CNC 8060 8065

Operating manual (MC).

(Ref: 1709)

FAGOR AUTOMATION

TRANSLATION OF THE ORIGINAL MANUAL

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

MACHINE SAFETY

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

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

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

HARDWARE EXPANSIONS

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

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

COMPUTER VIRUSES

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

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

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

DUAL-USE PRODUCTS

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

FAGOR AUTOMATION

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

The content of this manual and its validity for the product described here has been verified. Even so, involuntary errors are possible, hence no absolute match is guaranteed. However, the contents of this document are regularly checked and updated implementing the necessary corrections in a later edition. We appreciate your suggestions for improvement.

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

INDEX

FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

ABOUT THE PRODUCT - CNC 8060

BASIC CHARACTERISTICS.

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

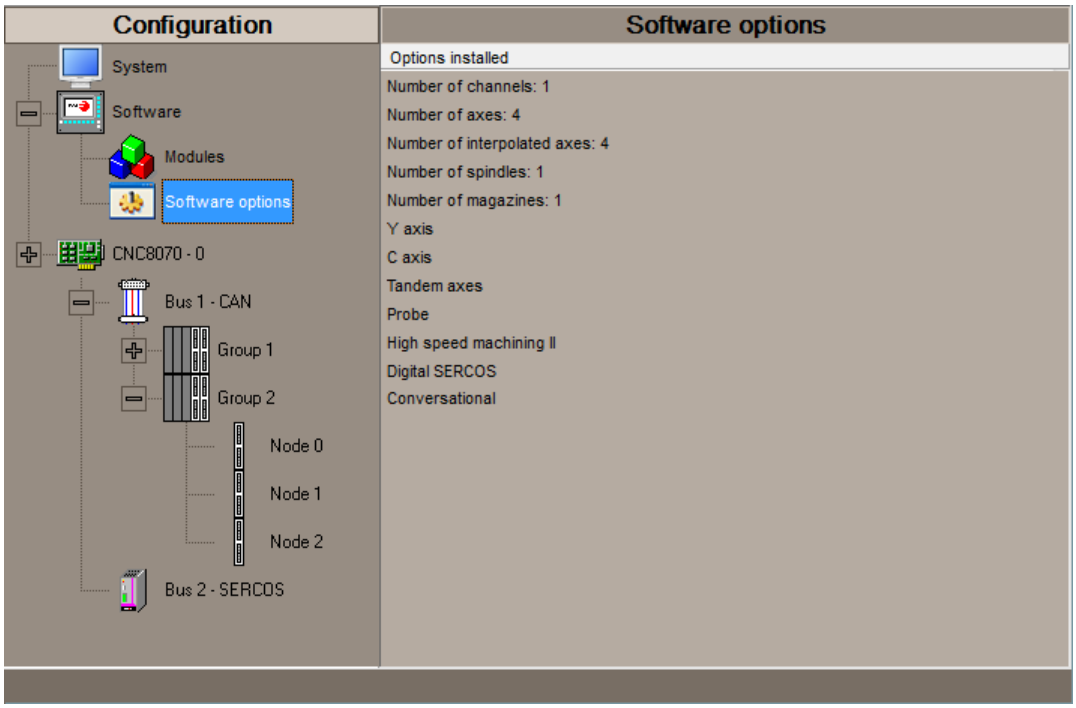
Remote modules.	RIOW	RIO5	RIO70	RIOR	RCS-S
Valid for CNC.	8070 8065 8060	8070 8065 8060	8070 8065 - - -	8070 8065 8060	8070 8065 8060
Communication with the remote modules.	CANopen	CANopen	CANfagor	CANopen	Sercos
Digital inputs per module.	8	24 / 48	16	48	- - -
Digital outputs per module.	8	16 / 32	16	32	- - -
Analog inputs per module.	4	4	8	- - -	- - -
Analog outputs per module.	4	4	4	- - -	4
Inputs for PT100 temperature sensors.	2	2	- - -	- - -	- - -
Feedback inputs.	- - -	- - -	4 (*)	- - -	4 (**)

DISCONTINUED

(*) Differential TTL / Sinusoidal 1 Vpp (**) TTL / Differential TTL / Sinusoidal1 Vpp / SSI protocol / FeeDat / EnDat

SOFTWARE OPTIONS.

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



Consult the ordering handbook for information on the software options available for your model.

SOFT 8060 ADDIT AXES
Additional shaft.
Add axes to the default configuration.

SOFT 8060 ADDIT SPINDLES
Additional spindle.
Add spindles to the default configuration.

SOFT 8060 ADDIT TOOL MAGAZ
Additional tool magazine.
Add tool magazines to the default configuration.

SOFT 8060 ADDIT CHANNELS
Additional channel.
Add channels to the default configuration.

SOFT DIGITAL SERCOS
Sercos digital bus.
Sercos digital bus.

SOFT EDIT/SIMUL
EDISIMU mode (editing and simulation).
It allows for the editing, modification and simulation of a part-program.

SOFT TOOL RADIUS COMP
Compensación de radio.
Tool compensation allows programming the contour to be machined based on part dimensions of the and without taking into account the dimensions of the tool that will be used later on. This avoids having to calculate and define the tool path based on the tool radius.

SOFT PROFILE EDITOR
Profile editor.
Allows for the part profiles to be edited graphically and to import dxf files.

SOFT 60 F3D GRAPHICS
F3D graphics.
High definition solid 3D graphics for the execution and simulation of part-programs and canned cycles of the editor.
During machining, the F3D graphics display the tool removing the material from the part in real time, allowing for the condition of the part to be seen at all times. F3D graphics can display up to 4 views of the part, where each can be rotated, zoomed in or zoomed out. Measurements can also be made on the part and even sections on the piece from any angle.

SOFT 60 IIP CONVERSATIONAL
Interactive Icon-based Pages (conversational mode).
IIP or conversational mode is specifically designed for people without any prior programming knowledge or for those that are not familiarized with Fagor CNC's.
Working in conversational mode is easier than in ISO mode, as it ensures proper data entry and minimizes the number of operations to be defined. There is no need to work with part-programs.

SOFT 60 RTCP
Dynamic RTCP (Rotating Tool Center Point).
The dynamic RTCP option is required for interpolation machining with 4, 5 or 6 axis.

SOFT 60 C AXIS
C axis.
It activates the kinematics for working with the C axis and the associated canned cycles. The CNC can control several C axes. The parameters of each axis indicate if it will function as a C axis or not, where it will not be necessary to activate another axis for the machine parameters.

SOFT 60 Y AXIS
Y axis for lathe.
It activates the kinematics for working with the Y axis and the associated canned cycles.

SOFT 60 TANDEM AXES
Tandem axes.
A tandem axis consists in two motors mechanically coupled (slaved) and making up a single transmission system (axis or spindle). A tandem axis helps provide the necessary torque to move an axis when a single motor is not capable of supplying enough torque to do it.
When activating this feature, it should be kept in mind that for each tandem axis of the machine, another axis must be added to the entire configuration. For example, on a large 3-axis lathe (X Z and tailstock), if the tailstock is a tandem axis, the final purchase order for the machine must indicate 4 axes.

SOFT 60 SYNCHRONISM
Synchronization of axes and spindles.
The axes and ballscrews may be synchronized in two ways: in terms of speed or position. The CNC configuration takes into consideration the synchronization of 2 axes or 2 spindles. Once synchronized, only the master displays and programs the element.

SOFT 60 HSSA I MACHINING SYSTEM
High Speed Surface Accuracy.
This is the new version of algorithms for high speed machining (HSC). This new HSSA algorithm allows for high speed machining optimization, where higher cutting speeds, smoother contours, a better surface finishing and greater precision are achieved.

SOFT 60 HSSA II MACHINING SYSTEM
HSSA-II machining system.
This is the new version of algorithms for high speed machining (HSC). This new HSSA algorithm allows for high speed machining optimization, where higher cutting speeds, smoother contours, a better surface finishing and greater precision are achieved.

SOFT 60 PROBE
Probing canned cycles.
The CNC may have two probes; usually a tabletop probe to calibrate tools and a measuring probe to measure the part.
This option activates the functions G100, G103 and G104 (for probe movements) and probe canned cycles (which help to measure part surfaces and to calibrate tools).
For the laser model, it only activates the non-cycle function G100.

SOFT 60 CONV USER CYCLES
Conversational user cycles.
Incorporation of user cycles in conversational mode.

SOFT 60 PROGTL3
ProGTL3 programming language
Another language apart from ISO for the programming of profiles using a geometric language without the need to use external CAD systems. This language allows for program functions to define lines and circles which define the points of intersection on a profile, in addition to macros for the creation of solids defined by a flat profile and one or more section profiles.

SOFT 60 PPTRANS
Part-program translator.
The program translator may be used to convert programs written in other languages into Fagor ISO codes.

SOFT THIRD PARTY CANOPEN
Third-party CANopen.
Enables the use of non-Fagor CANopen modules.

SOFT MAB SYSTEM.
MAB drives.
Sercos connection with MAB drives.

SOFT 60 PWM CONTROL
Pulse-Width Modulation.
This function is only available for Sercos bus controlled systems. It is mostly oriented toward laser machines for the cutting of very thick sheets, where the CNC generates a series of PWM pulses to control the power of the laser when drilling the starting point.
This feature is essential for cutting very thick sheets and it requires two quick digital outputs located on the central unit. With this new feature, the OEM does not need to install or program any external device, which reduces machine costs and installation times. The end user also benefits, since the “Cutting with PWM ” feature is much easier to use and program.

SOFT 60 GAP CONTROL
Gap control.
This is mostly oriented toward laser machines. Gap control makes it possible to maintain a set distance between the laser nozzle and the surface of the sheet. This distance is calculated by a sensor connected to the CNC, so that the CNC offsets the sensor variations on the distance programmed with additional movements in the axis programmed for the gap.

FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

BLANK PAGE

ABOUT THE PRODUCT - CNC 8065

BASIC CHARACTERISTICS.

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

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

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

Remote modules.	RIOW	RIO5	RIO70	RIOR	RCS-S
Valid for CNC.	8070 8065 8060	8070 8065 8060	8070 8065 - - -	8070 8065 8060	8070 8065 8060
Communication with the remote modules.	CANopen	CANopen	CANfagor	CANopen	Sercos
Digital inputs per module.	8	24 / 48	16	48	- - -
Digital outputs per module.	8	16 / 32	16	32	- - -
Analog inputs per module.	4	4	8	- - -	- - -
Analog outputs per module.	4	4	4	- - -	4
Inputs for PT100 temperature sensors.	2	2	- - -	- - -	- - -
Feedback inputs.	- - -	- - -	4 (*)	- - -	4 (**)

DISCONTINUED

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

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

FAGOR

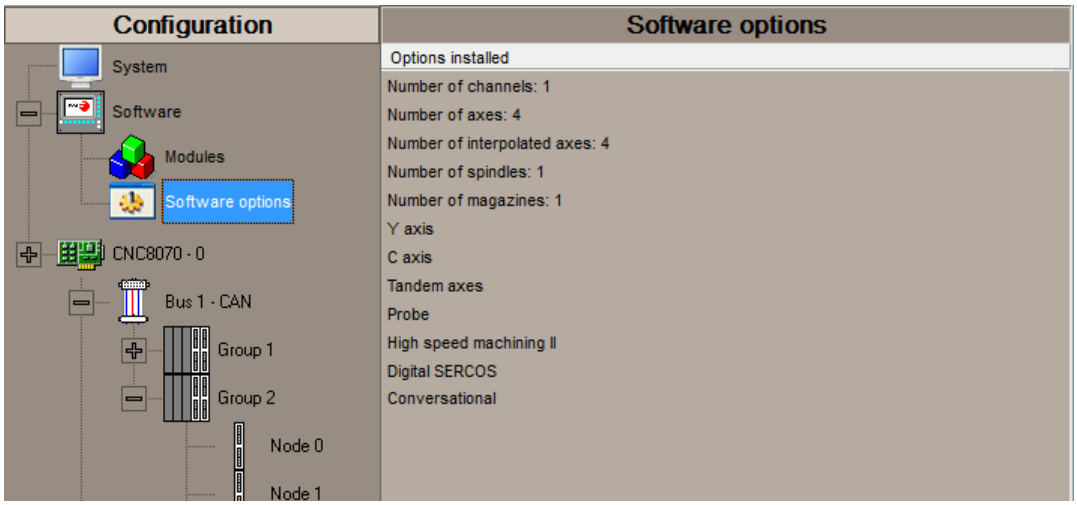
FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

SOFTWARE OPTIONS.

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



Consult the ordering handbook for information on the software options available for your model.

SOFT ADDIT AXES
Additional shaft.
Add axes to the default configuration.

SOFT ADDIT SPINDLES
Additional spindle.
Add spindles to the default configuration.

SOFT ADDIT TOOL MAGAZ
Additional tool magazine.
Add tool magazines to the default configuration.

SOFT ADDIT CHANNELS
Additional channel.
Add channels to the default configuration.

SOFT 4 AXES INTERPOLATION LIMIT
Limited to 4 interpolated axes.
It limits the number of axes to 4, where the CNC can also interpolate these at the same time.

SOFT OPEN SYSTEM
Open system.
The CNC is a closed system that offers all the features needed to machine parts. Nevertheless, at times there are some customers who use third-party applications to take measurements, perform statistics or other tasks apart from machining a part.
This feature must be active when installing this type of application, even if they are Office files. Once the application has been installed, it is recommended to close the CNC in order to prevent the operators from installing other kinds of applications that could slow the system down and affect the machining operations.

SOFT DIGITAL SERCOS
Sercos digital bus.
Sercos digital bus.

SOFT EDIT/SIMUL
EDISIMU mode (editing and simulation).
It allows for the editing, modification and simulation of a part-program.

SOFT DUAL-PURPOSE (M-T)
Dual-purpose machine.
A dual-purpose machine allows for both milling and turning cycles. On lathes with a Y axis, it allows making pockets, bosses and even irregular pockets with islands during milling cycles. Turning cycles can be used by milling machines which have a rotary axis that functions as a C axis.

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

SOFT TOOL RADIUS COMP
Compensación de radio.
Tool compensation allows programming the contour to be machined based on part dimensions of the and without taking into account the dimensions of the tool that will be used later on. This avoids having to calculate and define the tool path based on the tool radius.

SOFT PROFILE EDITOR
Profile editor.
Allows for the part profiles to be edited graphically and to import dxf files.

SOFT IIP CONVERSATIONAL

Interactive Icon-based Pages (conversational mode).

IIP or conversational mode is specifically designed for people without any prior programming knowledge or for those that are not familiarized with Fagor CNC's.

Working in conversational mode is easier than in ISO mode, as it ensures proper data entry and minimizes the number of operations to be defined. There is no need to work with part-programs.

SOFT HD GRAPHICS

HD graphics.

High definition solid 3D graphics for the execution and simulation of part-programs and canned cycles of the editor.

During machining, the HD graphics display, in real time, the tool removing the material from the part, allowing the condition of the part to be seen at all times. HD graphics can display up to 4 views of the part, where each can be rotated, zoomed in or zoomed out. Measurements can also be made on the part and even sections on the piece from any angle.

In a multi-channel system, this feature requires the MP-PLUS (83700201) processor.

SOFT RTCP

Dynamic RTCP (Rotating Tool Center Point).

The dynamic RTCP option is required for interpolation machining with 4, 5 or 6 axis.

This feature requires the MP-PLUS (83700201) processor.

SOFT C AXIS

C axis.

It activates the kinematics for working with the C axis and the associated canned cycles. The CNC can control several C axes. The parameters of each axis indicate if it will function as a C axis or not, where it will not be necessary to activate another axis for the machine parameters.

