................................... 81
2.6.1 Programming example ................................................................. 82

CHAPTER 3 CYCLE EDITOR

3.1 Configuring the cycle editor .............................................................. 85
3.2 Teach-in mode ............................................................................... 86
3.3 Selecting data, profiles and icons .................................................... 87
3.4 Simulate a canned cycle ................................................................. 88

CHAPTER 4 CANNED CYCLES OF THE EDITOR

4.1 Canned cycles available in the editor ............................................... 91
4.1.1 G functions associated with the execution of the cycles .................. 92
4.1.2 Work planes and shifting of the machining operations ................. 93
4.1.3 Value applied when the value of a parameter is 0 ......................... 95
4.1.4 Associate a multiple machining operation with a canned cycle ....... 96
4.2 Center punching ........................................... 97
4.2.1 Basic operation ........................................... 99
4.3 Drilling ........................................................ 100
4.3.1 Basic operation ........................................... 101
4.4 Deep hole drilling ........................................... 102
4.4.1 Basic operation ........................................... 103
4.5 Bore milling .................................................. 104
4.5.1 Basic operation ........................................... 106
4.6 Tapping ....................................................... 107
4.6.1 Basic operation ........................................... 109
4.7 Thread milling .............................................. 110
4.7.1 Basic operation ........................................... 112
4.8 Reaming ..................................................... 113
4.8.1 Basic operation ........................................... 114
4.9 Boring ......................................................... 115
4.9.1 Basic operation ........................................... 116
4.10 Boring with spindle orientation ....................... 117
4.10.1 Basic operation ........................................... 119
4.11 Simple rectangular pocket ......................... 120
4.11.1 Basic operation ........................................... 123
4.12 Rectangular pocket with rounding .................... 124
4.12.1 Basic operation ........................................... 128
4.13 Circular pocket ........................................... 129
4.13.1 Basic operation ........................................... 133
4.14 Pre-empted circular pocket ......................... 134
4.14.1 Basic operation ........................................... 137
4.15 2D profile pocket ........................................... 139
4.15.1 Executable pocket file ............................... 143
4.15.2 Basic operation ........................................... 144
4.15.3 Examples of how to define 2D profiles ............ 145
4.16 3D profile pocket with islands ....................... 148
4.16.1 Executable pocket file ............................... 154
4.16.2 Basic operation ........................................... 155
4.16.3 Examples of how to define 3D profiles ............ 156
4.17 Rectangular boss .......................................... 159
4.17.1 Basic operation ........................................... 163
4.18 Circular boss .............................................. 164
4.18.1 Basic operation ........................................... 167
4.19 Surface milling ............................................ 168
4.19.1 Basic operation ........................................... 171
4.20 Point-to-point profile milling .......................... 172
4.20.1 Basic operation ........................................... 175
4.21 Free profile milling ....................................... 176
4.21.1 Basic operation ........................................... 179
4.22 Slot milling .................................................. 180
4.22.1 Basic operation ........................................... 184
4.23 Multiple machining in straight line .................. 186
4.24 Multiple machining in arc pattern .................... 187
4.25 Multiple machining in rectangular pattern ........... 189
4.26 Multiple machining in grid pattern ................... 190
4.27 Random multiple machining ........................... 191
## ABOUT THE PRODUCT

### BASIC CHARACTERISTICS.

<table>
<thead>
<tr>
<th>Basic characteristics</th>
<th>-M-</th>
<th>-T-</th>
</tr>
</thead>
<tbody>
<tr>
<td>PC-based system.</td>
<td>Open system</td>
<td></td>
</tr>
<tr>
<td>Operating system.</td>
<td>Windows XP</td>
<td></td>
</tr>
<tr>
<td>Number of axes.</td>
<td>3 to 28</td>
<td></td>
</tr>
<tr>
<td>Number of spindles.</td>
<td>1 to 4</td>
<td></td>
</tr>
<tr>
<td>Number of tool magazines.</td>
<td>1 to 4</td>
<td></td>
</tr>
<tr>
<td>Number of execution channels.</td>
<td>1 to 4</td>
<td></td>
</tr>
<tr>
<td>Number of handwheels.</td>
<td>1 to 12</td>
<td></td>
</tr>
<tr>
<td>Type of servo system.</td>
<td>Analog / Digital Sercos / Digital Mechatrolink</td>
<td></td>
</tr>
<tr>
<td>Communications.</td>
<td>RS485 / RS422 / RS232 Ethernet</td>
<td></td>
</tr>
<tr>
<td>Integrated PLC.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PLC execution time.</td>
<td>&lt; 1ms/K</td>
<td></td>
</tr>
<tr>
<td>Digital inputs / Digital outputs.</td>
<td>1024 / 1024</td>
<td></td>
</tr>
<tr>
<td>Marks / Registers.</td>
<td>8192 / 1024</td>
<td></td>
</tr>
<tr>
<td>Timers / Counters.</td>
<td>512 / 256</td>
<td></td>
</tr>
<tr>
<td>Symbols.</td>
<td>Unlimited</td>
<td></td>
</tr>
<tr>
<td>Block processing time.</td>
<td>&lt; 1 ms</td>
<td></td>
</tr>
</tbody>
</table>

### Remote modules.

<table>
<thead>
<tr>
<th>Communication with the remote modules.</th>
<th>RIOW</th>
<th>RIO5</th>
<th>RIO70</th>
</tr>
</thead>
<tbody>
<tr>
<td>Digital inputs per module.</td>
<td>8</td>
<td>16 or 32</td>
<td>16</td>
</tr>
<tr>
<td>Digital outputs per module.</td>
<td>8</td>
<td>24 or 48</td>
<td>16</td>
</tr>
<tr>
<td>Analog inputs per module.</td>
<td>4</td>
<td>4</td>
<td>8</td>
</tr>
<tr>
<td>Analog outputs per module.</td>
<td>4</td>
<td>4</td>
<td>4</td>
</tr>
<tr>
<td>Inputs for PT100 temperature sensors.</td>
<td>2</td>
<td>2</td>
<td>-</td>
</tr>
<tr>
<td>Feedback inputs.</td>
<td>-</td>
<td>-</td>
<td>4</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Customizing.

PC-based open system, fully customizable.
-INI configuration files.
-FGUM visual configuration tool.
-Visual Basic®, Visual C++®, etc.
-Internal databases in Microsoft® Access.
-OPC compatible interface
Bear in mind that some of the features described in this manual depend on the software options that are installed. The information of the following table is informative only; when purchasing the software options, only the information provided in the ordering handbook is valid.

<table>
<thead>
<tr>
<th>Software options (-M· model).</th>
<th>8065 M</th>
<th>8065 M Power</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Basic</td>
<td>Pack 1</td>
</tr>
<tr>
<td>Open system. Access to the administrator mode.</td>
<td>- - -</td>
<td>Option</td>
</tr>
<tr>
<td>Number of execution channels</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Number of axes</td>
<td>3 to 6</td>
<td>5 to 8</td>
</tr>
<tr>
<td>Number of spindles</td>
<td>1</td>
<td>1 to 2</td>
</tr>
<tr>
<td>Number of tool magazines</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Limited to 4 interpolated axes</td>
<td>Option</td>
<td>Option</td>
</tr>
<tr>
<td>IEC 61131 language</td>
<td>- - -</td>
<td>Option</td>
</tr>
<tr>
<td>HD graphics</td>
<td>Option</td>
<td>Option</td>
</tr>
<tr>
<td>Conversational IIP</td>
<td>Option</td>
<td>Option</td>
</tr>
<tr>
<td>Dual-purpose machines (M-T)</td>
<td>Option</td>
<td>Option</td>
</tr>
<tr>
<td>&quot;C&quot; axis</td>
<td>Standard</td>
<td>Standard</td>
</tr>
<tr>
<td>Dynamic RTCP</td>
<td>- - -</td>
<td>Option</td>
</tr>
<tr>
<td>HSSA machining system.</td>
<td>Standard</td>
<td>Standard</td>
</tr>
<tr>
<td>Probing canned cycles</td>
<td>Option</td>
<td>Standard</td>
</tr>
<tr>
<td>Tandem axes</td>
<td>Option</td>
<td>Standard</td>
</tr>
<tr>
<td>Synchronism and cams</td>
<td>Option</td>
<td>Standard</td>
</tr>
<tr>
<td>Tangential control</td>
<td>Option</td>
<td>Standard</td>
</tr>
<tr>
<td>Volumetric compensation (up to 10 m³).</td>
<td>Option</td>
<td>Standard</td>
</tr>
<tr>
<td>Volumetric compensation (more than 10 m³).</td>
<td>Option</td>
<td>Standard</td>
</tr>
</tbody>
</table>
## Canned cycles (·M· model)

Software options (·T· model).

<table>
<thead>
<tr>
<th>Feature</th>
<th>8065 T Basic</th>
<th>8065 T Pack 1</th>
<th>8065 T Power Basic</th>
<th>8065 T Power Pack 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Open system, Access to the administrator mode.</td>
<td>- - -</td>
<td>- - -</td>
<td>Option</td>
<td>Option</td>
</tr>
<tr>
<td>Number of execution channels</td>
<td>1</td>
<td>1 to 2</td>
<td>1 to 2</td>
<td>1 to 4</td>
</tr>
<tr>
<td>Number of axes</td>
<td>3 to 5</td>
<td>5 to 7</td>
<td>5 to 12</td>
<td>8 to 28</td>
</tr>
<tr>
<td>Number of spindles</td>
<td>2</td>
<td>2</td>
<td>3 to 4</td>
<td>3 to 4</td>
</tr>
<tr>
<td>Number of tool magazines</td>
<td>1</td>
<td>1 to 2</td>
<td>1 to 2</td>
<td>1 to 4</td>
</tr>
<tr>
<td>Limited to 4 interpolated axes</td>
<td>Option</td>
<td>Option</td>
<td>Option</td>
<td>Option</td>
</tr>
<tr>
<td>IEC 61131 language</td>
<td>- - -</td>
<td>Option</td>
<td>Option</td>
<td>Option</td>
</tr>
<tr>
<td>HD graphics</td>
<td>Option</td>
<td>Option</td>
<td>Standard</td>
<td>Standard</td>
</tr>
<tr>
<td>Conversational IIP</td>
<td>Option</td>
<td>Option</td>
<td>Option</td>
<td>Option</td>
</tr>
<tr>
<td>Dual-purpose machines (T-M)</td>
<td>- - -</td>
<td>- - -</td>
<td>Option</td>
<td>Standard</td>
</tr>
<tr>
<td>&quot;C&quot; axis</td>
<td>Option</td>
<td>Standard</td>
<td>Standard</td>
<td>Standard</td>
</tr>
<tr>
<td>Dynamic RTCP</td>
<td>- - -</td>
<td>- - -</td>
<td>Option</td>
<td>Standard</td>
</tr>
<tr>
<td>HSSA machining system</td>
<td>Option</td>
<td>Standard</td>
<td>Standard</td>
<td>Standard</td>
</tr>
<tr>
<td>Probing canned cycles</td>
<td>Option</td>
<td>Standard</td>
<td>Standard</td>
<td>Standard</td>
</tr>
<tr>
<td>Tandem axes</td>
<td>- - -</td>
<td>Option</td>
<td>Standard</td>
<td>Standard</td>
</tr>
<tr>
<td>Synchronism and cams</td>
<td>- - -</td>
<td>Option</td>
<td>Option</td>
<td>Standard</td>
</tr>
<tr>
<td>Tangential control</td>
<td>- - -</td>
<td>- - -</td>
<td>Option</td>
<td>Standard</td>
</tr>
<tr>
<td>Volumetric compensation (up to 10 m³)</td>
<td>- - -</td>
<td>- - -</td>
<td>Option</td>
<td>Option</td>
</tr>
<tr>
<td>Volumetric compensation (more than 10 m³)</td>
<td>- - -</td>
<td>- - -</td>
<td>Option</td>
<td>Option</td>
</tr>
</tbody>
</table>
DECLARATION OF CONFORMITY

The manufacturer:
Fagor Automation S. Coop.
Barrio de San Andrés Nº 19, C.P.20500, Mondragón -Guipúzcoa- (Spain).

Declares:
The manufacturer declares under their exclusive responsibility the conformity of the product:

8065 CNC

Consisting of the following modules and accessories:
8065-M-ICU
8065-T-ICU
MONITOR-LCD-10, MONITOR-LCD-15
HORIZONTAL-KEYB, VERTICAL-KEYB, OP-PANEL
BATTERY
Remote Modules RIOW, RIO5, RIO70

Note. Some additional characters may follow the model references indicated above. They all comply with the directives listed here. However, compliance may be verified on the label of the unit itself.

Referred to by this declaration with following directives:
Low-voltage regulations.

Regulation on electromagnetic compatibility.


In Mondragón, October 1st 2011.

Fagor Automation, S Coop.

Pedro Ruiz de Aguirre
Canned cycles (·M· model)

VERSION HISTORY

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

---

**Ref. 1103**

First version.

---

**Ref. 1201**

**Software V04.22**

- Cycle editor. The [DEL] key deletes a profile from the list.
- Cycles of the editor. The table to define the points of the profile admits 25 points.
- Cycles of the editor. New icon to delete all the points of the table.

  - Point-to-point profile milling.

---

**Ref. 1209**

**Software V04.24**

- Cycles of the editor. The starting point of the pocket may be in its center.

  - Simple rectangular pocket.
  - Rectangular pocket with rounding.
Canned cycles (·M· model)

CNC 8065

(REF: 1209)

SAFETY CONDITIONS

Read the following safety measures in order to prevent harming people or damage to this product and those products connected to it. Fagor Automation shall not be held responsible of any physical damage or defective unit resulting from not complying with these basic safety regulations.

Before start-up, verify that the machine that integrates this CNC meets the 89/392/CEE Directive.

PRECAUTIONS BEFORE CLEANING THE UNIT

If the CNC does not turn on when actuating the start-up switch, verify the connections.

Do not get into the inside of the unit.

Only personnel authorized by Fagor Automation may manipulate the inside of this unit.

Do not handle the connectors with the unit connected to AC power.

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

PRECAUTIONS DURING REPAIR

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

Do not get into the inside of the unit.

Only personnel authorized by Fagor Automation may manipulate the inside of this unit.

Do not handle the connectors with the unit connected to AC power.

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

PRECAUTIONS AGAINST PERSONAL DAMAGE

Interconnection of modules.

Use the connection cables provided with the unit.

Use proper cables.

To prevent risks, use the proper cables for mains, Sercos and Bus CAN recommended for this unit.

In order to avoid electrical shock at the central unit, use the proper power (mains) cable. Use 3-wire power cables (one for ground connection).

Avoid electrical overloads.

In order to avoid electrical discharges and fire hazards, do not apply electrical voltage outside the range selected on the rear panel of the central unit.

Ground connection.

In order to avoid electrical discharges, connect the ground terminals of all the modules to the main ground terminal. Before connecting the inputs and outputs of this unit, make sure that all the grounding connections are properly made.

In order to avoid electrical shock, before turning the unit on verify that the ground connection is properly made.

Do not work in humid environments.

In order to avoid electrical discharges, always work under 90% of relative humidity (non-condensing) and 45 °C (113 °F).

Do not work in explosive environments.

In order to avoid risks or damages, do no work in explosive environments.
PRECAUTIONS AGAINST PRODUCT DAMAGE

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

Install the unit in the right place. It is recommended, whenever possible, to install the CNC away from coolants, chemical product, blows, etc. that could damage it. This unit complies with the European directives on electromagnetic compatibility. Nevertheless, it is recommended to keep it away from sources of electromagnetic disturbance such as:
- Powerful loads connected to the same AC power line as this equipment.
- Nearby portable transmitters (Radio-telephones, Ham radio transmitters).
- Nearby radio/TV transmitters.
- Nearby arc welding machines.
- Nearby High Voltage power lines.

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

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 and the remote modules.

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

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

Ambient conditions. The storage temperature must be between +5 ºC and +45 ºC (41 ºF and 113 ºF).
- The storage temperature must be between -25 ºC and 70 ºC (-13 ºF and 158 ºF).

Central unit enclosure. Make sure that the needed gap is kept between the central unit and each wall of the enclosure.
- Use a DC fan to improve enclosure ventilation.

Main AC power switch. This switch must be easy to access and at a distance between 0.7 and 1.7 m (2.3 and 5.6 ft) off the floor.

PROTECTIONS OF THE UNIT ITSELF

Remote modules. All the digital inputs and outputs have galvanic isolation via optocouplers between the CNC circuitry and the outside.
## SAFETY SYMBOLS

### Symbols that may appear on the manual.

- **Danger or prohibition symbol.**
  It indicates actions or operations that may hurt people or damage products.

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

- **Obligation symbol.**
  It indicates actions and operations that must be carried out.

- **Information symbol.**
  It indicates notes, warnings and advises.