SOFT Y AXIS

Y axis for lathe.

It activates the kinematics for working with the Y axis and the associated canned cycles.

SOFT TANDEM AXES

Tandem axes.

A tandem axis consists in two motors mechanically coupled (slaved) and making up a single transmission system (axis or spindle). A tandem axis helps provide the necessary torque to move an axis when a single motor is not capable of supplying enough torque to do it.

When activating this feature, it should be kept in mind that for each tandem axis of the machine, another axis must be added to the entire configuration. For example, on a large 3-axis lathe (X Z and tailstock), if the tailstock is a tandem axis, the final purchase order for the machine must indicate 4 axes.

SOFT SYNCHRONISM

Synchronization of axes and spindles.

The axes and ballscrews may be synchronized in two ways: in terms of speed or position. The CNC configuration takes into consideration the synchronization of 2 axes or 2 spindles. Once synchronized, only the master displays and programs the element.

SOFT KINEMATIC CALIBRATION

Kinematics calibration.

This work mode may be used to calibrate kinematics for the first time and to re-calibrate it every so often to correct any possible deviations originated by the day-to-day machine work.

SOFT HSSA II MACHINING SYSTEM

HSSA-II machining system.

This is the new version of algorithms for high speed machining (HSC). This new HSSA algorithm allows for high speed machining optimization, where higher cutting speeds, smoother contours, a better surface finishing and greater precision are achieved.

SOFT TANGENTIAL CONTROL

Tangential control.

"Tangential Control" maintains a rotary axis always in the same orientation with respect to the programmed tool path. The machining path is defined on the axes of the active plane and the CNC maintains the orientation of the rotary axis along the entire tool path.

SOFT PROBE

Probing canned cycles.

The CNC may have two probes; usually a tabletop probe to calibrate tools and a measuring probe to measure the part.

This option activates the functions G100, G103 and G104 (for probe movements) and probe canned cycles (which help to measure part surfaces and to calibrate tools).

SOFT CONV USER CYCLES

Conversational user cycles.

Incorporation of user cycles in conversational mode.

SOFT 70 PROGTL3

ProGTL3 programming language

Another language apart from ISO for the programming of profiles using a geometric language without the need to use external CAD systems. This language allows for program functions to define lines and circles which define the points of intersection on a profile, in addition to macros for the creation of solids defined by a flat profile and one or more section profiles.

SOFT PPTRANS

Part-program translator.

The program translator may be used to convert programs written in other languages into Fagor ISO codes.

SOFT THIRD PARTY CANOPEN

Third-party CANopen.

Enables the use of non-Fagor CANopen modules.

SOFT FVC UP TO 10m3

SOFT FVC MORE TO 10m3

Medium and large volumetric compensation.

5-axis machines are generally used during the manufacturing of large parts. The accuracy of the parts is limited by the machine manufacturing tolerances and is effected by temperature variations during machining.

In sectors such as the aerospace industry, machining demands mean that classic compensation tools are becoming suboptimal. Volumetric compensation FVC comes in to complement the machine adjusting tools. When mapping the total work volume of the machine, the CNC knows the exact position of the tool at all times. After applying the required compensation, the resulting part is made with the desired precision and tolerance.

There are 2 choices, which depend on the size of the machine, being up to 10 m^3 and over 10 m^3.

DECLARATION OF CE CONFORMITY AND WARRANTY CONDITIONS

DECLARATION OF CONFORMITY

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

WARRANTY TERMS

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

BLANK PAGE

VERSION HISTORY - CNC 8060

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

Ref. 1402

Software V01.00
First version.

Ref. 1412

Software V01.10
User cycles.

Software V01.10.03
The editor lets associate a multiple machining operation with the 2D profile pocket cycle

Ref. 1709

Software V01.60
Simulation options.

BLANK PAGE

VERSION HISTORY - CNC 8065

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

Ref. 1103

Software V04.20
First version.

Ref. 1201

Software V04.22
Canned cycles. Point-to-point profile milling. The table to define the points of the profile admits 25 points.
Canned cycles. Point-to-point profile milling. New icon to delete all the points of the table.
Canned cycles. The [DEL] key deletes a profile from the list.
Canned cycles. Pressing [RECALL] on a tool gives access to the tool table.
Keyboard shortcuts [CTRL][C] and [CTRL][V] may be used on the programs list to copy and paste a program.
Selecting a program for editing no longer involves selecting it also for execution. Use the "Execute Program" softkey to select a program for execution.

Ref. 1209

Software V04.24
Canned cycles. Simple rectangular pocket. The starting point of the pocket may be in its center.
Canned cycles. Rectangular pocket with rounding. The starting point of the pocket may be in its center.

Ref. 1709

Software V05.60
Simulation options.

BLANK PAGE

SAFETY CONDITIONS

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

Before start-up, verify that the machine that integrates this CNC meets the 2006/42/EC Directive.

PRECAUTIONS BEFORE CLEANING THE UNIT

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

PRECAUTIONS DURING REPAIRS

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

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

PRECAUTIONS AGAINST PERSONAL HARM

Interconnection of modules.	Use the connection cables provided with the unit.
Use proper cables.	To prevent risks, only use cables and Sercos fiber recommended for this unit. To prevent a risk of electrical shock at the central unit, use the proper connector (supplied by Fagor); use a three-prong power cable (one of them being ground).
Avoid electric shocks.	To prevent electrical shock and fire risk, do not apply electrical voltage out of the indicated range.
Ground connection.	In order to avoid electrical discharges, connect the ground terminals of all the modules to the main ground terminal. Also, before connecting the inputs and outputs of this product, make sure that the ground connection has been done. In order to avoid electrical shock, before turning the unit on verify that the ground connection is properly made.
Do not work in humid environments.	In order to avoid electrical discharges, always work with a relative humidity (non-condensing).
Do not work in explosive environments.	In order to avoid risks, harm or damages, do not work in explosive environments.

PRECAUTIONS AGAINST DAMAGE TO THE PRODUCT

Work environment.

This unit is ready to be used in industrial environments complying with the directives and regulations effective in the European Community.

Fagor Automation shall not be held responsible for any damage suffered or caused by the CNC when installed in other environments (residential, homes, etc.).

Install this unit in the proper place.

It is recommended, whenever possible, to install the CNC away from coolants, chemical product, blows, etc. that could damage it.

This unit meets the European directives on electromagnetic compatibility. Nevertheless, it is recommended to keep it away from sources of electromagnetic disturbance such as:

- Powerful loads connected to the same mains as the unit.
- Nearby portable transmitters (radio-telephones, Ham radio transmitters).
- Nearby radio / TC transmitters.
- Nearby arc welding machines.
- Nearby high voltage lines.

Enclosures.

It is up to the manufacturer to guarantee that the enclosure where the unit has been installed meets all the relevant directives of the European Union.

Avoid disturbances coming from the machine.

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

Use the proper power supply.

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

Connecting the power supply to ground.

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

Analog inputs and outputs connection.

Use shielded cables connecting all their meshes to the corresponding pin.

Ambient conditions.

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

Central unit enclosure.

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

Power switch.

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

SAFETY SYMBOLS

Symbols that may appear in the manual.

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

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

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

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

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

Symbols that the product may carry.

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

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

BLANK PAGE

RETURNING CONDITIONS

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

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

BLANK PAGE

CNC MAINTENANCE

CLEANING

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

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

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

PRECAUTIONS BEFORE CLEANING THE UNIT

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

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

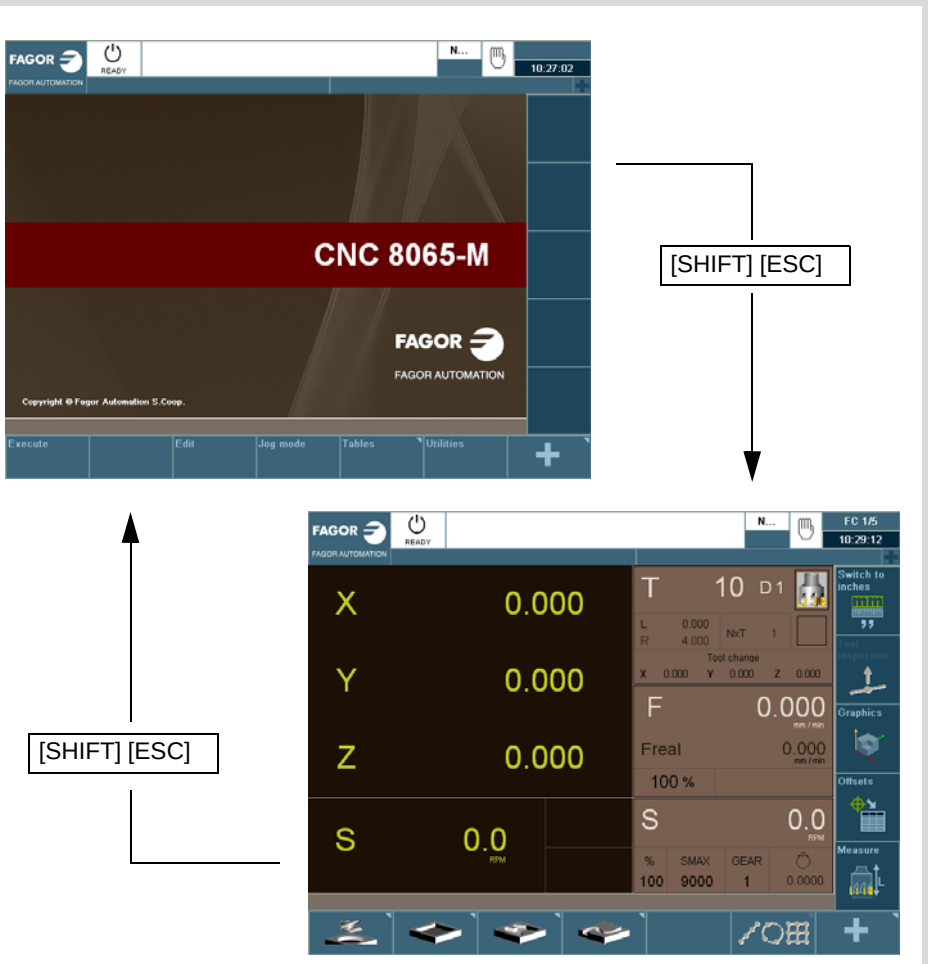
BLANK PAGE

GENERAL CONCEPTS

1

1.1 Accessing the conversational mode

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



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

1.

GENERAL CONCEPTS

Accessing the conversational mode

FAGOR

FAGOR AUTOMATION

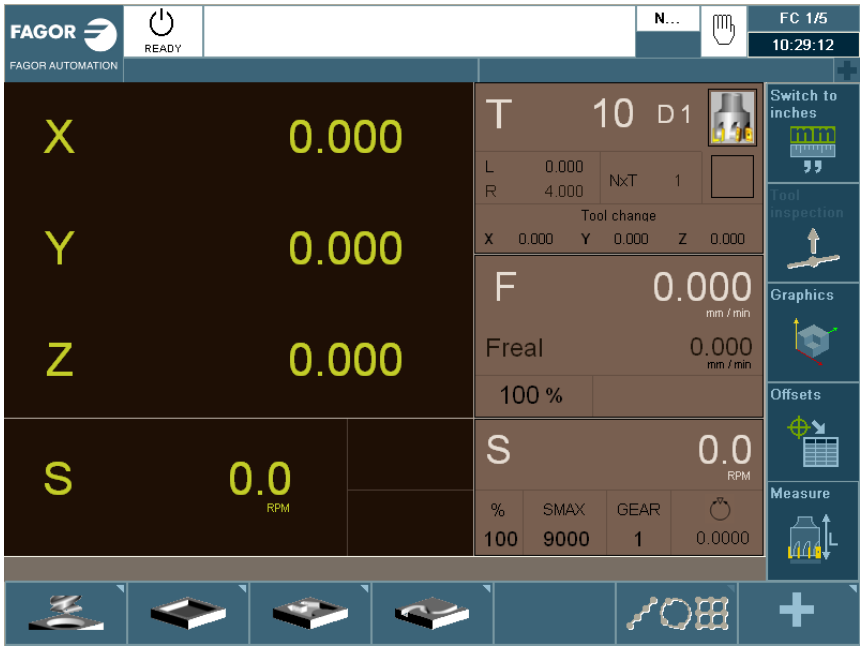
CNC 8060
CNC 8065

(REF: 1709)

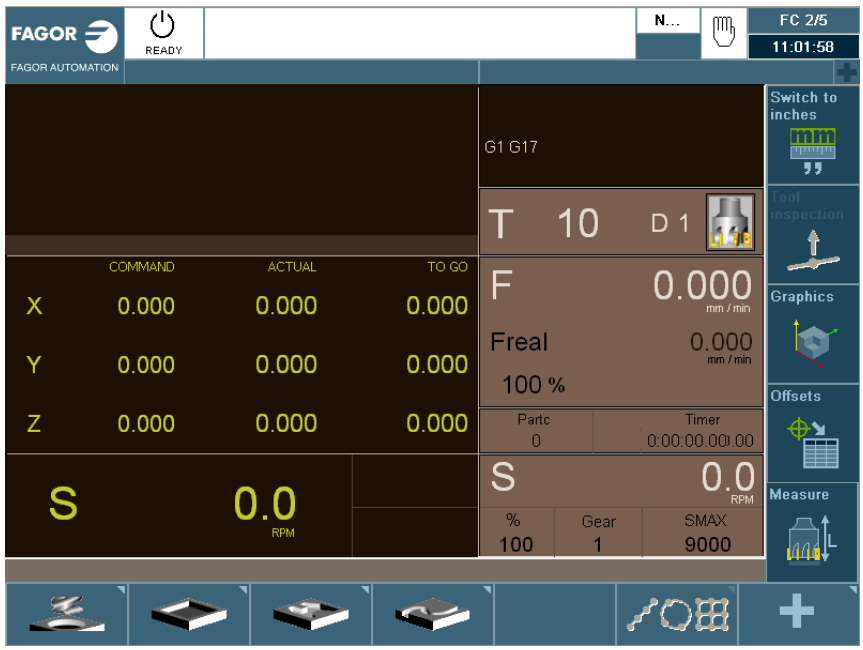
OPERATING IN JOG MODE

2

The standard screen of the MC mode is the following:



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



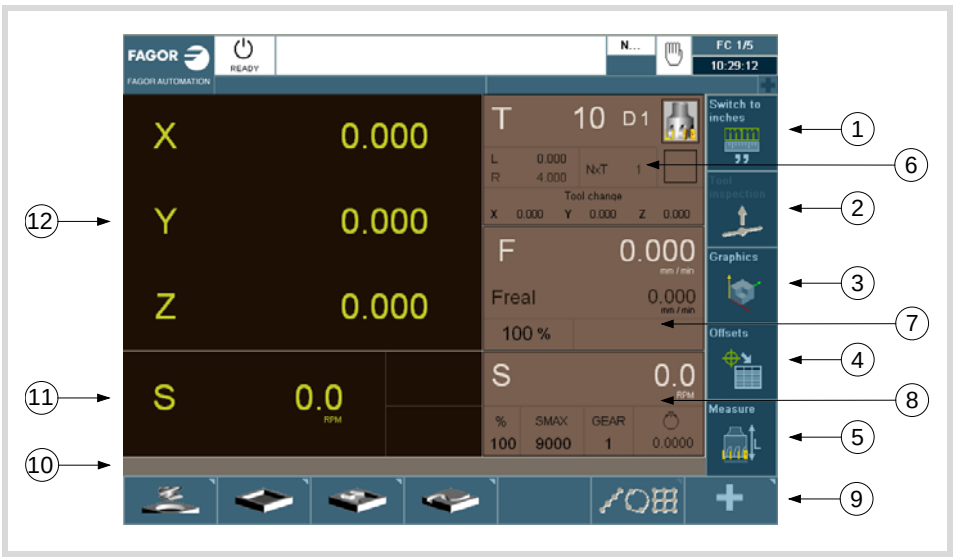
FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