### Symbols that the product may carry.

- **Ground protection symbol.**
  It indicates that that point must be under voltage.
WARRANTY TERMS

INITIAL WARRANTY

All products manufactured or marketed by FAGOR carry a 12-month warranty for the end user which could be controlled by our service network by means of the warranty control system established by FAGOR for this purpose.

In order to prevent the possibility of having the time period from the time a product leaves our warehouse until the end user actually receives it run against this 12-month warranty, FAGOR has set up a warranty control system based on having the manufacturer or agent inform FAGOR of the destination, identification and on-machine installation date, by filling out the document accompanying each FAGOR product in the warranty envelope. This system, besides assuring a full year of warranty to the end user, enables our service network to know about FAGOR equipment coming from other countries into their area of responsibility.

The warranty starting date will be the one appearing as the installation date on the above mentioned document. FAGOR offers the manufacturer or agent 12 months to sell and install the product. This means that the warranty starting date may be up to one year after the product has left our warehouse so long as the warranty control sheet has been sent back to us. This translates into the extension of warranty period to two years since the product left our warehouse. If this sheet has not been sent to us, the warranty period ends 15 months from when the product left our warehouse.

This warranty covers all costs of material and labour involved in repairs at FAGOR carried out to correct malfunctions in the equipment. FAGOR undertakes to repair or replace their products within the period from the moment manufacture begins until 8 years after the date on which it disappears from the catalogue.

It is entirely up to FAGOR to determine whether the repair is or not under warranty.

EXCLUDING CLAUSES

Repairs will be carried out on our premises. Therefore, all expenses incurred as a result of trips made by technical personnel to carry out equipment repairs, despite these being within the above-mentioned period of warranty, are not covered by the warranty.

Said warranty will be applied whenever the equipment has been installed in accordance with instructions, has not been mistreated, has not been damaged by accident or by negligence and has not been tampered with by personnel not authorised by FAGOR. If, once servicing or repairs have been made, the cause of the malfunction cannot be attributed to said elements, the customer is obliged to cover the expenses incurred, in accordance with the tariffs in force.

Other warranties, implicit or explicit, are not covered and FAGOR AUTOMATION cannot be held responsible for other damages which may occur.
WARRANTY ON REPAIRS

In a similar way to the initial warranty, FAGOR offers a warranty on standard repairs according to the following conditions:

<table>
<thead>
<tr>
<th>PERIOD</th>
<th>12 months.</th>
</tr>
</thead>
<tbody>
<tr>
<td>CONCEPT</td>
<td>Covers parts and labor for repairs (or replacements) at the network's own facilities.</td>
</tr>
<tr>
<td>EXCLUDING CLAUSES</td>
<td>The same as those applied regarding the chapter on initial warranty. If the repair is carried out within the warranty period, the warranty extension has no effect.</td>
</tr>
</tbody>
</table>

When the customer does not choose the standard repair and just the faulty material has been replaced, the warranty will cover just the replaced parts or components within 12 months.

For sold parts the warranty is 12 months length.

SERVICE CONTRACTS

The SERVICE CONTRACT is available for the distributor or manufacturer who buys and installs our CNC systems.
MATERIAL RETURNING TERMS

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

1. Get a cardboard box whose three inside dimensions are at least 15 cm (6 inches) larger than those of the unit. The cardboard being used to make the box must have a resistance of 170 Kg (375 lb.).

2. Attach a label indicating the owner of the unit, person to contact, type of unit and serial number. In case of malfunction also indicate symptom and a brief description of the problem.

3. Wrap the unit in a polyethylene roll or similar material to protect it. When sending a central unit with monitor, protect especially the screen.

4. Pad the unit inside the cardboard box with poly-etherane foam on all sides.

5. Seal the cardboard box with packing tape or industrial staples.
CNC MAINTENANCE

CLEANING

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

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

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

PRECAUTIONS BEFORE CLEANING THE UNIT

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

- Do not handle the connectors with the unit connected to AC power. Before handling these connectors (I/O, feedback, etc.), make sure that the unit is not connected to main AC power.
- Do not get into the inside of the unit. Only personnel authorized by Fagor Automation may manipulate the inside of this unit.
- If the CNC does not turn on when actuating the start-up switch, verify the connections.
1.1 General concepts

Some canned cycles are edited in ISO code (described in this chapter) and others are generated in conversational mode (described in chapter "4 Canned cycles of the editor"). 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.
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:

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

Drilling canned cycle.
G81 ...
#LATHECY ON
G81 ...
#LATHECY OFF

Turning canned cycle with straight sections.
G81 ...
#MILLCY ON
G81 ...
#MILLCY OFF

General concepts
On a mill model CNC (milling software installed).
By default, it will execute the turning canned cycles. To execute the milling canned cycles, use the following instructions:

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:

- To activate the turning canned cycles.
#LATHECY ON
G81 ...
G87 ...
#LATHECY OFF

- To deactivate the turning canned cycles.
G81 ...
#MILLCY ON
G81 ...
G86 ...
#MILLCY OFF
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 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 program 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.

<table>
<thead>
<tr>
<th>G99 G81 Z2 I-20</th>
</tr>
</thead>
<tbody>
<tr>
<td>Definition and execution of the drilling canned cycle.</td>
</tr>
<tr>
<td>G90 G01 X85</td>
</tr>
<tr>
<td>Move to point X85 and drill a new hole.</td>
</tr>
<tr>
<td>G91 Y85 NR3</td>
</tr>
<tr>
<td>The CNC repeats the movement and the drilling three times.</td>
</tr>
<tr>
<td>G90 G01 X0 NR0</td>
</tr>
<tr>
<td>Move to point X0 without drilling.</td>
</tr>
</tbody>
</table>

**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.
1.1.2 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).

![Diagram of canned cycles](image)

- 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
```
1.1.4 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 #TOO 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. General concepts

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 M3
G81 X2=2 I-8 K1
```
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
```
Canned cycles (·M· model)

1. General concepts

T1 M6
G00 G90 X0 Y0 Z60 F1000
Starting point.
G99 X15 Y25 G81 Z32 I18
Drilling canned cycle. Drilling A.
G98 X25
Drilling canned cycle. Drilling B.
V.C.Z=52
New reference plane and machining penetration.
G99 X35
Drilling canned cycle. Drilling C.
G98 X45
Drilling canned cycle. Drilling D.
V.C.Z=32
New reference plane and machining penetration.
G99 X55
Drilling canned cycle. Drilling E.
G98 X65
Drilling canned cycle. Drilling F.
M30
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".
1.2.1 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
1.3 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 (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).

- **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.
Canned cycles (M· model)

G82. Drilling canned cycle with variable peck

1. 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".

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, it assumes 1 mm.

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.
5 Dwell at the bottom of the hole. The time indicated by "K" in seconds.

6 Rapid withdrawal (G0) to the starting plane (Z) 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".
1.3.1 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
1.4 G83. Deep-hole drilling canned cycle with constant peck

Programming format in Cartesian coordinates:

\[
\text{G83 } \text{Z } \text{I } \text{J } \text{B } \text{K}
\]

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 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 (Zi) 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).
4 Dwell at the bottom of the hole. The time indicated by "K" in seconds.

5 Rapid withdrawal (G0) to the starting plane (Z) if function G98 is active or to the reference plane (Z) if function G99 is active.
1.4.1 Programming example

### 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
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:

\[ \text{G84 } \text{Z I K R J} \]

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.

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. It is executed at 100% of the feedrate “F” and spindle speed “S” programmed. Tapping cannot be interrupted.
4. If “K” other than 0, spindle stop (M05) and dwell.
5. Reverse the spindle turning direction.
   - Withdrawal, exit the tap, to the reference plane. At 100% of the feedrate “F” and spindle speed “S” programmed. 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).
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
1.6 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 (Zi).
  - If not programmed, it assumes as reference plane the current position of the tool (Z=Zi).

- **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 (Zi) 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 (Zi).
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
1.7 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.
Canned cycles (M model)

CNC 8065

MILLING CANNED CYCLES.

1.86. Boring canned cycle

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.

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.
1.7.1 Programming example

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
1.8 G87. Rectangular pocket canned cycle.

Programming format in Cartesian coordinates:

\[
G87 \quad \text{Z I D A J K M Q B C L H V}
\]

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.

**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).

**Parameters:**

- **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.
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.

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 (Z) if function G98 is active or to the reference plane (Z) if function G99 is active.
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 ····· 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 Y35 F800
G87 Z2 I-20 D2 A15 J40 K20 M1 Q10 B5 C5 ····· V50
```