2.1 Introduction

2.1.1 Standard screen of the conversational mode



1 Softkey for selecting units mm/inches.

2 Softkey to go into tool inspection.

3 Softkey to access the graphics in execution mode.

4 Softkey for selecting OFFSETS.

5 Softkey for tool calibration.

6 Window that shows:

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

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

8 Window showing spindle related information:

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

9 Softkeys for cycle editing.

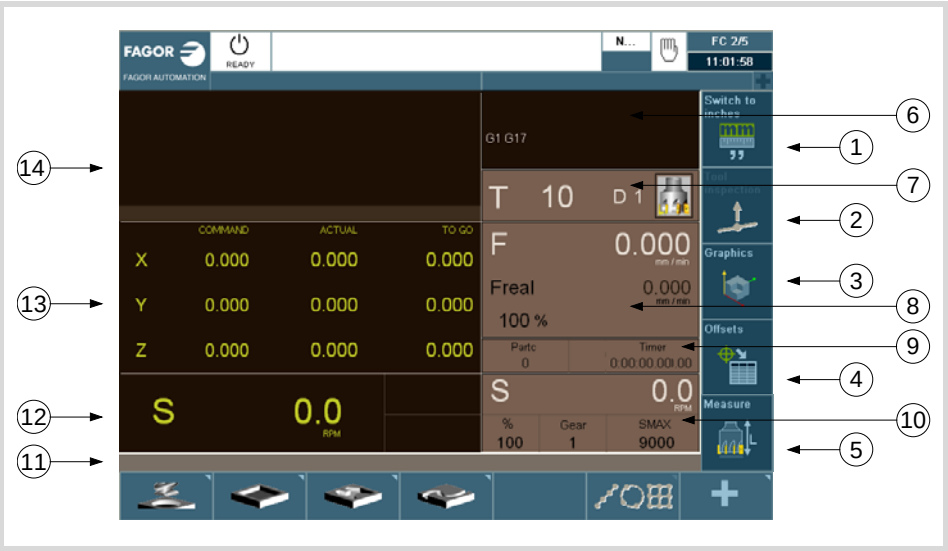
10 Message bar.

11 Real spindle rpm.

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

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

2.1.2 Auxiliary screen of the conversational mode



1 Softkey for selecting units mm/inches.

2 Softkey to go into tool inspection.

3 Softkey to access the graphics in execution mode.

4 Softkey for selecting OFFSETS.

5 Softkey for tool calibration.

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

7 Window that shows:

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

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

9 Window that shows the value of the variables:

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

10 Window with spindle related information:

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

11 Message bar.

12 Window with spindle related information:

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

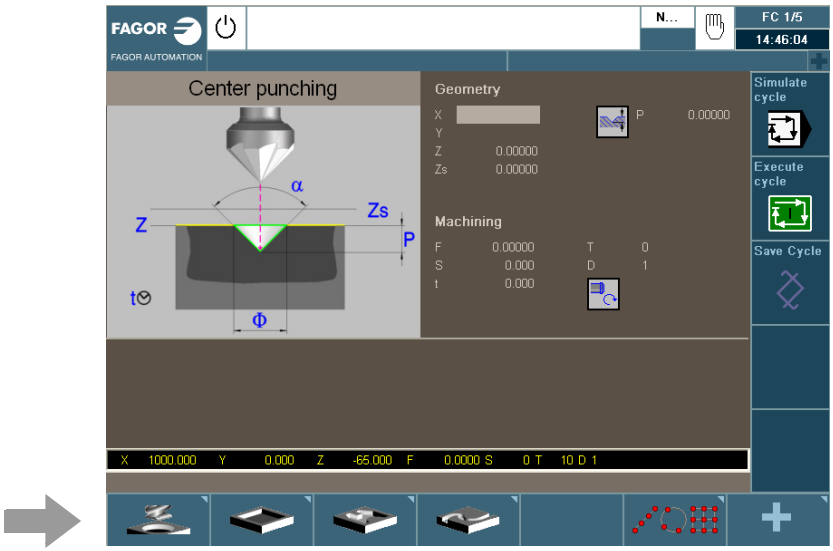
13 Window with axis related information:

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

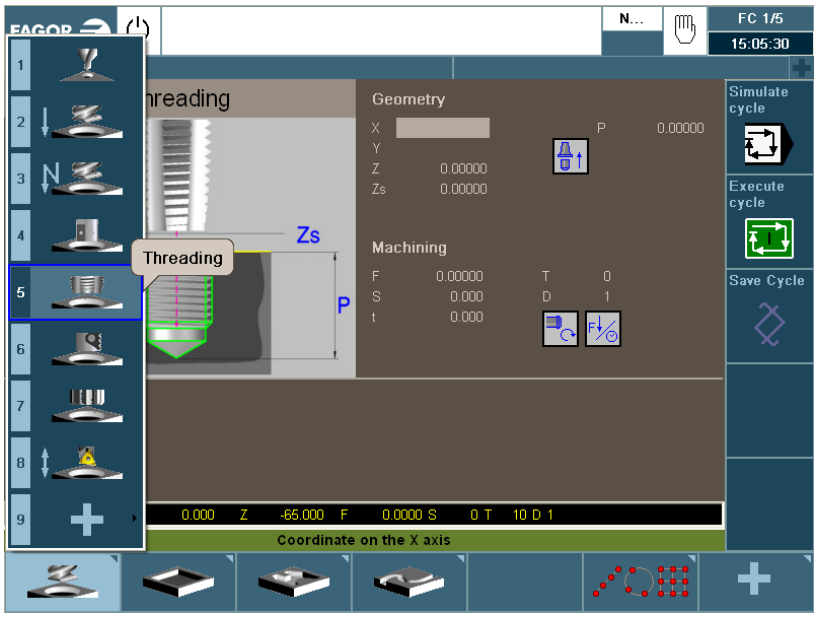
14 Window that shows the lines of the program being executed.

2.1.3 Cycle editing

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



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



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

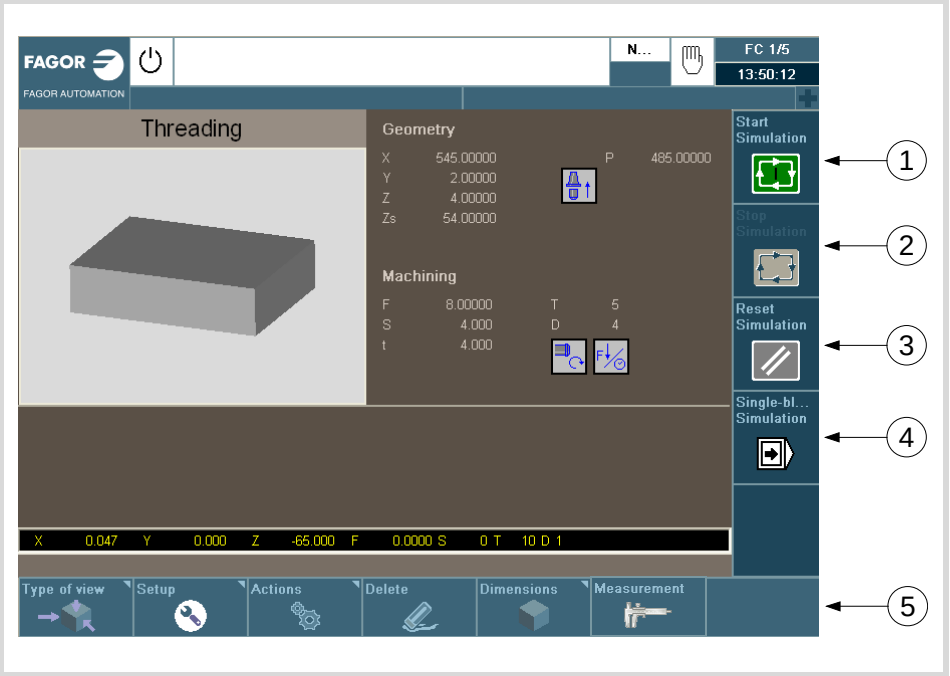
For further information on editing cycles, see the chapter *"3 Working with operations or cycles"*.

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

For further information on saving cycles, see the chapter *"4 Saving programs"*.

2.1.4 Cycle simulation

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



1 Softkey to start cycle simulation.

2 Softkey to stop cycle simulation.

3 Softkey to reset the simulation.

4 Softkey to simulate the cycle block by block.

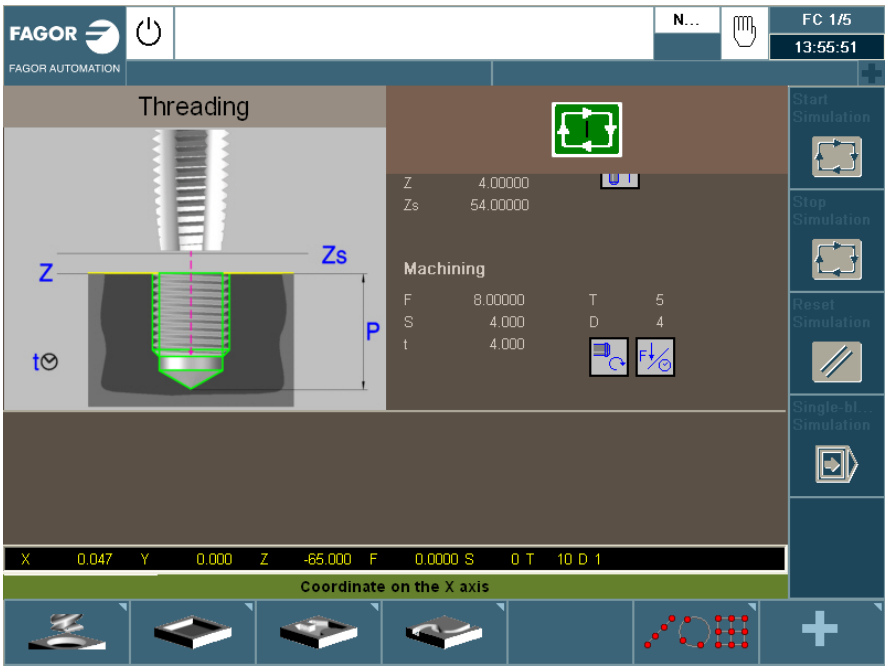
5 The horizontal softkeys may be used to configure how to display the simulated cycle.

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

2.1.5 Cycle execution

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

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



2.2 Operations with the axes.

2.2.1 Home search.

Home search is the operation used to synchronize the system. This operation must be carried out when the CNC loses the position of the origin point (e.g. by turning the machine off).

When "searching home", the axes move to the machine reference point and the CNC assumes the coordinate values assigned to that point by the machine manufacturer, referred to machine zero. When using distance-coded reference marks or absolute feedback, the axes will only move the distance necessary to verify their position.

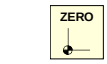
The axes may be homed manually (axis by axis from the operator panel) or automatically (using a subroutine).

Manual home search (one axis at a time).

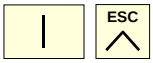
The axis-by-axis home search cancels the zero offset, the fixture offset and the measuring offset. The CNC assumes the machine reference zero point (home) as the new part zero.

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

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



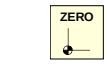
2 Press the homing key [ZERO]. The CNC will display the "1" symbol in the numeric area indicating that a home search will take place.



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

Automatic home search (with subroutine).

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

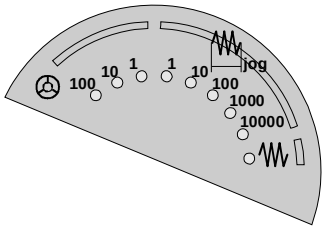


1 Press the homing key [ZERO]. The CNC will display a dialog box requesting confirmation to execute the home search.

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

2.2.2 Jog

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



Continuous jog

Incremental jog

Handwheels

The jog keyboard and the feedrate selector

JOG keypad.

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

X+	Y+	Z+
X-	Y-	Z-
7+	∿	7-

The keypad has two keys for each axis. One to jog the axis in the positive direction and another one to move it in the negative direction.

To move a single axis, press the axis key and the one for its jogging direction.

X	Y	Z
4	5	6
+	∿	—

The keypad has a key for each axis and two keys for moving direction, common to all the axes.

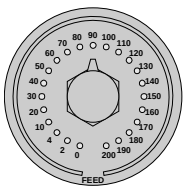
To jog an axis requires activating both the axis key and the moving direction. There are two options, depending on how the jog keyboard has been configured.

- The axis will move while both keys are pressed, the axis key and the direction key.
- When pressing the axis key, the key remains active. The axis will move while the direction key is kept pressed. To de-select the axis, press [ESC] or [STOP].

User keys as jog keys

The CNC offers the OEM the possibility to enable the user keys as jog keys. The user keys defined this way behave like the jog keys.

Feedrate selector.



The movement is carried out at the feedrate defined by the OEM. The feedrate may be varied between 0% and 200% using the feedrate override switch on the operator panel.

Movement in continuous jog.

In continuous jog, the axes keep moving while the jog keyboard is acted upon. Continuous jog allows moving several axes at the same time.

1 Turn the jog selector switch of the operator panel to the continuous jog position on the dial.

2 Jog the desired axis using the JOG panel (keypad). If while moving, a second axis is selected, the new one will move at the same time and under the same conditions.

If while the axes are moving, the rapid key is pressed, the axes will move at the rapid rate set by the machine manufacturer. This feedrate will be applied while that key is kept pressed and, when released, the axes will recover their previous feedrate. This rapid rate may be varied between 0% and 200% with the feedrate override switch on the operator panel.

Movement in incremental jog.

In incremental jog, the axis moves a specific distance every time the key is pressed. In incremental jog, the axes may be jogged simultaneously.

1 Turn the jog selector switch of the operator panel to one of the incremental jog positions. Each position will move the axis a fixed distance; the typical values are the following.

Position.	Movement for each key push.
1	0.001 mm or 0.0001 inch.
10	0.010 mm or 0.0010 inches.
100	0.100 mm or 0.0100 inches.
1000	1.000 mm or 0.1000 inches.
10000	10.000 mm or 1.0000 inches.

2 Jog the desired axis using the JOG panel (keypad). Every time the JOG panel is acted upon, the axis will move the distance indicated on the dial of the jog selector switch. If while moving, a second axis is selected, the new one will move at the same time and under the same conditions.

2.2.3 Jogging the axes with handwheels

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

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

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

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

General handwheel

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

- If there are several axes selected in handwheel mode, the general handwheel will move all of them.
- If an axis has been selected which has an individual handwheel selected with it, this axis may be moved with the general handwheel, with the individual one or with both at the same time. When using both handwheels simultaneously, the CNC will add or subtract the pulses provided by both handwheels depending on which direction they are turned.
- If the CNC has several general handwheels, any of them can move the axes selected in handwheel mode. When using several handwheels simultaneously, each axis involved will be applied the sum of the increments of all the handwheels.

These are the steps to follow for moving one or several axes with the general handwheel.

1. Select the axis or axes to be jogged. The CNC will highlight the selected axes. When selecting an axis or quitting the handwheel mode using the movement selector, the previous one is automatically deselected.
2. Once the axis has been selected, the CNC will move it as the handwheel is turned depending on the setting of the selector switch and on the turning direction of the handwheel.

The feedrate depends on how fast the handwheel is turned.

Selecting the axes to be jogged

There are two ways to select the axes.

1. On the JOG keyboard, press one of the keys for the axis to be jogged. Selecting an axis de-selects the previous one. To select several axes, press one of the keys of each key at the same time.

   An axis needs not belong to the active channel in order to be selected. An axis from one channel may be set in handwheel mode from another channel, if the channel of the axis is also in jog mode.

2. When using a handwheel with a push-button, the push-button may be used to select, sequentially, the axes to be jogged. Pushing the button selects the first one of the axes being displayed. If an axis has already been selected, it de-selects it and selects the next one. If it was the last one, it selects the first one again.

   Only the axes being displayed in the active channel may be selected, regardless of the channel they belong to. The axes of another channel cannot be selected if they are not being displayed.

An axis is de-selected when quitting the handwheel mode using the movement selector and after a reset. If an axis has been set in handwheel mode from the PLC, it can only be deactivated from the PLC; a reset does not deactivate it.

Selecting an axis from the automatic mode

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

Individual handwheel

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

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

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

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

Feed handwheel.

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

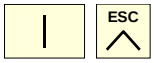
This feature must be managed from the PLC. Usually, this feature is turned on and off using an external push button or key configured for that purpose.

2.2.4 Moving an axis to a particular position (coordinate)

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

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

2 Enter the coordinate of the target point.



3 Press [START] to execute the movement or [ESC] to cancel the operation.

Feedrate behavior

The moving feedrate depends on whether G00 or G01 is active. This feedrate may be varied between 0% and 200% using the feedrate override switch on the operator panel. The percentage will be applied on to all the movements carried out in G00 and in G01.

- If G00 is active, the movement is carried out at the rapid rate defined by the machine manufacturer.
- If G01 is active, the movement is carried out at the active feedrate. If no feedrate is active, the movement is executed at the feedrate defined by the machine manufacturer.

2.2.5 Coordinate preset

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

1 Use the alphanumeric keyboard to select the axis whose position value (coordinate) is to be preset. The CNC will highlight that axis to indicate that it is selected.

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

2 Key in the desired preset value.

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

2.3 Spindle control

Displaying the data of several spindles.

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

Spindle control

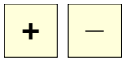
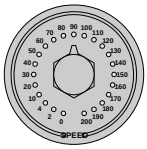
The spindle may be controlled manually using the following keys of the operator panel. The keys always refer to the master spindle of the active channel.

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

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

Vary the speed override from the operator panel.

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

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

2.4 Tool selection and tool change

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

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

2.5 Setting the feedrate and spindle speed.

Setting a new feedrate in the channel.

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

Follow these steps to set a new feedrate.

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

Setting a new spindle speed.

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

Follow these steps to set a new spindle speed.

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

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

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

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

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

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

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

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

2.7 Tool calibration

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

	Tool calibration in a mill model.

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

Types of calibration

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


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

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

Manual calibration. Calibration without a probe.

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

Semi-automatic calibration. Calibration with a probe.

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

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

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

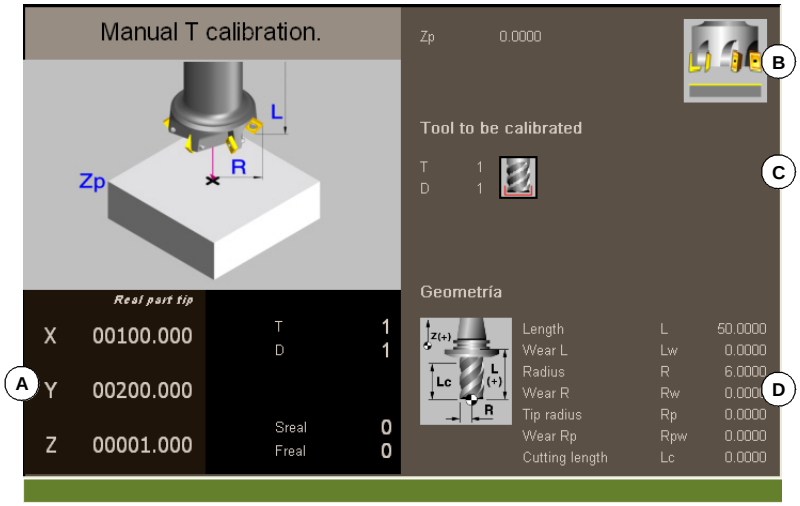
Probe selection

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

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

2.7.1 Manual calibration. Calibration without a probe

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



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

B Data of the part used for calibration and drawing showing that calibration is possible. If the window does not show this drawing, some of the data is missing.

C Necessary data for calibration.

D Tool data.

Tool calibration

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

Selecting a tool

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

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

Tool calibration

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

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

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

Validating the calibration.

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

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

Definition of data

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

For a lathe tool.

Data	Meaning
Zp Xp	Dimensions of the reference part being used in the calibration. These coordinates are referred to the main axes of the tool.
T	Tool to be calibrated.
D	Tool offset to be calibrated.
Off Y Off X	Tool offsets on each axis.
Lw	Offset wear on each axis.

For a milling tool.

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

Tool calibration steps

To calibrate the length, follow these steps.

1 Define the dimensions of the reference part being used in the calibration.

2 Select the tool and the offset to be calibrated. After the selection, the CNC shows the dimensions defined in the tool table for that offset.

To calibrate a tool, it must be the active tool. When selecting a tool and pressing [ENTER], the CNC only shows the data for that tool. Press [CYCLE START] for the CNC to make the tool change so it becomes the active tool. See *"Selecting a tool"* on page 46.

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

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

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

To calibrate another tool, repeat steps 2 and 3.

Considerations for the offsets and their wear.

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

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

Sign criteria for the offsets and their wear.

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

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

Entering wear values incrementally or absolutely.

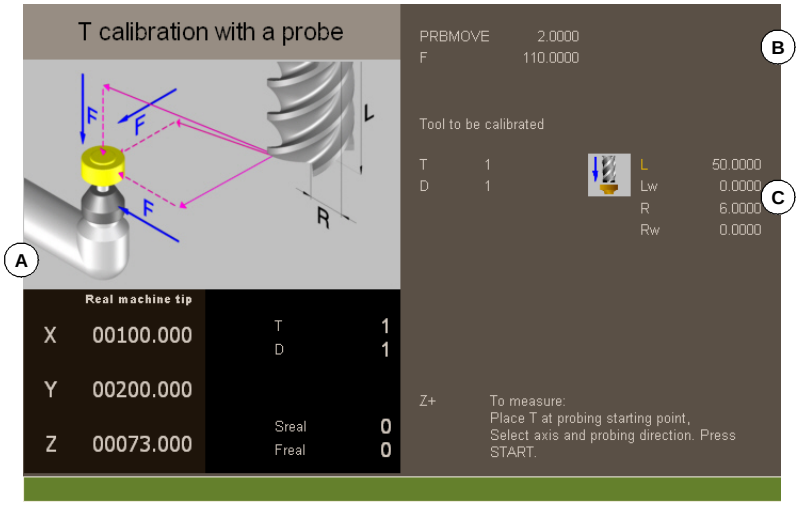
In the tool table, it is possible to define whether the wear value being entered must be incremental or absolute.

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

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

2.7.2 Semi-automatic calibration. Calibration with a probe

This option is only available when using a tabletop probe installed on the machine. On a milling model, it may be used to calibrate the length and radius of the milling tools and the offsets of the lathe tools.



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

B Data of probing movement.

C Necessary data for calibration.

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

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

Tool calibration

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

Selecting a tool

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

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

Tool calibration

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

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

Validating the calibration

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

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

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

Definition of data

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

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

Tool calibration steps

To calibrate the tool, follow these steps:

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

   To calibrate a tool, it must be the active tool. When selecting a tool and pressing [ENTER], the CNC only shows the data for that tool. Press [CYCLE START] for the CNC to make the tool change so it becomes the active tool. See *"Tool calibration"* on page 49.
3. Manually approach the tool to the probe until it is placed on the path that will be used for probing. To calibrate the radius with a cylindrical probe, the path must coincide with the probe's center point; if not, the radius will be calculated wrong.
4. Calibrate the tool. Select the axis and the probing direction on the softkey menu and press [START].

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

Considerations for the offsets and their wear.

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

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

Sign criteria for the offsets and their wear.

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

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

Entering wear values incrementally or absolutely.

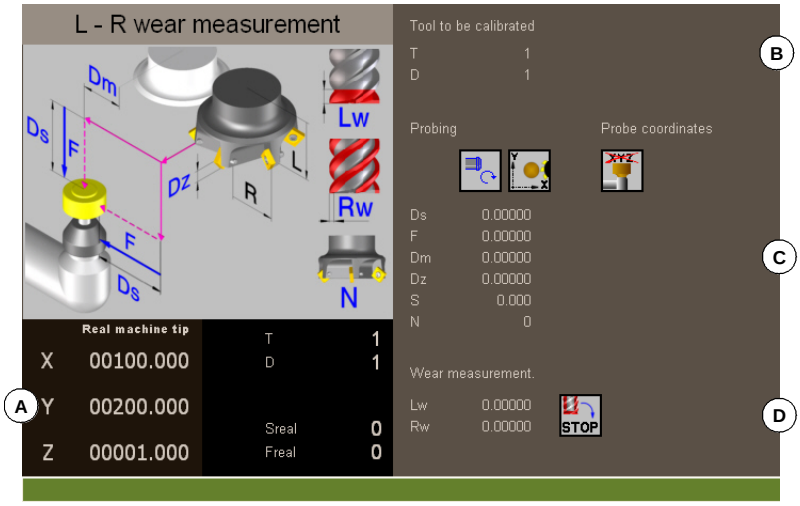
In the tool table, it is possible to define whether the wear value being entered must be incremental or absolute.

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

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

2.7.3 Automatic calibration with a probe and a canned cycle

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



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

B Tool to be calibrated.

C Data for probe calibration and position.

D Data for tool wear measurement.

Tool calibration

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

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

Selecting a tool

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

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

Select another position for the probe.

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

Tool calibration

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

- Calibrate the offsets and resets the wears to zero.

- Calibrate the length and radius and measure the wears.

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

Definition of data

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

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

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

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

To calibrate the offsets of a milling or lathe tool.

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

Tool calibration steps

To calibrate the length, follow these steps.

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

WORKING WITH OPERATIONS OR CYCLES

3

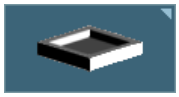
3.1 General concepts.

Select the machining cycles.

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

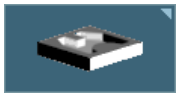
Z axis machining.

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



Pockets / Bosses.

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

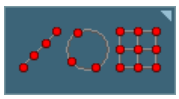


2D/3D profile pockets.

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

Roughing.

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



Multiple machining.

Points in line, points in arc, points in a rectangular pattern, points in a grid pattern, points in a random pattern (several points defined by the user). Multiple machining may be associated with canned cycles so it can be repeated in several points.

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 *"3.1.3 Teach-in mode."* on page 58.

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).

3.1.1 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.

Function.	Canned cycle.
G281	Center punching.
G282	Drilling.
G283	Deep hole drilling.
G284	Tapping.
G285	Reaming.
G286	Boring.
G287	Rectangular pocket with rounding.
G288	Circular pocket.
G289	Simple rectangular pocket.
G290	Surface milling.
G291	Rectangular boss.
G292	Circular boss.
G293	Point-to-point profile milling.
G294	Free profile milling.
G295	Slot milling.
G296	Pre-emptied circular pocket.
G297	Boring with spindle orientation.
G298	Bore milling.
G299	Thread milling.

3.1.2 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 one of them, define it 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.

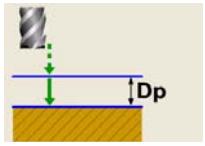
Programming the next tool.

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.

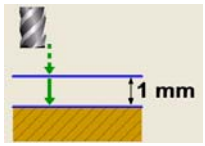
On the cycle screens, the display must be activated in order to see and define the next tool; otherwise, the data will not be displayed.

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.

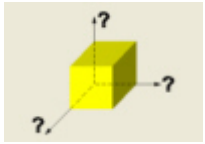


This option is enabled. The cycles show parameter ·Dp· for programming the part surface approach distance.



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

Select axis configuration.



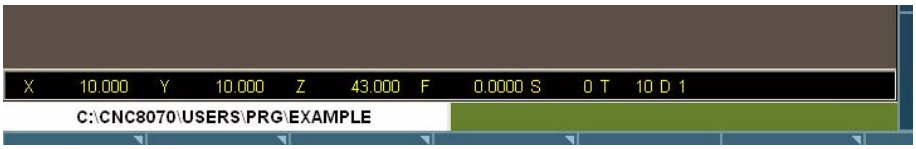
Setting an axis configuration for the cycle editor. The defined axis configuration is only good for making it easier to edit the cycle, because it shows the data related to coordinates and plans according to the selected axis configuration.

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

3.1.3 Teach-in mode.

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

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



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

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

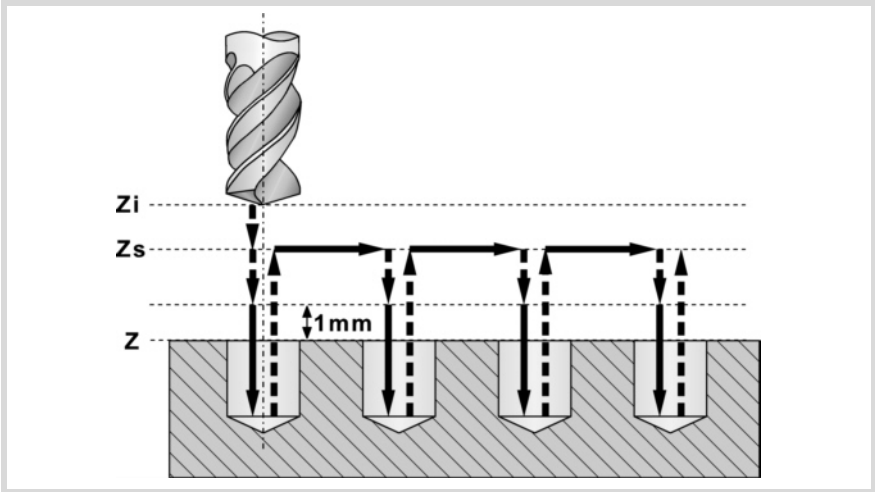
3.1.4 Work planes and shifting of the machining operations.

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.

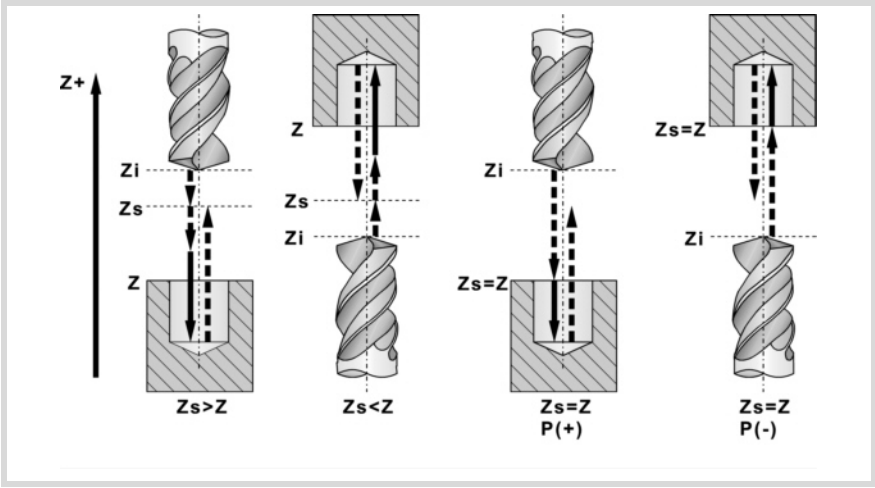
These are the four work planes available in all operations (assuming the Z axis as longitudinal).