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 G99 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-45 D2 A15 J40 K20 M1 Q10 B5 C5 L1 H300 V50
N20 X140 Y100
N30 G98 X150 Y100
M30
```
1.9  G88. Circular pocket canned cycle

Programming format in Cartesian coordinates:

\[ \text{G88 Z I D J B C L H V} \]

**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.
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.
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.
1.9.1 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.

![Diagram of a pocket with dimensions and coordinates]

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 G99 X350 Y235
M30

**Incremental programming:**

T8 D1 M6
G0 G90 X0 Y0 Z45 S1000 M3 M8 M41 F800
N10 G99 G91 X60 Y60
G87 Z-10 I-35 D10 J20 B5 C5 L1 H300 V50
N20 X140 Y75
N30 G98 X150 Y100
M30
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. Each plane can be the current position of the tool or referred to the start 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 machining depth. If not programmed, it assumes 0.
- **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. If not programmed or programmed with a 0 value, it means that no hole has been previously drilled.
- **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.
- **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.
**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).
1.11 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.
Canned cycles (M model)

1. MILLING CANNED CYCLES.

G21 Inside thread milling cycle

**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.

   - **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.
Canned cycles (Motions model)

1. CNC 8065

MILLING CANNED CYCLES:
G211, Inside thread milling cycle

1. Rapid movement to the center of the hole (X, Y).
2. Rapid movement to the thread entry coordinate along the longitudinal axis.
3. Repetition of steps 3 to 8 until reaching the depth of the finishing stock.
4. Repetition of steps 3 to 8 until reaching the bottom of the thread.
5. Rapid movement to the reference plane (G99) or to the starting plane (G98).
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.
Canned cycles (·M· model)

CNC 8065

MILLING CANNED CYCLES.

G212: Outside thread milling cycle

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 (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).

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.

σ 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.

σ 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.

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

When selecting the "X-I" format, bear in mind that the resulting number of machining operations must be an integer, otherwise, the CNC will issue the relevant error message.

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
Canned cycles (·M· model)

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.
2.1.1 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
2.2 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 selecting the "X-I" format, bear in mind that the resulting number of machining operations must be an integer, otherwise, the CNC will issue the relevant error message.

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

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.
### 2.2.1 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:

```plaintext
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:

```plaintext
G161 A30 X700 K8 J60 P2.005 Q9.011
G161 A30 I100 K8 Y180 D4 P2.005 Q9.011
```
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".

\[ \text{G162 A B X I K Y J D P Q R S T U V} \]

- **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 selecting the "X-I" format, bear in mind that the resulting number of machining operations must be an integer, otherwise, the CNC will issue the relevant error message.

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.
Canned cycles (·M· model)

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.

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
2.3.1 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
2.4 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 360º, 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
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.
2.4.1 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
2.5 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".

\[ \text{G164 X Y B I K F P Q R S T U V} \]

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.
- **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
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.
### 2.5.1 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
```
2.6 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.
2.6.1 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
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.

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).

Activating 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 "3.2 Teach-in mode." on page 86.

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

Accessing the probing cycles.
The "+" softkey shows the softkey to access the probing cycles or those of the lathe mode (if available).
3.1 Configuring the cycle editor.

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

**Programming M functions in each operation.**

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

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

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

**Programming the next tool.**

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

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

**Programming the part surface approach distance.**

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

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

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

**Select axis configuration.**

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

The canned cycles have no work plane associated with them, they are executed in the current active work plane.
### 3.2 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.
3.3 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.
3.4 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.

![START STOP RESET icons]

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.

## 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 RESEITing 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.**

<table>
<thead>
<tr>
<th>Hotkey</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>[CTRL] + [F2]</td>
<td>In the parameter window, it toggles between the cycle parameters and the positioning parameters.</td>
</tr>
<tr>
<td>[CTRL]+[G]</td>
<td>It selects the graphics window.</td>
</tr>
<tr>
<td></td>
<td>It shrinks or expands the graphics window.</td>
</tr>
<tr>
<td></td>
<td>It shows the dialog area for the graphics data.</td>
</tr>
<tr>
<td>[CTRL]+[D]</td>
<td>It activates the periodic calculation of the best display area.</td>
</tr>
<tr>
<td>[SHIFT]+[G]</td>
<td>It shows the graphics window when a simulation is running and the parameter editing window is active.</td>
</tr>
<tr>
<td>[G]</td>
<td></td>
</tr>
<tr>
<td>[ESC]</td>
<td>If the graphics are shown at full screen, it shows the cycle editor screen.</td>
</tr>
</tbody>
</table>
Canned cycles (•M• model)

3. CYCLE EDITOR

Simulate a canned cycle.
4.1 Canned cycles available in the editor.

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.

**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). Multiple machining may be associated with canned cycles so it can be repeated in several points.
4.1.1 G functions associated with the execution of the cycles.

While executing these canned cycles, the CNC shows the following "G" functions in the window for active functions.

<table>
<thead>
<tr>
<th>Function</th>
<th>Canned cycle</th>
</tr>
</thead>
<tbody>
<tr>
<td>G281</td>
<td>Center punching</td>
</tr>
<tr>
<td>G282</td>
<td>Drilling</td>
</tr>
<tr>
<td>G283</td>
<td>Deep hole drilling</td>
</tr>
<tr>
<td>G284</td>
<td>Tapping</td>
</tr>
<tr>
<td>G285</td>
<td>Reaming</td>
</tr>
<tr>
<td>G286</td>
<td>Boring</td>
</tr>
<tr>
<td>G287</td>
<td>Rectangular pocket with rounding</td>
</tr>
<tr>
<td>G288</td>
<td>Circular pocket</td>
</tr>
<tr>
<td>G289</td>
<td>Simple rectangular pocket</td>
</tr>
<tr>
<td>G290</td>
<td>Surface milling</td>
</tr>
<tr>
<td>G291</td>
<td>Rectangular boss</td>
</tr>
<tr>
<td>G292</td>
<td>Circular boss</td>
</tr>
<tr>
<td>G293</td>
<td>Point-to-point profile milling</td>
</tr>
<tr>
<td>G294</td>
<td>Free profile milling</td>
</tr>
<tr>
<td>G295</td>
<td>Slot milling</td>
</tr>
<tr>
<td>G296</td>
<td>Pre-empted circular pocket</td>
</tr>
<tr>
<td>G297</td>
<td>Boring with spindle orientation</td>
</tr>
<tr>
<td>G298</td>
<td>Bore milling</td>
</tr>
<tr>
<td>G299</td>
<td>Thread milling</td>
</tr>
</tbody>
</table>
4.1.2 Work planes and shifting of the machining operations.

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

Work planes along the longitudinal axis.

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

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

Machining direction.

The machining direction is set by the position of the part surface (Z) and of the safety plane (Zs). If they are both the same, the direction is set by the sign of total machining depth (parameter P). If Z=Zs and P>0 machining in the negative direction of the longitudinal axis (Z-), if Z=Zs and P<0 machining in the positive direction (Z+).
Movments in the work planes.

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

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

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

The approach plane permits, as in the case of the figure, a fast approach to the machining surface when the safety plane (Zs) is far away from the part surface.
4.1.3 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 β=0 and θ=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.
4.1.4 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.
- Rectangular boss and circular boss.

How to select multiple machining.

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

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

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

**Geometric parameters:**

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

Depth programming type (icon).

- Programming the total depth.
- Programming the angle and the diameter.

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

**Machining parameters:**

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

Spindle turning direction (icon).

- Clockwise.
- Counterclockwise.

Programming of M functions.

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

- Activating or deactivating the preparation of the next tool.
- The editor will only show this option if the user has configured the editor to allow programming the next tool.
4.2.1 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. 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.
4.3 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.
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.
4.3.1 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) 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, 6, 7.
4.4 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.
- 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.
- 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.
4.4.1 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" 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. Repeats steps 3, 4, 5, 6, 7.
4.5 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.

![Diagram of Bore milling process]

**Geometric parameters:**

**Machining direction.**

- Defines the direction of the helical drilling path.

**Bottom milling.**

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

- **X, Y** Machining point.
- **Z** Part surface coordinate.
- **Zs** Safety plane coordinate.
- **P** Total depth.
- **Dp** Part surface approach distance.

  *The editor will only show this data if the user has configured the editor to allow it. If this option is not enabled, the cycle assumes an approach distance of 1 mm.*

- **φ** Hole diameter.
- **φK** Pre-hole diameter.

  *Starting with a hole previously drilled, this parameter defines the diameter of that hole. If not programmed or programmed with a 0 value, it means that no hole has been previously drilled. The tool must meet the following conditions:
  - The tool radius must be smaller than J/2.
  - The tool radius must be equal to or greater than (J-K)/4.

  *If these two conditions are not met, the CNC issues the corresponding error message.*

- **B** Helical penetration step.

**Machining parameters:**

- **F** Feedrate.
- **S** Spindle speed.
- **T** Tool.
- **D** Tool offset.
Spindle turning direction (icon).
- Clockwise.
- Counterclockwise.

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

Next tool.
- Activating or deactivating the preparation of the next tool.
- The editor will only show this option if the user has configured the editor to allow programming the next tool.
4.5.1 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).
4.6 Tapping.

Geometric parameters:

- \( X, Y \) Machining point.
- \( Z \) Part surface coordinate.
- \( Z_s \) Safety plane coordinate.
- \( D_p \) 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.
- \( K_f \) 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 \( (K_f) \) and the speed adapts to the new feedrate.

Type of tapping (icon):

- Tapping with a clutch.
- Rigid tapping.

Machining parameters:

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

Spindle turning direction (icon):

- Clockwise.
- Counterclockwise.

Type of feedrate (icon):

- In \( \text{mm/min or (inch/min)} \)
- In \( \text{mm/turn} \)
4. CANNED CYCLES OF THE EDITOR

Tapping.

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.
4.6.1 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. It is executed at 100% of the feedrate "F" and spindle speed "S" programmed. Tapping with a clutch cannot be interrupted. 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. If tapping with a clutch, it reverses the spindle turning direction.

7. Withdrawal, exit the tap, to the approach plane.
   At 100% of the feedrate "F" and spindle speed "S" programmed. The thread exit cannot be interrupted when tapping with a clutch. In rigid tapping, the feedrate override percentage may be changed and even stopped (0% override).

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

9. 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:

10. Rapid movement (G0) to the next point.

11. Repeats steps 3, 4, 5, 6, 7, 8, 9.
4.7 Thread milling.

Geometric parameters:

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

Machining direction.
- Defines the direction of the helical drilling path.

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

Type of tapping.
- It depends on the type of tool being used.

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

φ Thread diameter.
K Thread depth.
B Thread pitch.
Ds Approach distance.
α Thread entry (start) angle.
Angle (in degrees) of the segment formed by the center of the hole and the thread entry point with respect to the abscissa axis.
N Number of cutter edges
- It defines the number of cutting edges of the cutter (only when machining with a cutter of n edges).
Canned cycles (·M· model)

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.
4.7.1 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).
4.8 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.
4.8.1 **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.

![Diagram of reaming operation](image)

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.
4.9  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.
4.9.1 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".
   - 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.
### 4.10 Boring with spindle orientation.

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

**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.
4.

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.
4.10.1 Basic operation.

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

3. Rapid movement (G0) up to the approach plane.
4. Penetration at feedrate "F".
5. Dwell "t".
6. The spindle stops and the tool is oriented in the "β" position (M19).
7. It gets the cutter off the wall. It moves the distance indicated by "Δx, Δy".
8. Rapid withdrawal (G0) up to the approach plane.
9. The tool returns to its position (XY) and starts the spindle in the direction it was turning.