- 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.

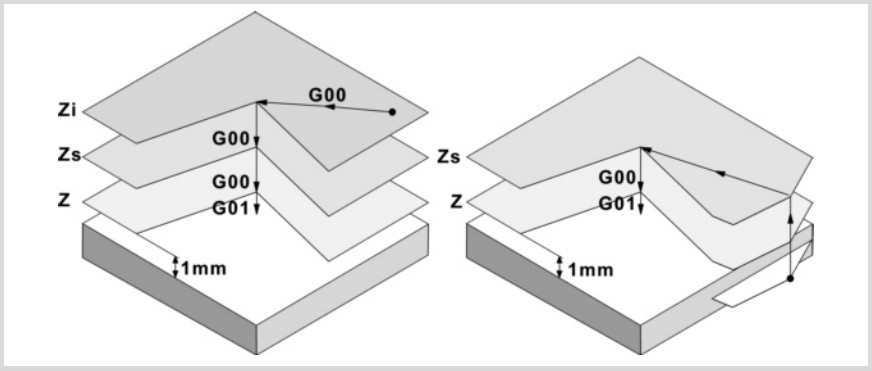
The machining direction is set by the position of the part surface (Z) and of the safety plane (Zs). If they are both the same, the direction is set by the sign of total machining depth (parameter P). If Z=Zs and P>0 machining in the negative direction of the longitudinal axis (Z-), if Z=Zs and P<0 machining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



Movments in the work planes.

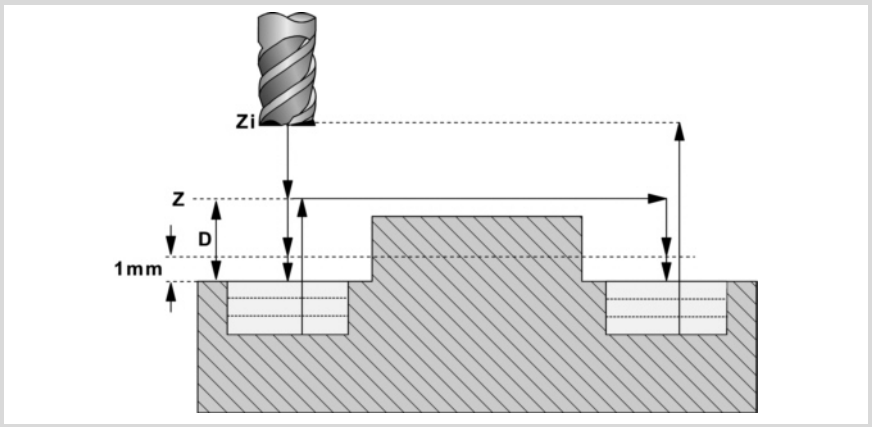
When beginning to execute the cycle, the tool moves in rapid (G0) from the starting plane (Zi) to the safety plane (Zs).

- 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.



3.1.5 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.

Go to the tool table.

Pressing [RECALL] on a tool gives access to the tool table and to the data of that tool.

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.

3.1.6 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.

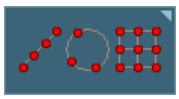
An error will be issued if they are different.

3.1.7 Associate a multiple machining operation with a canned cycle

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

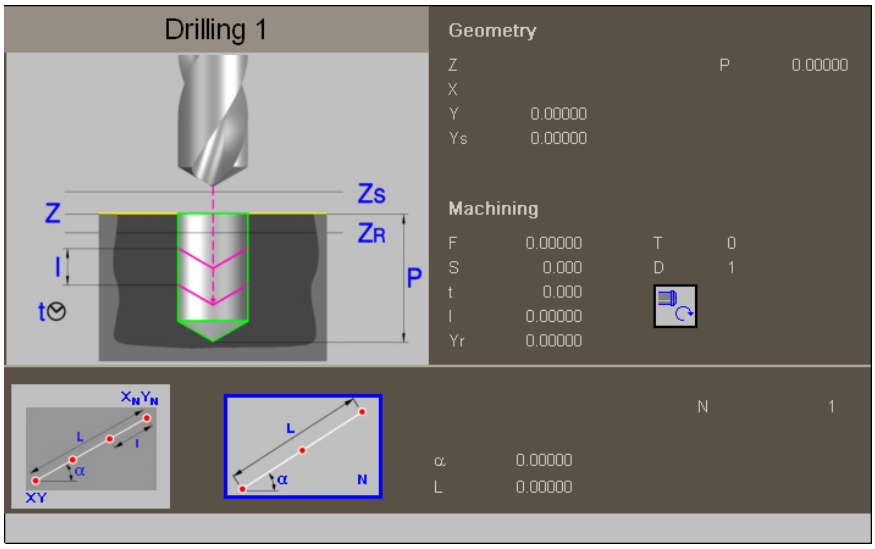
- Center punching, drilling, deep hole drilling, bore milling, tapping, thread milling, reaming, boring, boring with spindle orientation.
- Simple rectangular pocket, rectangular pocket with rounding, circular pocket and pre-emptied circular pocket and 2D profile pocket.
- Rectangular boss and circular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

How to select multiple machining.



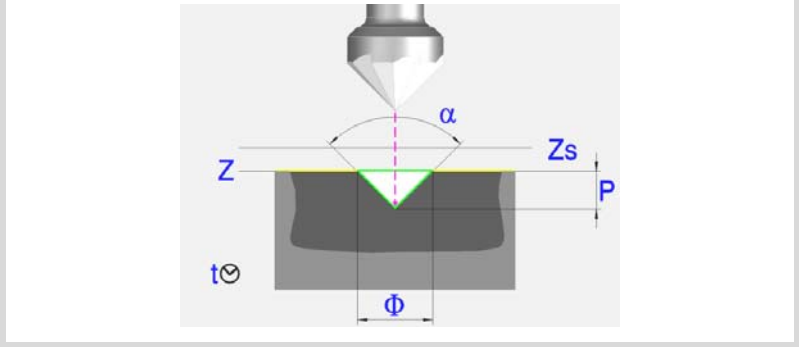
To associate multiple machining to a cycle, first select and define a machining cycle from those allowed. Then, without quitting the editing of the cycle, press the softkey associated with multiple machining and select one of them.

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



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

3.2 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.

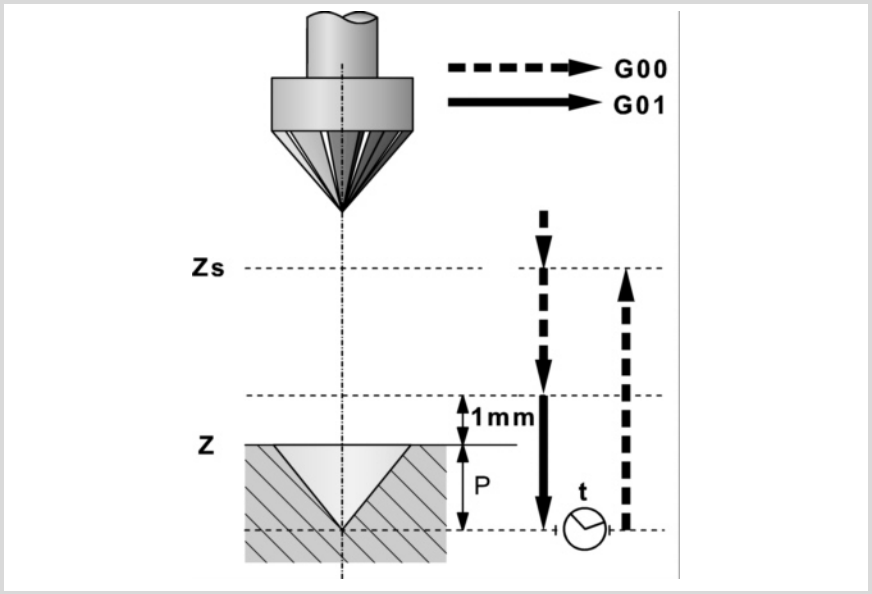
Activating or deactivating the preparation of the next tool.

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

3.2.1 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".

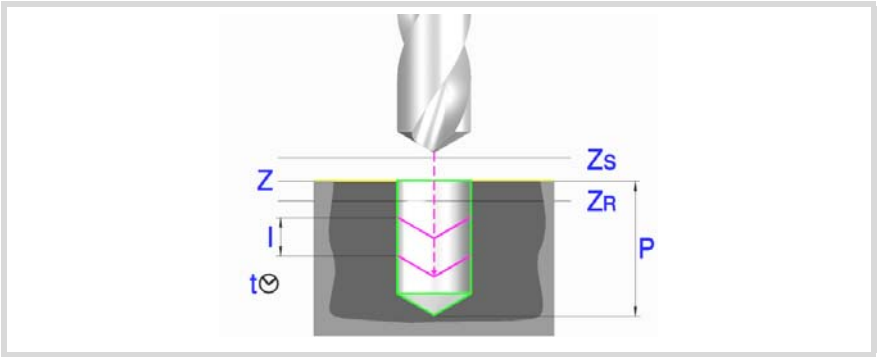
6 Rapid withdrawal (G0) up to the safety plane (Zs).

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.

3.3 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.

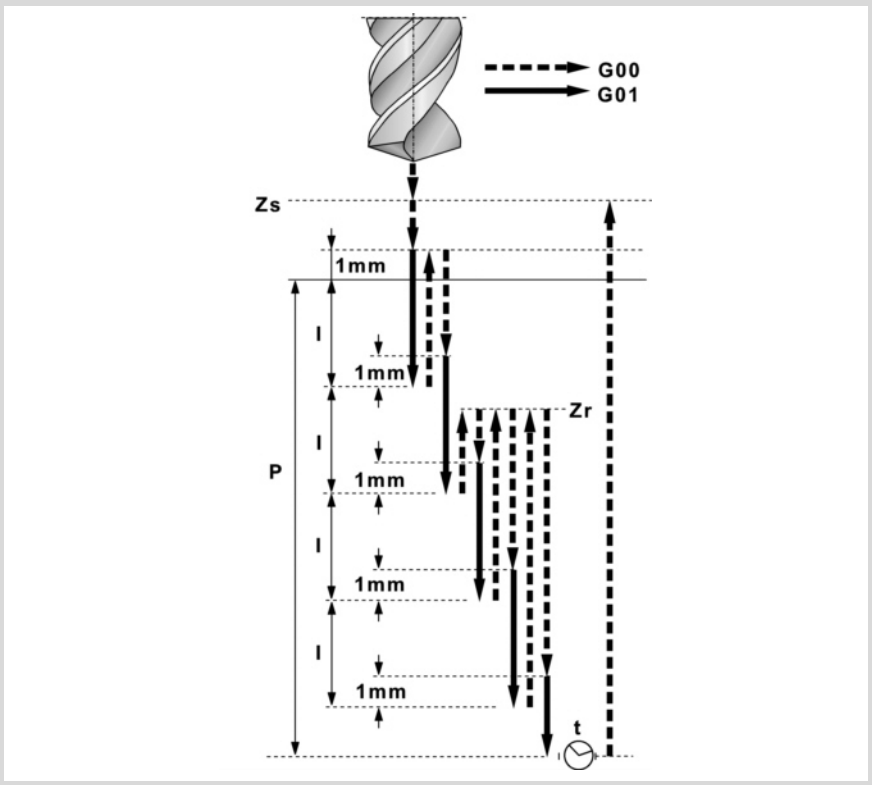
Next tool.

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.

3.3.1 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".

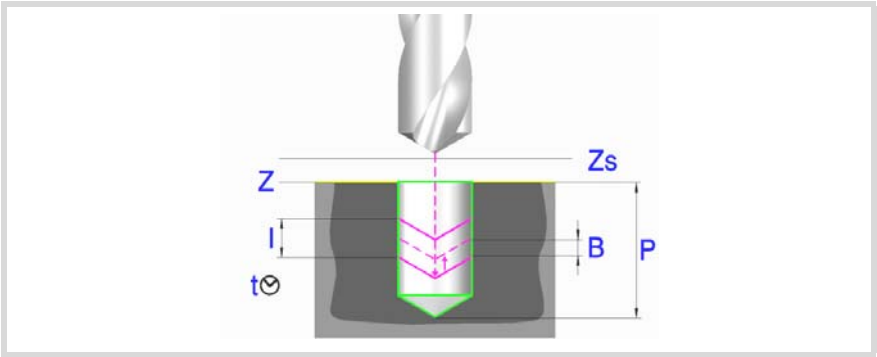
7 Rapid withdrawal (G0) up to the safety plane (Zs).

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, 5, 6, 7.

3.4 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.

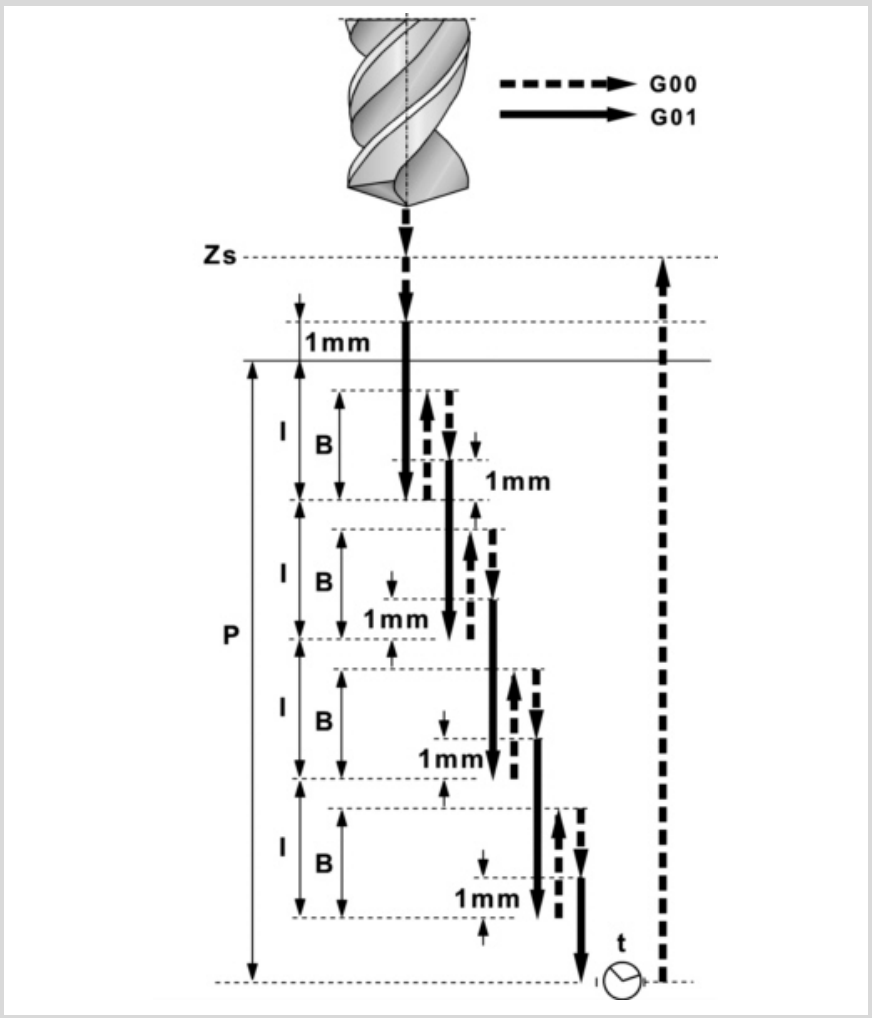
Next tool.


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.

3.4.1 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).