10. Rapid movement (G0) up to the safety plane (Zs).

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

11. Rapid movement (G0) to the next point.
12. Repeats steps 3, 4, 5, 6, 7, 8, 9, 10.
4.11 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.

**Machining parameters:**

- $\Delta$: 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.

- $\delta$: Finishing stock on the side walls.
I Penetration step.

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

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

Fz Penetration feedrate.

F Surface milling feedrate.
S Spindle speed.
T Tool.
D Tool offset.

Spindle turning direction (icon).
- Clockwise.
- Counterclockwise.

Machining direction (icon).
- Clockwise.
- Counterclockwise.

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

Activating or deactivating the preparation of the next tool.

The editor will only show this option if the user has configured the editor to allow programming the next tool.
4.11.1 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.
4.12 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 at one corner of the pocket.
- Starting point at the center of the pocket.
- L, H Pocket dimensions.
  - When the starting point of the pocket is one of its corners, the sign indicates the orientation with respect to the XY point.
- Z Part surface coordinate.
- Zs Safety plane coordinate.
- P Total depth.
- α Angle, in degrees, between the pocket and the abscissa axis. The turn is carried out on the defined corner, X,Y point.
Type of corner (icon).

- Square corner with icon.
- Rounded corner with icon.
- Chamfered corner with icon.

$r$  
Rounding radius or chamfer size.

**Roughing parameters:**

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

$\delta$  
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 ($I+$), 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.

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

$Fz$  
Penetration feedrate.

$\beta$  
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.
Canned cycles (M model)

CNC 8065

4.

CANNED CYCLES OF THE EDITOR
Rectangular pocket with rounding.

F Surface milling feedrate.
S Spindle speed.
T Roughing tool.
If programmed T=0, there is no roughing.

Spindle turning direction (icon).
Clockwise.
Counterclockwise.

Machining direction (icon).
Clockwise.
Counterclockwise.

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

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

The finishing operation defining parameters are:
δ Finishing stock on the side walls.
δz Finishing stock at the bottom of the pocket.
Δ Milling pass or width at the bottom of the pocket.
The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.
N Number of penetration passes (steps) for the side finishing. If the resulting step is greater than the cutting length assigned to the table in the tool table, the step will be limited to that value.
θ Penetrating angle.

The penetration is carried out at the feedrate set by roughing parameter "Fz" starting and ending at the center of the pocket. If defined with a value greater than the one assigned to the tool in the tool table, it assumes the table value.

F Surface and side milling feedrate.
S Spindle speed.
T Finishing tool.
D Tool offset.

Spindle turning direction (icon).
- Clockwise.
- Counterclockwise.

Machining direction (icon).
- Clockwise.
- Counterclockwise.

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

Next tool.
- Activating or deactivating the preparation of the next tool.
  The editor will only show this option if the user has configured the editor to allow programming the next tool.
4.12.1 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 \( \delta_z \).
   First, penetration \( l \) at feedrate \( F_z \) at an angle \( \beta \). Then, milling of the pocket surface up to a distance \( \delta \) from the pocket wall. It is carried out at feedrate \( F \) and, if necessary, it recalculates the pass \( \Delta \) 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 \( F_z \) at an angle \( \theta \). Milling of the bottom of the pocket up to a distance \( \delta \) from the pocket wall. It is carried out at finishing feedrate \( F \) and, if necessary, it recalculates the finishing pass \( \Delta \) 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.
4.13 Circular pocket.

Geometric parameters:
\[ X_c, Y_c \quad \text{Center of the pocket.} \]
\[ R \quad \text{Pocket radius.} \]
\[ Z \quad \text{Part surface coordinate.} \]
\[ Z_s \quad \text{Safety plane coordinate.} \]
\[ P \quad \text{Total depth.} \]

Roughing parameters:
The roughing operation empties the pocket leaving the following finishing stocks. Both stocks are defined as finishing parameters.
\[ \delta \quad \text{Finishing stock on the side walls.} \]
\[ \delta_z \quad \text{Finishing stock at the bottom of the pocket.} \]

The roughing operation defining parameters are:
\[ \Delta \quad \text{Maximum milling pass or width.} \]
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.
\[ I \quad \text{Penetration step.} \]
- 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.
In either case, the cycle limits the step to the cutting length assigned to the tool in the tool table.
\[ F_z \quad \text{Penetration feedrate.} \]
Circular pocket.

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.

Surface milling feedrate.

Spindle speed.

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

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:

- $\delta$ Finishing stock on the side walls.
- $\delta z$ Finishing stock at the bottom of the pocket.
- $\Delta$ 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.
- $\theta$ 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.

- $F$ 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.
4.13.1 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 \( \delta z \).
   First, penetration "I" at feedrate "Fz" at an angle "\( \beta \)". Then, milling of the pocket surface up to a distance \( \delta \) from the pocket wall. It is carried out at feedrate "F" and, if necessary, it recalculates the pass (\( \Delta \)) 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 "\( \theta \)". Milling of the bottom of the pocket up to a distance \( \delta \) from the pocket wall. It is carried out at finishing feedrate "F" and, if necessary, it recalculates the finishing pass (\( \Delta \)) 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.
4.14 Pre-empted circular pocket.

Geometric parameters:
- \( X_c, Y_c \): Center of the pocket.
- \( R \): Pocket radius.
- \( r \): Pre-emptying 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.
- \( \delta \): 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 (\( I^+ \)), 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.

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

- 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.
- **Fz** Penetration feedrate.
- **θ** Penetrating angle.

The penetration is carried out along a helical path at the feedrate set by finishing parameter "Fz" starting and ending at the center of the pocket. If defined with a value greater than the one assigned to the tool in the tool table, it assumes the table value.
4. CANNED CYCLES OF THE EDITOR

Pre-empted 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.

F Surface and side milling feedrate.
S Spindle speed.
T Finishing tool.
  - If programmed T=0, there is no finishing.
D Tool offset.

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.

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.
4.14.1 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).
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.
4.15 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.
4. CANNED CYCLES OF THE EDITOR

2D profile pocket.

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 diameter of the drilling tool must not exceed the radius of the roughing tool: or that of the roughing at the bottom if there is no roughing operation.

The cycle calculates the drilling point depending on the programmed profile and the roughing tool.

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.

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

Fz Penetration 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.

F Surface milling feedrate.

S Spindle speed.
Canned cycles (\texttt{M} model)

\begin{itemize}
\item \textbf{T} Roughing tool.
  
  If programmed \texttt{T}=0, there is no roughing.
\end{itemize}

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 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:

- \(\delta\) Finishing stock on the side walls.
- \(\delta z\) Finishing stock at the bottom of the pocket.
- \(\Delta\) 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.

- \(\theta\) Penetrating angle.
  
  The penetration is carried out at the feedrate set by roughing parameter \texttt{"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.
Canned cycles (M-model)

4. CANNED CYCLES OF THE EDITOR

2D profile pocket.

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.

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.

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.
4.15.1 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 P2D) 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.

The executable file is also updated after a software update and when executing or simulating a pocket.
4.15.2 Basic operation.

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

1. Drilling operation. Only if it has been programmed.
2. It selects the roughing tool and starts the spindle in the requested direction.
3. Rapid movement (G0) to the roughing starting point and the safety plane (Zs). Depending on the starting plane, it first moves in XY and then in Z or vice versa.

4. Rapid movement (G0) up to the approach plane.
5. Roughing operation. It is carried out in layers, until reaching the total depth minus the finishing stock at the bottom "δz".
   First, penetration "I" at feedrate "Fz" at an angle "β". 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. And last, rapid withdrawal (G0) up to 1 mm off the machined surface.
6. Rapid withdrawal (G0) up to the safety plane (Zs).
7. It selects the finishing tool and it approaches in rapid (G0) down to 1 mm from the roughed-out bottom.
8. Finishing of the bottom of the pocket.
   First, penetration at feedrate "Fz" at an angle "θ". Then, milling of the bottom of the pocket 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.
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).
4.15.3 Examples of how to define 2D profiles

Profile P.XY  FAGOR 101  [RECALL]

Configuration:

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

Profile:

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

Corners

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

End:

Save profile
### Canned Cycles (M model)

**Canned Cycles of the Editor**  
*(REF: 1209)*

### 2D profile pocket.

<table>
<thead>
<tr>
<th>Profile P.XY</th>
<th>FAGOR 102</th>
<th>[RECALL]</th>
</tr>
</thead>
</table>

#### Configuration:

<table>
<thead>
<tr>
<th>Abscissa axis: X</th>
<th>Ordinate axis: Y</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yes</td>
<td>Validate</td>
</tr>
</tbody>
</table>

#### Profile (outside profile):

<table>
<thead>
<tr>
<th>Starting point</th>
<th>X 20</th>
<th>Y 0</th>
<th>Validate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Straight</td>
<td>X 20</td>
<td>Y -40</td>
<td>Validate</td>
</tr>
<tr>
<td>Straight</td>
<td>X 145</td>
<td>Y -40</td>
<td>Validate</td>
</tr>
<tr>
<td>Straight</td>
<td>X 145</td>
<td>Y 40</td>
<td>Validate</td>
</tr>
<tr>
<td>Straight</td>
<td>X 20</td>
<td>Y 40</td>
<td>Validate</td>
</tr>
</tbody>
</table>

#### Corners

<table>
<thead>
<tr>
<th>Chamfer</th>
<th>[ENTER]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Select the lower left corner</td>
<td>[ENTER]</td>
</tr>
<tr>
<td>Chamfer 15</td>
<td>[ENTER]</td>
</tr>
<tr>
<td>Select the lower right corner</td>
<td>[ENTER]</td>
</tr>
<tr>
<td>Chamfer 15</td>
<td>[ENTER]</td>
</tr>
<tr>
<td>Select the upper right corner</td>
<td>[ENTER]</td>
</tr>
<tr>
<td>Chamfer 15</td>
<td>[ENTER]</td>
</tr>
<tr>