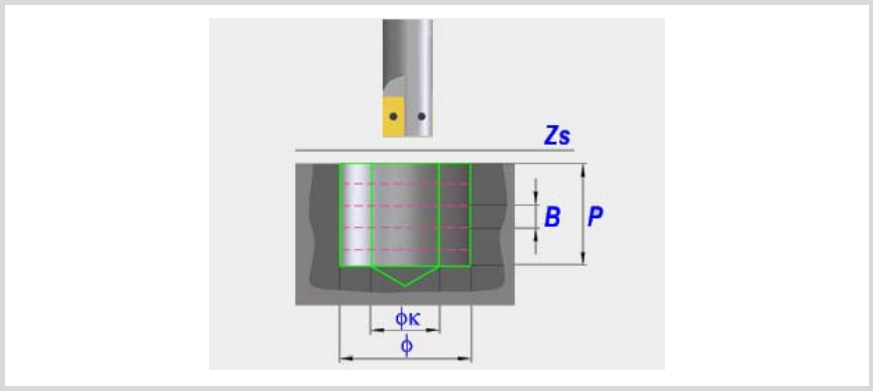
If it has a multiple machining operation associated with it, it executes the following steps as often as necessary:

8 Rapid movement (G0) to the next point.

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

3.5 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.

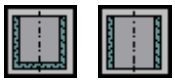


Geometric parameters:

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.

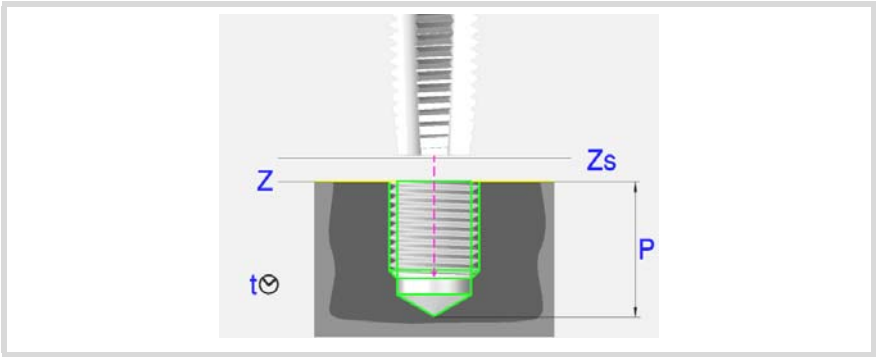
Next tool.

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.

3.5.1 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).

3.6 Tapping.



Geometric parameters:

X, Y	Machining point.
Z	Part surface coordinate.
Zs	Safety plane coordinate.
Dp	Part surface approach distance. The editor will only show this data if the user has configured the editor to allow it. If this option is not enabled, the cycle assumes an approach distance of 1 mm.
P	Total depth.
Kf	Feedrate factor for the exit. Rigid tapping allows a rapid exit from the tap maintaining always the synchronism between the feedrate and the speed. The withdrawal feedrate is multiplied by this factor (Kf) and the speed adapts to the new feedrate.

Type of tapping (icon).


Tapping with a clutch.


Rigid tapping.

Machining parameters:

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

Spindle turning direction (icon).


Clockwise.


Counterclockwise.

Type of feedrate (icon).


In mm/min or (inch/min)


In mm/turn.

Programming of M functions.

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

Next tool.

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

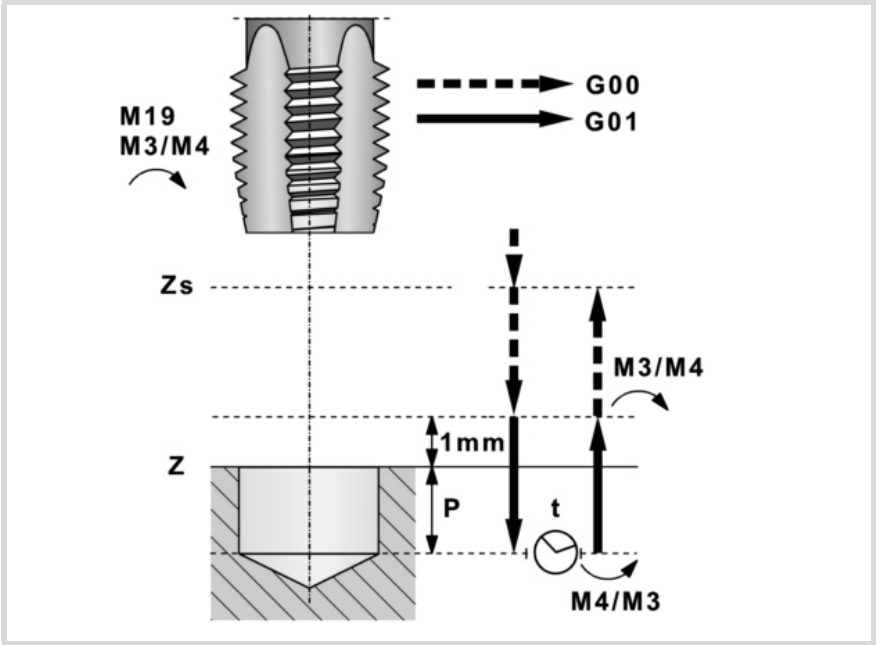
FAGOR AUTOMATION
CNC 8060
CNC 8065
(REF: 1709)

3.6.1 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).

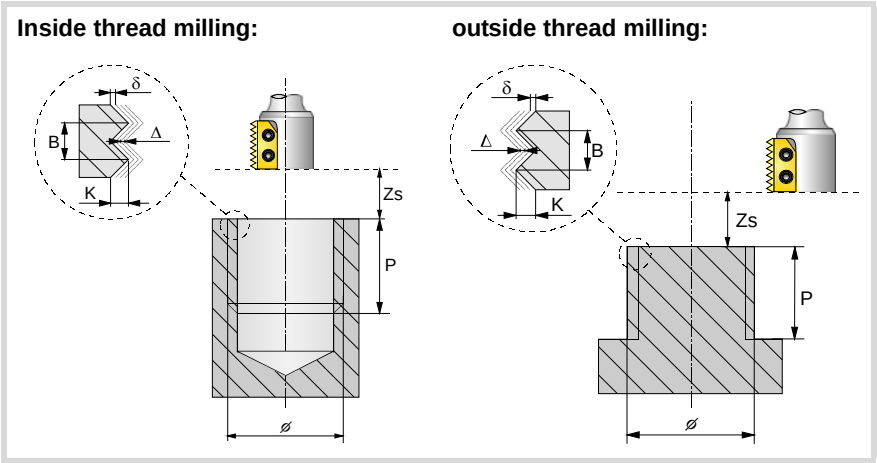
9 Rapid withdrawal (G0) up to the safety plane (Zs).

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.

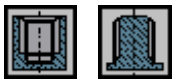
11 Repeats steps 3, 4, 5, 6, 7, 8, 9.

3.7 Thread milling.



Geometric parameters:

Type of tapping.

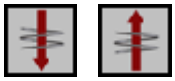


Defines the type of threading to be carried out (inside or outside).

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).

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.

3.7.1 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).

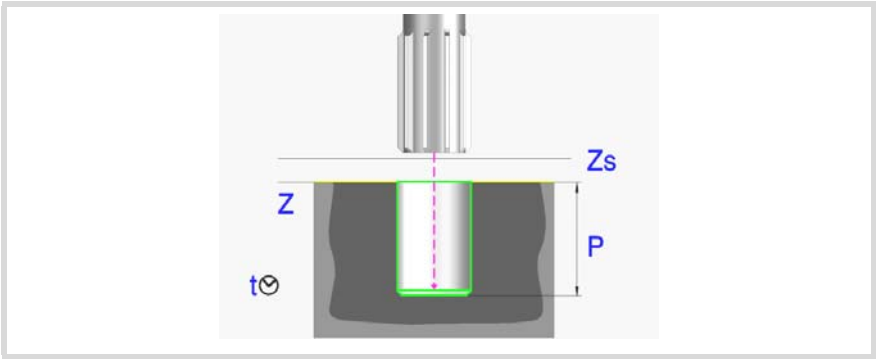
8 Rapid movement to the thread entry coordinate along the longitudinal axis.

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).

3.8 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.

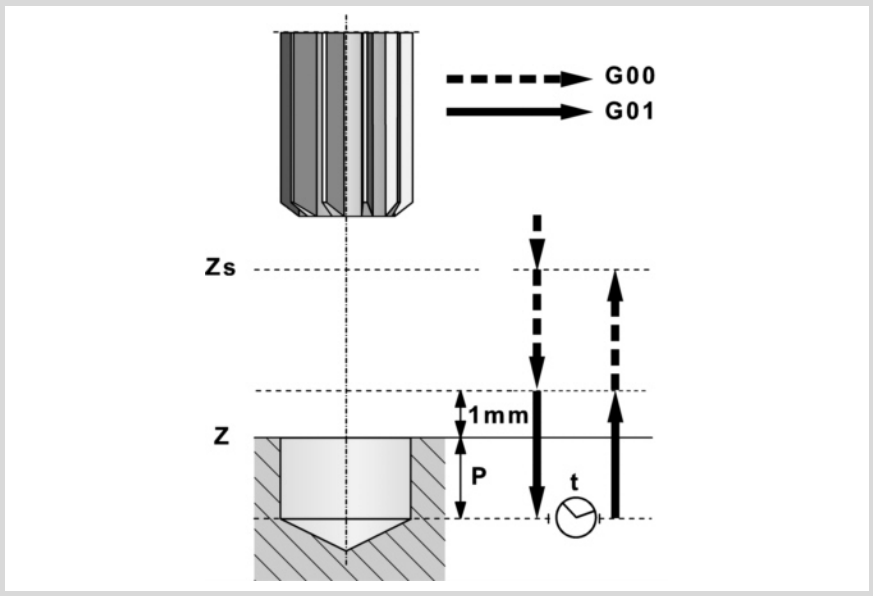
Next tool.

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.

3.8.1 Basic operation.

1 It starts the spindle in the requested direction.

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



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

4 Penetration at feedrate "F".

5 Dwell "t".

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

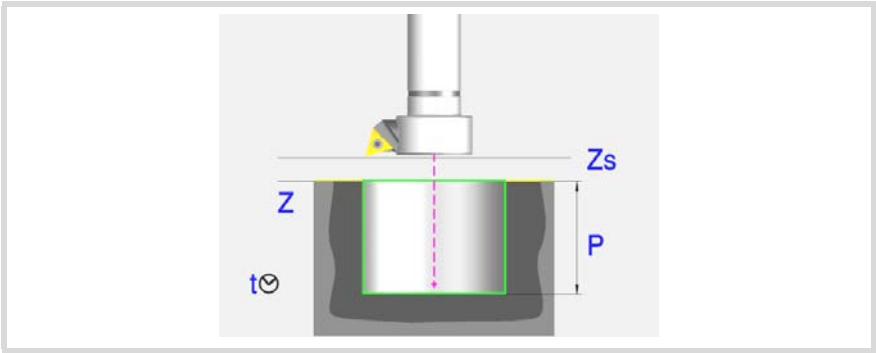
7 Rapid movement (G0) up to the safety plane (Zs).

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.

3.9 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.

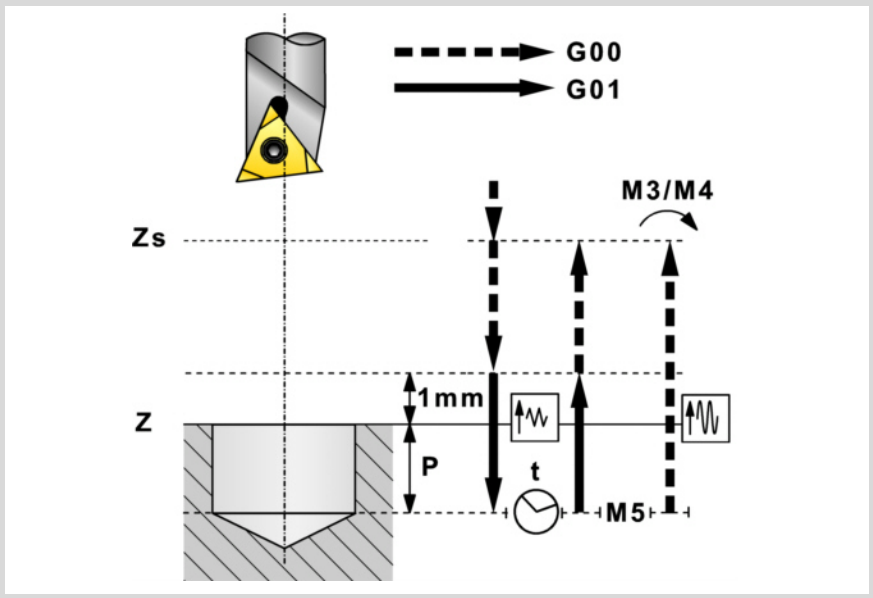
Activating or deactivating the preparation of the next tool.

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

3.9.1 Basic operation.

1 It starts the spindle in the requested direction.

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

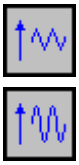


3 Rapid movement (G0) up to the approach plane.

4 Penetration at feedrate "F".

5 Dwell "t".

6 Withdrawal.



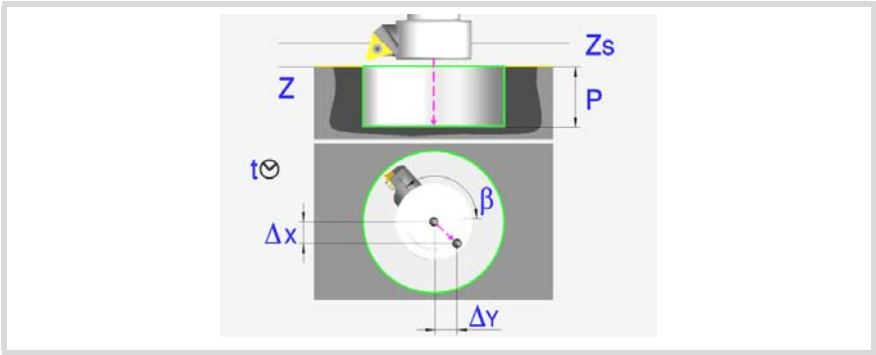
- It withdraws at feedrate "F" to the approach plane (at 1 mm above the surface "Z") and then in rapid (G0) to the safety plane Zs.
- Spindle stop. It withdraws in rapid (G0) to the safety plane Zs and then starts the spindle in the direction it was turning.

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

7 Rapid movement (G0) to the next point.

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

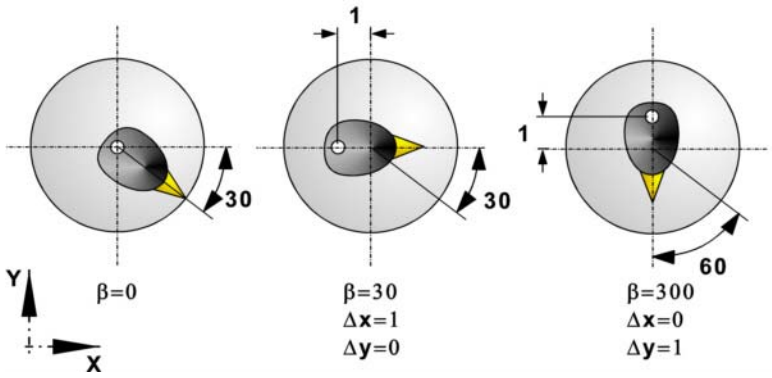
3.10 Boring with spindle orientation.



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.

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



Machining parameters:

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

Spindle turning direction (icon).


Clockwise.


Counterclockwise.

Programming of M functions.