<td>Select the upper left corner</td>
<td>[ENTER]</td>
</tr>
<tr>
<td>Chamfer 15</td>
<td>[ENTER]</td>
</tr>
</tbody>
</table>

[ESC]
Canned cycles (M model)

New profile (island):

<table>
<thead>
<tr>
<th></th>
<th>X 115</th>
<th>Y -25</th>
<th>Validate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Starting point</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Straight</td>
<td>X 115</td>
<td>Y 0</td>
<td>Validate</td>
</tr>
<tr>
<td>Clockwise arc</td>
<td>Xf 90</td>
<td>Yf 25</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Xc 115</td>
<td>Yc 25</td>
<td>R 25</td>
</tr>
<tr>
<td>Straight</td>
<td>X 50</td>
<td>Y 25</td>
<td>Validate</td>
</tr>
<tr>
<td>Straight</td>
<td>X 50</td>
<td>Y 0</td>
<td>Validate</td>
</tr>
<tr>
<td>Clockwise arc</td>
<td>Xf 75</td>
<td>Yf -25</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Xc 50</td>
<td>Yc -25</td>
<td>R 25</td>
</tr>
<tr>
<td>Straight</td>
<td>X 115</td>
<td>Y -25</td>
<td>Validate</td>
</tr>
</tbody>
</table>

End:
Save profile
4.16 '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.

`P3D` 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).
Canned cycles (·M· model)

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.

![Diagram of a 3D profile pocket with islands]

P.Z1 P.Z2 P.Z3 P.Z4
Name of the depth profiles.
They correspond 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).

![Diagram of depth profiles]

All the profiles must be open and without direction changes along their travel (not zigzagging).
Vertical depth profiles for the outside contour and for the islands that reach the surface plane need not be programmed.
The figure shows three programming examples.
When defining the contours in the surface profile, all these cases follow the sequence A-B-C-D.

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

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

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

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

\[ Z \] Part surface coordinate.

\[ Z_s \] Safety plane coordinate.

\[ P \] Total depth.

\section*{Roughing parameters:}

The roughing operation empties the pocket leaving the finishing stock \( \delta \) on the side walls:

This stock is defined as finishing parameter.

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.

\[ I_1 \] 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.

In either case, the cycle limits the step to the cutting length assigned to the tool in the tool table.
4. Canned cycles (·M· model)

Fz Penetration 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.

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.
Canned cycles (·M· model)

The pre-finishing operation defining parameters are:

- **I2** 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.

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.
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.
4.16.1 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.
4.16.2 Basic operation.

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

1. It selects the roughing tool and starts the spindle in the requested direction.
2. Rapid movement (G0) to the roughing starting point and the safety plane (Zs). Depending on the starting plane, it first moves in XY and then in Z or vice versa.
3. Rapid movement (G0) up to the approach plane.
4. Roughing operation. It is carried out in layers until the total depth is reached.
   First, penetration "I1" at feedrate "Fz" at an angle "β". Then, milling of the pocket surface up to a distance "δ" from the pocket wall. It is carried out at feedrate "F" and, if necessary, it recalculates the pass (Δ) so all the passes are identical.
   The pocket is machined following paths concentric to the profile, in the same direction as the outside profile was defined. The islands are machined in the opposite direction.
   And last, rapid withdrawal (G0) up to 1 mm off the machined surface.
5. Rapid withdrawal (G0) up to the approach plane.
6. It selects the pre-finishing tool and starts the spindle in the requested direction.
7. Pre-finishing operation for the side walls. It is carried out in layers until the total depth is reached. The cycle will not run the pre-finishing passes that coincide with any previous roughing pass.
   It is carried out with the pass indicated by "I2" and at the pre-finishing feedrate "F". The outside profile in the same direction that was defined and the islands in the opposite direction.
8. Rapid withdrawal (G0) up to the approach plane.
9. It selects the finishing tool and starts the spindle in the requested direction.
10. Finishing of the side walls. It is carried out with the pass "e" and direction indicated by the icon.
    Rapid withdrawal (G0) up to the safety plane (Zs).
4.16.3 Examples of how to define 3D profiles

Pocket P.3D  FAGOR-A

Profile P.XY  FAGOR 110  [RECALL]

Configuration:
- Abscissa axis: X
- Ordinate axis: Y
- Autozoom: Yes

Profile (outside profile):
- Starting point: X 20  Y 0  Validate
- Straight: X 20  Y -40  Validate
- Straight: X 145  Y -40  Validate
- Straight: X 145  Y 40  Validate
- Straight: X 20  Y 40  Validate
- Straight: X 20  Y 0  Validate

End:
- Save profile

Profile P.Z1  FAGOR 211  Recall
Configuration:

<table>
<thead>
<tr>
<th>Abscissa axis</th>
<th>Ordinate axis</th>
<th>Autozoom</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>Z</td>
<td>Yes</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Validate</td>
</tr>
</tbody>
</table>

Profile (depth profile):

<table>
<thead>
<tr>
<th>Starting point</th>
<th>X 20</th>
<th>Z0</th>
<th>Validate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Straight</td>
<td>X 30</td>
<td>Z-20</td>
<td>Validate</td>
</tr>
</tbody>
</table>

End:

Save profile

Pocket P.3D FAGOR-B

Profile P.XY FAGOR 120 [RECALL]
Canned cycles (M model)

4.

3D profile pocket with islands.

**Configuration:**

<table>
<thead>
<tr>
<th>Abscissa axis: X</th>
<th>Ordinate axis: Y</th>
</tr>
</thead>
<tbody>
<tr>
<td>Autozoom: Yes</td>
<td>Validate</td>
</tr>
</tbody>
</table>

**Profile (outside profile):**

<table>
<thead>
<tr>
<th>Starting point</th>
<th>X 20</th>
<th>Y 0</th>
<th>Validate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Straight</td>
<td>X 20</td>
<td>Y -40</td>
<td>Validate</td>
</tr>
<tr>
<td>Straight</td>
<td>X 145</td>
<td>Y -40</td>
<td>Validate</td>
</tr>
<tr>
<td>Straight</td>
<td>X 145</td>
<td>Y 40</td>
<td>Validate</td>
</tr>
<tr>
<td>Straight</td>
<td>X 20</td>
<td>Y 40</td>
<td>Validate</td>
</tr>
<tr>
<td>Straight</td>
<td>X 20</td>
<td>Y 0</td>
<td>Validate</td>
</tr>
</tbody>
</table>

**New profile (island):**

<table>
<thead>
<tr>
<th>Circle</th>
<th>X 62,5</th>
<th>Y 0</th>
<th>Xc 82,5</th>
<th>Yc 0</th>
<th>Validate</th>
</tr>
</thead>
</table>

**End:**

Save profile

**Profile P.Z1 FAGOR 221 [RECALL]**

**Configuration:**

<table>
<thead>
<tr>
<th>Abscissa axis: X</th>
<th>Ordinate axis: Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>Autozoom: Yes</td>
<td>Validate</td>
</tr>
</tbody>
</table>

**Profile (outside depth profile):**

<table>
<thead>
<tr>
<th>Starting point</th>
<th>X 20</th>
<th>Z0</th>
<th>Validate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Straight</td>
<td>X 30</td>
<td>Z-20</td>
<td>Validate</td>
</tr>
</tbody>
</table>

**End:**

Save profile

**Profile P.Z2 FAGOR 222 Recall**

**Configuration:**

<table>
<thead>
<tr>
<th>Abscissa axis: X</th>
<th>Ordinate axis: Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>Autozoom: Yes</td>
<td>Validate</td>
</tr>
</tbody>
</table>

**Profile (island depth profile):**

<table>
<thead>
<tr>
<th>Starting point</th>
<th>X 62,5</th>
<th>Z-20</th>
<th>Validate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Straight</td>
<td>X 77,5</td>
<td>Z0</td>
<td>Validate</td>
</tr>
</tbody>
</table>

**End:**

Save profile
4.17 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:

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

Type of corner (icon):

- Square corner with icon.
- Rounded corner with icon.
- Chamfered corner with icon.

r Rounding radius or chamfer size.
Roughing parameters:
The roughing operation machines the boss leaving the following finishing stocks. Both stocks are defined as finishing parameters.

\( \delta \)  
Finishing stock on the side walls.

\( \delta z \)  
Finishing stock at the base of the boss.

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

\( F_z \)  
Penetration 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.
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.

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:

- \(\delta\)  
  Finishing stock on the side walls.

- \(\delta z\)  
  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.

- \(F\)  
  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.
4. CANNED CYCLES OF THE EDITOR

Canned cycles (M model)

Rectangular boss.

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.
4.17.1 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.
4.18  Circular boss.

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

Roughing parameters:
The roughing operation machines the boss leaving the following finishing stocks. Both stocks are defined as finishing parameters.
- **δ**  Finishing stock on the side walls.
- **δz**  Finishing stock at the base of the boss.

The roughing operation defining parameters are:
- **Δ**  Maximum milling pass or width.

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

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 boss is machined with the given pass (step) except the last pass that machines the rest.

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

Fz Penetration feedrate.
F Surface milling feedrate.
S Spindle speed.
T Roughing tool.
D Tool offset.

Spindle turning direction (icon).
- Clockwise.
- Counterclockwise.

Machining direction (icon).
- Clockwise.
- Counterclockwise.

Programming of M functions.
- Activating or deactivate 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.
- F Surface and side milling feedrate.
- S Spindle speed.
- T Finishing tool.
- If programmed T=0, there is no finishing.
Canned cycles (·M· model)

4.

CANNED CYCLES OF THE EDITOR
Circular boss:

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.
4.18.1 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 “sz”.

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

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

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

7. Finishing of the base of the boss.

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

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

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

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

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

11. Rapid movement (G0) to the next point.

12. Repeats steps 3, 4, 5, 6, 7, 8, 9, 10.
4.19 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.
Canned cycles (M· model)

Surface milling.

Roughing parameters:
The roughing operation leaves a finishing stock \( \delta z \) defined as finishing parameter. 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.

- \( E \): Overshooting distance of the tool off the surface being milled.

- \( F_z \): Penetration feedrate.

- \( I \): Penetration step.
  - If programmed with a positive sign (\( I^+ \)), the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
  - If programmed with a negative sign (\( I^- \)), the milling is carried out with the given pass (step) except the last pass that machines the rest.
  - 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:

- \( \delta z \): Finishing stock.
- \( \Delta \): 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.

- \( F \): Surface milling feedrate.

- \( S \): Spindle speed.

- \( T \): Roughing tool.
  - If programmed \( T=0 \), there is no roughing.

- \( D \): Tool offset.
4. CANNED CYCLES OF THE EDITOR

Surface milling.

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.
4.19.1 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 \( \delta z \).
   
   First, penetration \( i \) at feedrate \( F_z \). Then, milling at feedrate \( F \) and, if necessary, it recalculates the pass \( \Delta \) 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 \( F_z \). Milling at finishing feedrate \( F \) and, if necessary, it recalculates the finishing pass \( \Delta \) so all the passes are identical.

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

---

4. CANNED CYCLES OF THE EDITOR

Surface milling.
4.20 **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.

**Roughing parameters:**

The roughing operation mills the profile leaving the finishing stock $\delta$. This stock is defined as finishing parameter.
The roughing operation defining parameters are:

- **Fz**: Penetration feedrate.
- **I**: Penetration step.
  - If programmed with a positive sign (+I), the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
  - If programmed with a negative sign (-I), the milling is carried out with the given pass (step) except the last pass that machines the rest.
  In either case, the cycle limits the step to the cutting length assigned to the tool in the tool table.
- **F**: Surface milling feedrate.
- **S**: Spindle speed.
- **T**: Roughing tool.
  - If programmed T=0, there is no roughing.
- **D**: Tool offset.

Spindle turning direction (icon).
- **Clockwise**.
- **Counterclockwise**.

Tool radius compensation (icon).
- **Without compensation**.
- **Left-hand compensation**.
- **Right-hand compensation**.

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

Finishing parameters:
In order to carry out the finishing operation, the roughing must be defined with tool radius compensation. The operation removes the finishing stock (δ).
Canned cycles (M model)

4.

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

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.

  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.
4.20.1 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).
4.21 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:

\begin{itemize}
\item X, Y Profile entry point
\item Z Part surface coordinate.
\item Zs Safety plane coordinate.
\item P Total depth.
\end{itemize}

Roughing parameters:

The roughing operation mills the profile leaving the finishing stock \( \delta \). This stock is defined as finishing parameter.

The roughing operation defining parameters are:

\begin{itemize}
\item Fz Penetration feedrate.
\item I Penetration step.
\end{itemize}

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

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

Tool radius compensation (icon).
   Without compensation.
   Left-hand compensation.
   Right-hand compensation.

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

Finishing parameters:

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

The finishing 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.
4.21.1 Basic operation.

1. It selects the roughing tool and starts the spindle in the requested direction.
2. Rapid movement (G0), up to the XY point and the safety plane (Zs). Depending on the starting plane, it first moves in XY and then in Z or vice versa.
3. Rapid movement (G0) up to the approach plane.
4. Roughing operation. It is carried out in layers until the total depth is reached. First, penetration "I" at feedrate "Fz". Then, profile milling at feedrate "F". If roughing was defined with tool radius compensation, the milling is carried out at a "δ" distance from the wall. And last, rapid movement (G0), first up to the safety plane (Zs) and then to the starting point X1Y1.
5. It selects the finishing tool and starts the spindle in the requested direction.
6. Finishing operation.
7. Penetration to the bottom at feedrate "Fz". Profile milling at feedrate "F".
8. Rapid withdrawal (G0) up to the safety plane (Zs).
4.22 Slot milling.

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

**Geometric parameters:**

Type of slot milling (icon).

There are 6 possible types.

4 for slot mill each corner of the part.

2 for milling a slot across the part.

X, Y Corner where the slot is to be milled.

L, H Slot dimensions.

The sign indicates the orientation referred to the XY point.

Z Part surface coordinate.

Zs Safety plane coordinate.

P Total depth.

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

**Roughing parameters:**

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

δ Finishing stock on the side walls.

δz Finishing stock at the bottom of the pocket.
The roughing operation defining parameters are:

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

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

- **Fz** Penetration feedrate.

- **I** Penetration step.
  - If programmed with a positive sign (\( I^+ \)), the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
  - If programmed with a negative sign (\( I^- \)), the slot milling is carried out with the given pass (step) except the last pass that machines the rest.
  
  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:

- $\delta$: Finishing pass on the side walls.
- $\delta z$: Finishing pass at the bottom.
- $\Delta$: 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.
- $F$: 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.
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.
4.22.1 Basic operation.

1. It selects the roughing tool and starts the spindle in the requested direction.
2. Rapid movement (G0) to the roughing starting point and the safety plane (Zs). Depending on the starting plane, it first moves in XY and then in Z or vice versa.
3. Rapid movement (G0) up to the approach plane.

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

First, penetration "I" at feedrate "Fz". Then, slot milling of the boss surface up to a distance "δ" from the side wall. The slot is carried out at feedrate "F" and, if necessary, it recalculates the pass (Δ) so all the passes are identical. After done roughing, rapid movement (G0), first up to the safety plane (Zs) and then to the starting point. And last, rapid approach (G0) up to 1 mm off the machined surface.
5. Rapid withdrawal (G0) up to the safety plane (Zs).
6. It selects the finishing tool and it approaches in rapid (G0) down to 1 mm from the roughed-out bottom.
7. Finishing of the bottom of the slot.
   Penetration at feedrate "Fz". Milling of the bottom of the slot up to a distance "δ" from the pocket. It is carried out at finishing feedrate "F" and, if necessary, it recalculates the finishing pass (Δ) so all the passes are identical.
8. Rapid withdrawal (G0) up to the safety plane (Zs).
9. Finishing of the side walls. Finishing is carried out in "N" passes at the finishing feedrate "F".
10. Rapid withdrawal (G0) up to the safety plane (Zs).
4.23 Multiple machining in straight line.

Definition format (icon).

There are 5 different ways to define the machining operation.

To select the desired one, place the cursor on the icon and press the space bar.

The number of machining operations "N" must also include the one for the cycle defining point.

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
   Total number of machining operations
   Xn 100, Yn 100
   N 4

2) Angle of the path
   Distance to travel
   α 45
   L 106.066
   N 4

3) Angle of the path
   Total number of machining operations
   Distance between machining operations
   α 45
   N 4
   I 35.3553

4) Coordinates of the end point
   Distance between machining operations
   Xn 100, Yn 100
   I 35.3553

5) Angle of the path
   Distance to travel
   α 45
   Distance between machining operations
   L 106.066
   I 35.3553
4.24 Multiple machining in arc pattern.

Definition format (icon).

There are 9 different ways to define the machining operation.

To select the desired one, place the cursor on the icon and press the space bar.

The movement in arc is made counterclockwise. To do it clockwise, define the angular distance between machining operations $\beta$ with a negative sign.

The number of machining operations "N" must also include the one for the cycle defining point.

**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) Center coordinates $X_a$ 50, $Y_a$ 50
   Total number of machining operations $N$ 7
   Angle of the end point $\tau$ 270

2) Center coordinates $X_a$ 50, $Y_a$ 50
   Total number of machining operations $N$ 7
   Angular distance between machining operations $\beta$ 45

3) Radius $R$ 40
   Total number of machining operations $N$ 7
   Angle of the starting point $\alpha$ 0
   Angle of the end point $\tau$ 270

4) Radius $R$ 40
   Total number of machining operations $N$ 7
   Angle of the starting point $\alpha$ 0
   Angular distance between machining operations $\beta$ 45

5) Center coordinates $X_a$ 50, $Y_a$ 50
   Angle of the end point $\tau$ 270
   Angular distance between machining operations $\beta$ 45

6) Radius $R$ 40
   Angle of the starting point $\alpha$ 0
   Angle of the end point $\tau$ 270
   Angular distance between machining operations $\beta$ 45
7) Center coordinates Xa 50, Ya 50
   Radius R 40
   Total number of machining operations N 7
   Angle of the starting point α 0
   Angular distance between machining operations β 45

8) Center coordinates Xa 50, Ya 50
   Radius R 40
   Total number of machining operations N 7
   Angle of the starting point α 0
   Angle of the end point τ 270
   Angular distance between machining operations β 45

9) Center coordinates Xa 50, Ya 50
   Radius R 40
   Angle of the starting point α 0
   Angle of the end point τ 270
   Angular distance between machining operations β 45
4.25 Multiple machining in rectangular pattern.

Definition format (icon).

There are 3 different ways to define the machining operation. To select the desired one, place the cursor on the icon and press the space bar.

The cycle assumes the lower left point as the starting point. If it is not, define with the proper sign the distances between holes \( lx \) and \( ly \).

The number of machining operations "\( N \)" must also include the one for the cycle defining point.

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) Lengths in \( X, Y \) 
   \( Lx \ 75, Ly \ 50 \) 
   Number of machining operations in \( X \) and \( Y \) 
   \( Nx \ 4, Ny \ 3 \) 
   Rotation angle 
   \( \alpha \ 0 \) 
   Angle between paths 
   \( \beta \ 90 \)

2) Number of machining operations in \( X \) and \( Y \) 
   \( Nx \ 4, Ny \ 3 \) 
   Distance between machining operations in \( X \) and \( Y \) 
   \( lx \ 25, ly \ 25 \) 
   Rotation angle 
   \( \alpha \ 0 \) 
   Angle between paths 
   \( \beta \ 90 \)

3) Lengths in \( X, Y \) 
   \( Lx \ 75, Ly \ 50 \) 
   Distance between machining operations in \( X \) and \( Y \) 
   \( lx \ 25, ly \ 25 \) 
   Rotation angle 
   \( \alpha \ 0 \) 
   Angle between paths 
   \( \beta \ 90 \)
4.26 Multiple machining in grid pattern.

Definition format (icon).

There are 3 different ways to define the machining operation.

To select the desired one, place the cursor on the icon and press the space bar.

The cycle assumes the lower left point as the starting point. If it is not, define with the proper sign the distances between holes lx and ly.

The number of machining operations "N" must also include the one for the cycle defining point.

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) Lengths in X, Y  
   Lx 75, Ly 50  
   Number of machining operations in X and Y  
   Nx 4, Ny 3  
   Rotation angle  
   $\alpha$ 0  
   Angle between paths  
   $\beta$ 90

2) Number of machining operations in X and Y  
   Nx 4, Ny 3  
   Distance between machining operations in X and Y  
   Ix 25, Iy 25  
   Rotation angle  
   $\alpha$ 0  
   Angle between paths  
   $\beta$ 90

3) Lengths in X, Y  
   Lx 75, Ly 50  
   Distance between machining operations in X and Y  
   Ix 25, Iy 25  
   Rotation angle  
   $\alpha$ 0  
   Angle between paths  
   $\beta$ 90
### 4.27 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.
When not using all the points, define the first unused point with the same coordinates as those of the last point of the profile.

**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.
Since there are only 7 points, you must define (P8) = (P7).

- (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
- (P8) X 100 Y 75
Canned cycles (M model)

4. CANNED CYCLES OF THE EDITOR
Random multiple machining.

CNC 8065

(REF: 1209)