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

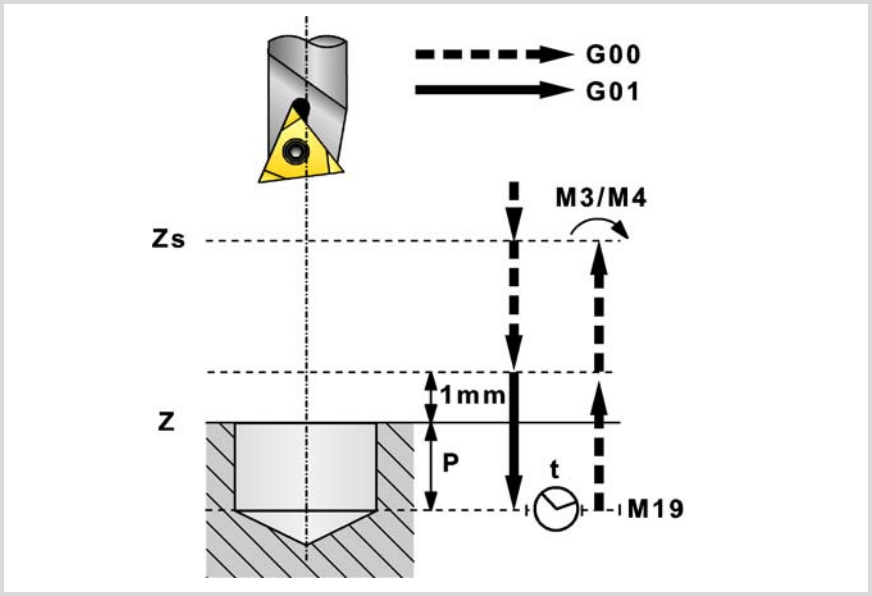
Next tool.

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

3.10.1 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).

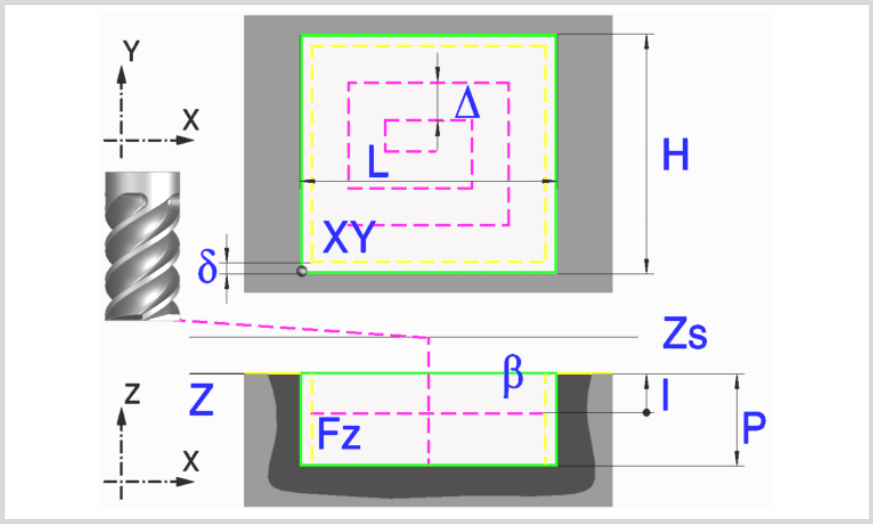
If it has a multiple machining operation associated with it, it executes the following steps as often as necessary:

11 Rapid movement (G0) to the next point.

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

3.11 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.

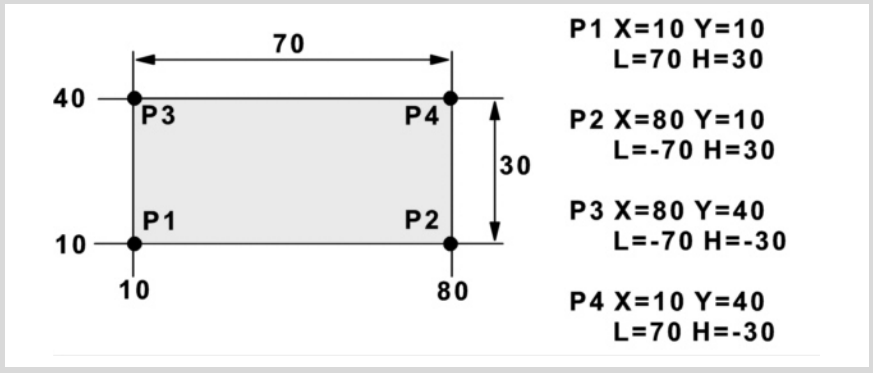
Starting point of the pocket (icon).

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.



Z Part surface coordinate.

Zs Safety plane coordinate.

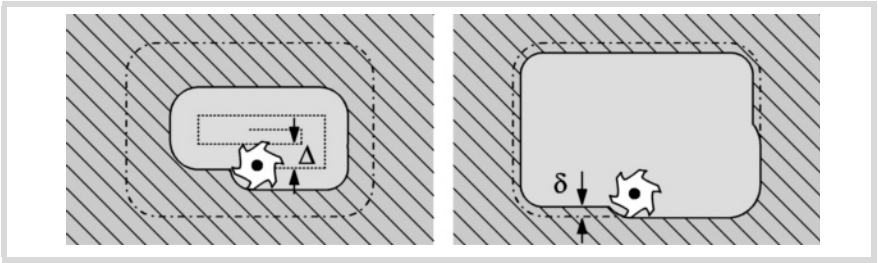
P Total depth.

Machining parameters:

Δ Maximum milling pass or width.

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

δ Finishing stock on the side walls.

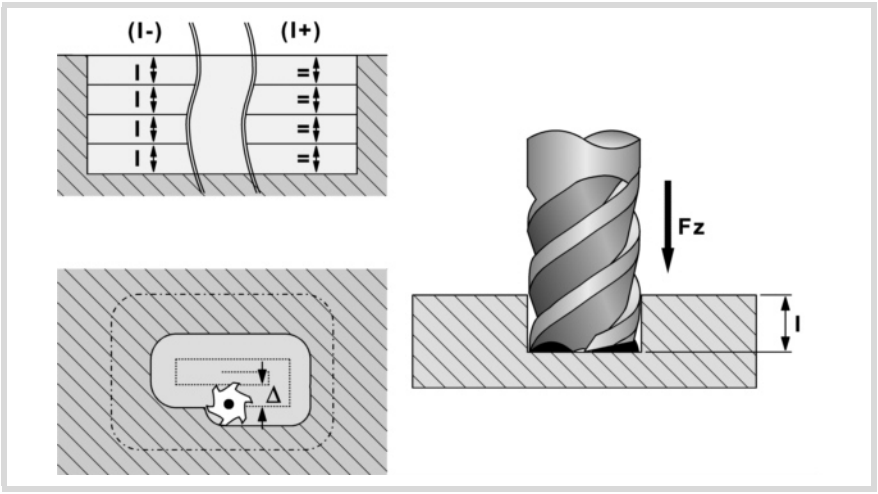


I Penetration step.

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

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.

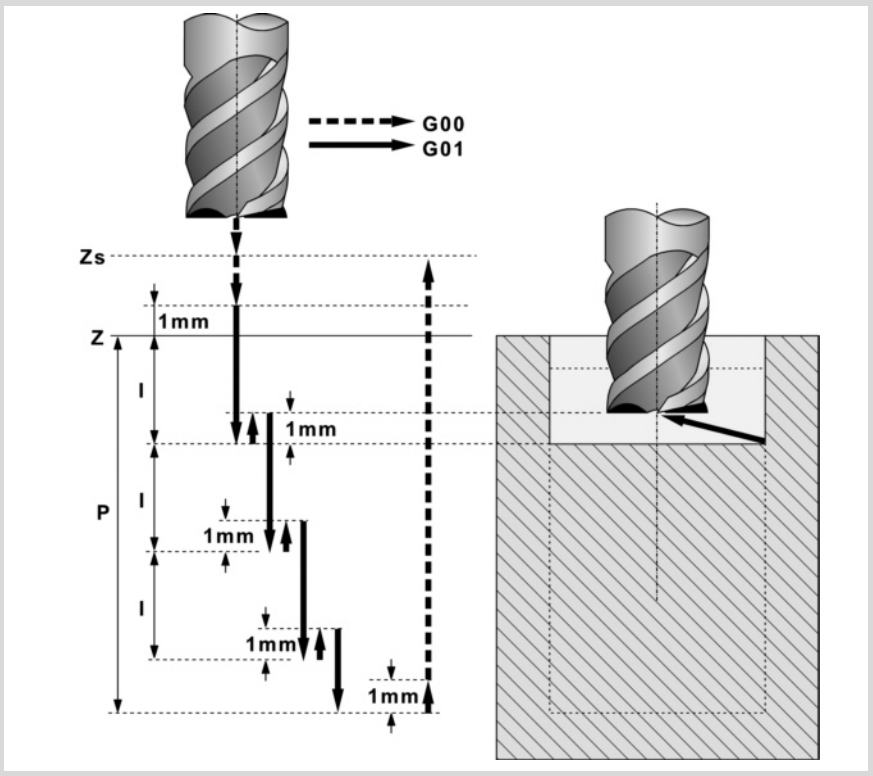
Activating or deactivating the preparation of the next tool.

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

3.11.1 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).

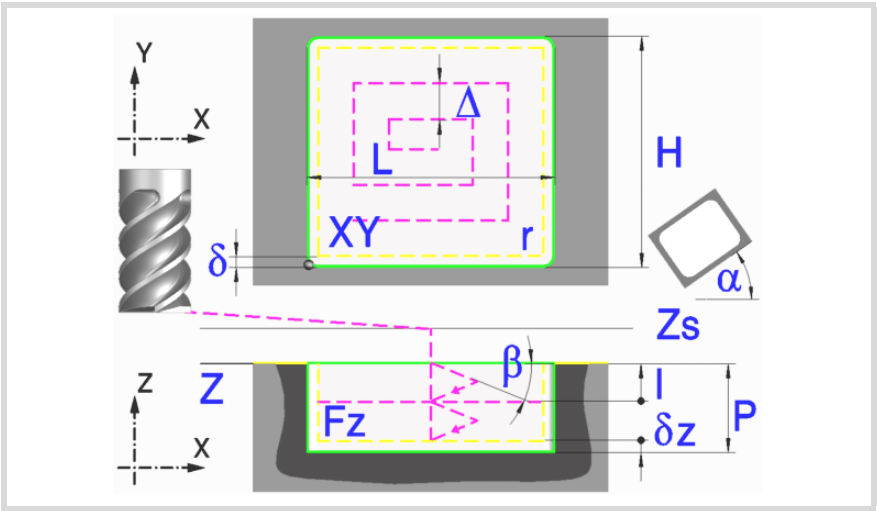
If it has a multiple machining operation associated with it, it executes the following steps as often as necessary:

9 Rapid movement (G0) to the next point.

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

3.12 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.

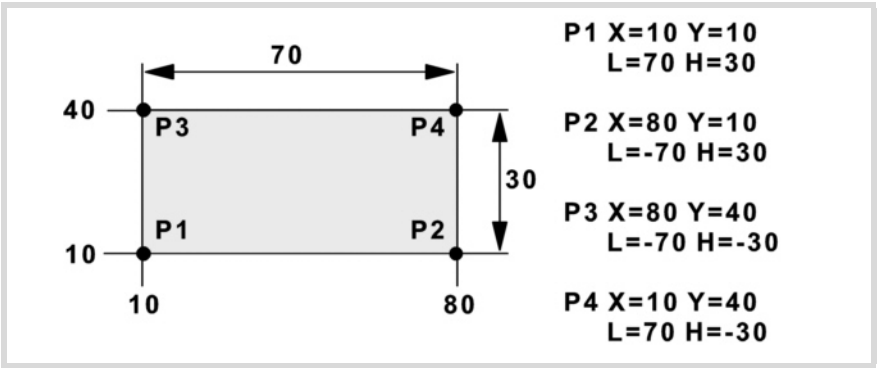
Starting point of the pocket (icon).

Starting point at one corner of the pocket.

Starting point at the center of the pocket.

L, H Pocket dimensions.

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



Z Part surface coordinate.

Zs Safety plane coordinate.

P Total depth.

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

Type of corner (icon).

Square corner with icon.

Rounded corner with icon.

Chamfered corner with icon.

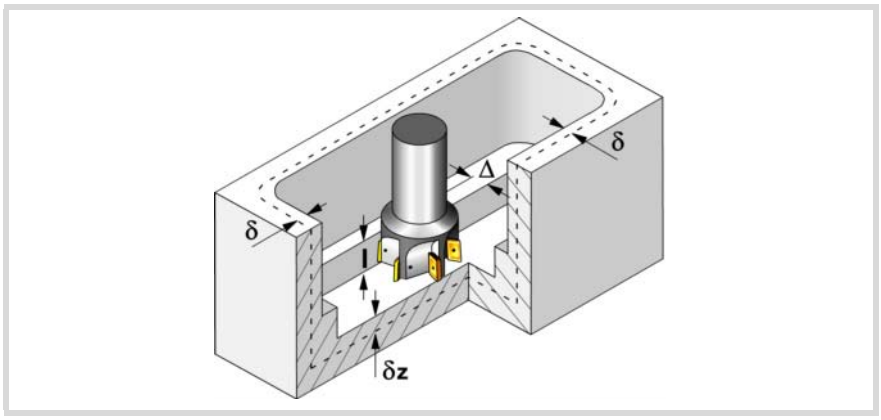
r Rounding radius or chamfer size.

Roughing parameters:

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

δ Finishing stock on the side walls.

δz Finishing stock at the bottom of the pocket.



The roughing operation defining parameters are:

Δ Maximum milling pass or width.

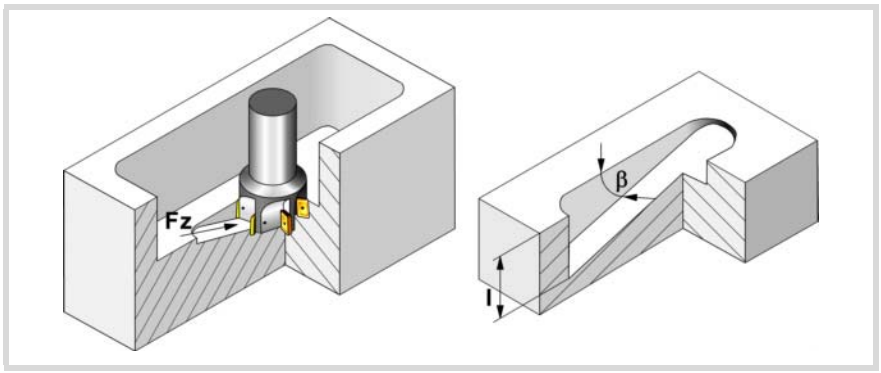
The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

I Penetration step.

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

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 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.

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.

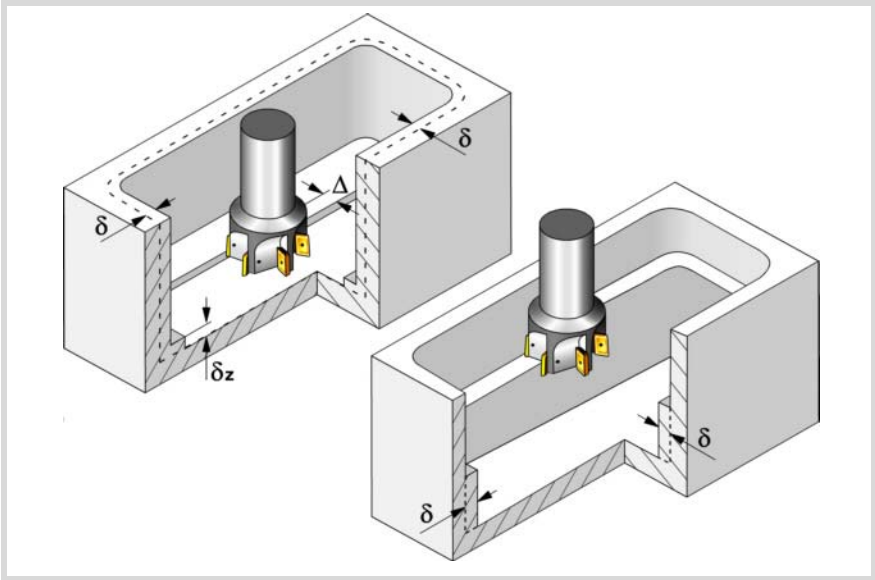
Counterclockwise.

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.

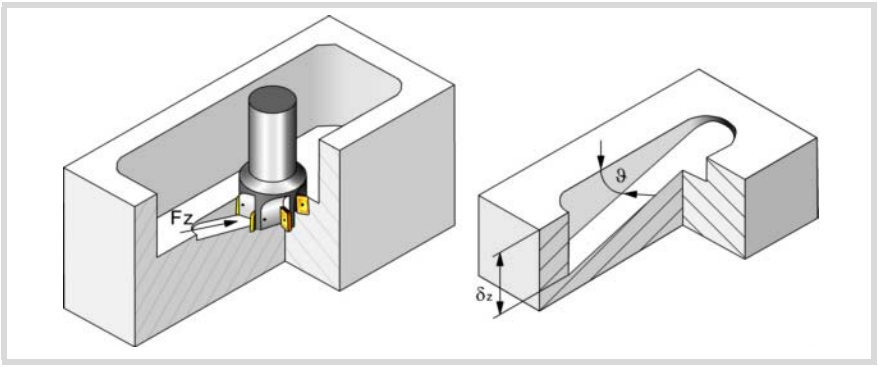
Δ Milling pass or width at the bottom of the pocket.

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.

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.

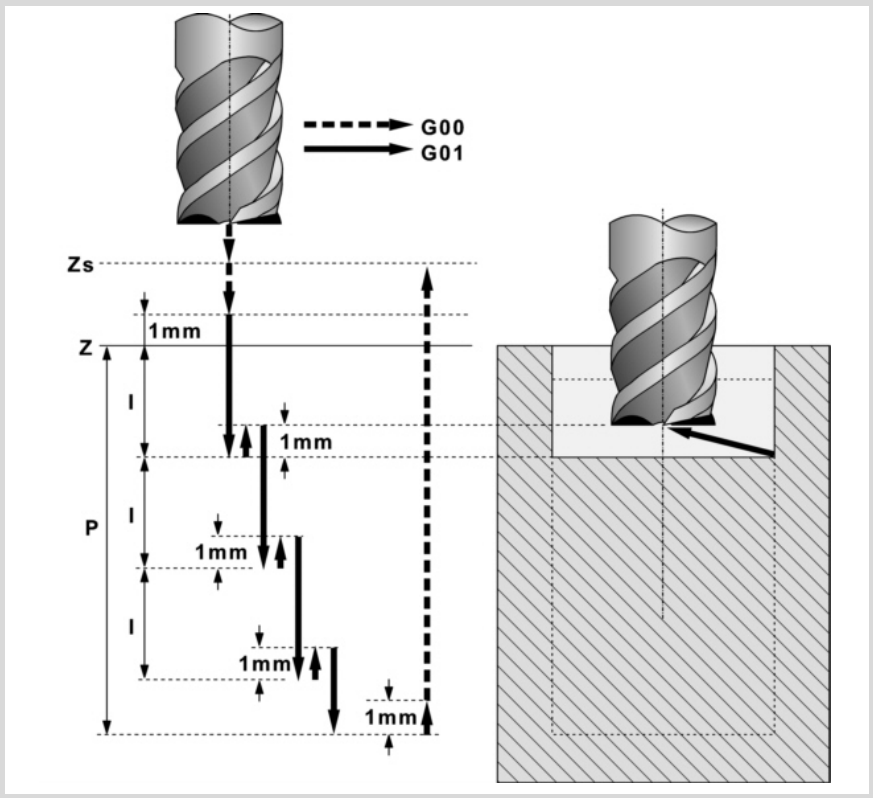
Next tool.


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.

3.12.1 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 "δz".

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

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

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

7 Finishing of the bottom of the pocket.

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

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

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

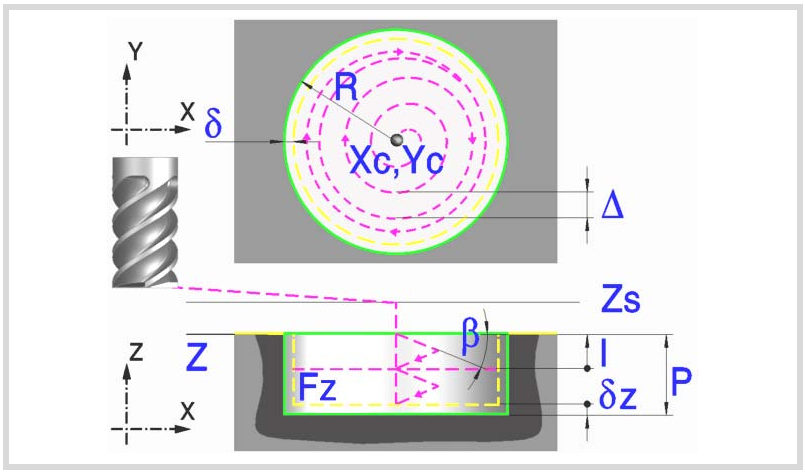
10 Rapid withdrawal (G0) to the center of the pocket in the safety plane (Zs).

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

11 Rapid movement (G0) to the next point.

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

3.13 Circular pocket.



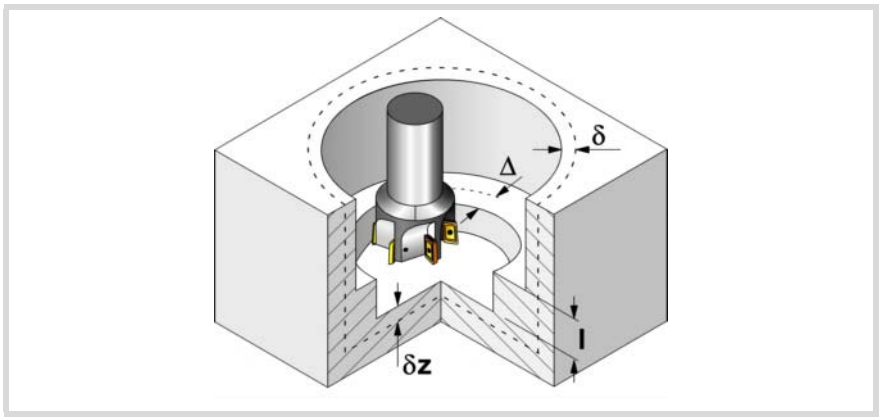
Geometric parameters:

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

Roughing parameters:

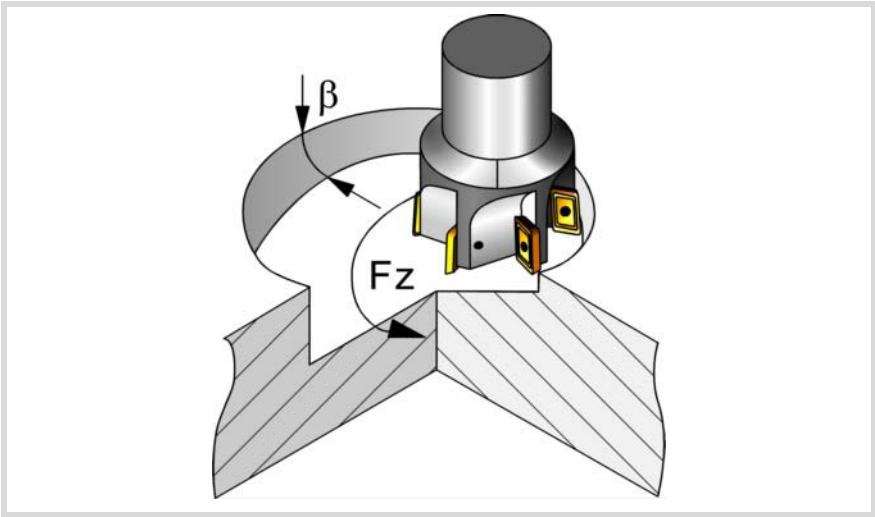
The roughing operation empties the pocket leaving the following finishing stocks. Both stocks are defined as finishing parameters.

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



The roughing operation defining parameters are:

Δ	Maximum milling pass or width. The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.
I	Penetration step. • If programmed with a positive sign (I+), the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed. • If programmed with a negative sign (I-), the pocket is machined with the given pass (step) except the last pass that machines the rest. 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 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.

F Surface milling feedrate.

S Spindle speed.

T Roughing tool.

If programmed T=0, there is no roughing.

D Tool offset.

Spindle turning direction (icon).

 Clockwise.

 Counterclockwise.

Machining direction (icon).

 Clockwise.

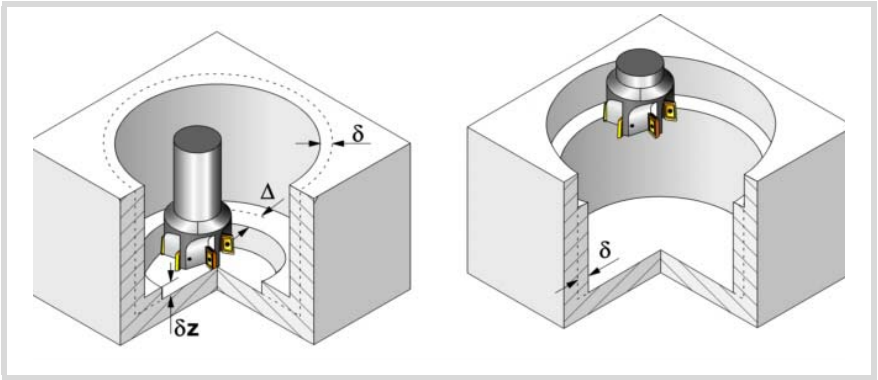
 Counterclockwise.

Programming of M functions.

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

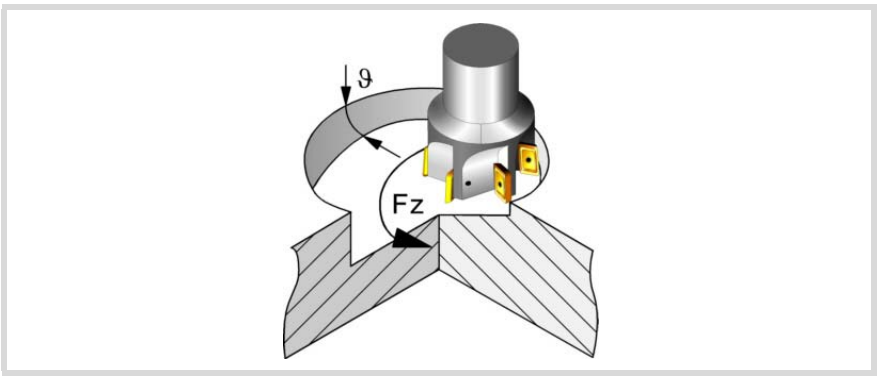
Finishing parameters:

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



The finishing operation defining parameters are:

δ	Finishing stock on the side walls.
δz	Finishing stock at the bottom of the pocket.
Δ	Milling pass or width at the bottom of the pocket. The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.
N	Number of penetration passes (steps) for the side finishing. If the resulting step is greater than the cutting length assigned to the table in the tool table, the step will be limited to that value.
θ	Penetrating angle. The penetration is carried out along a helical path at the feedrate set by roughing parameter "Fz" starting and ending at the center of the pocket. If defined with a value greater than the one assigned to the tool in the tool table, it assumes the table value.



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.

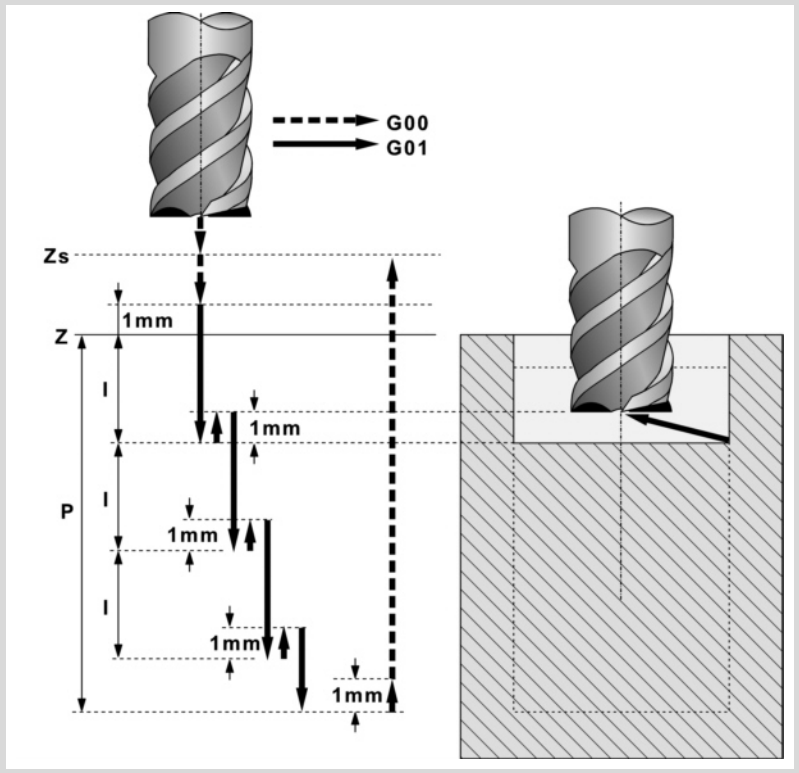
Activating or deactivating the preparation of the next tool.
The editor will only show this option if the user has configured the editor to allow programming the next tool.

3.13.1 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 "δz".

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

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

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

7 Finishing of the bottom of the pocket.

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

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

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

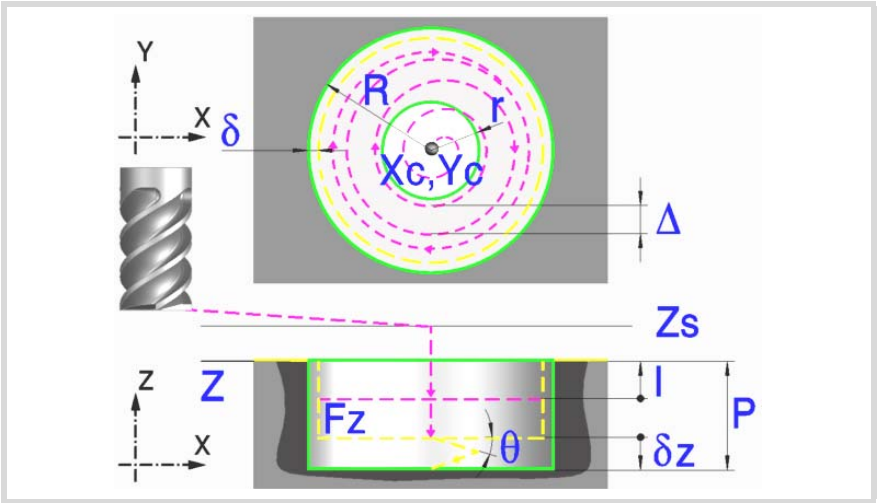
10 Rapid withdrawal (G0) to the center of the pocket in the safety plane (Zs).

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

11 Rapid movement (G0) to the next point.

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

3.14 Pre-emptied circular pocket.



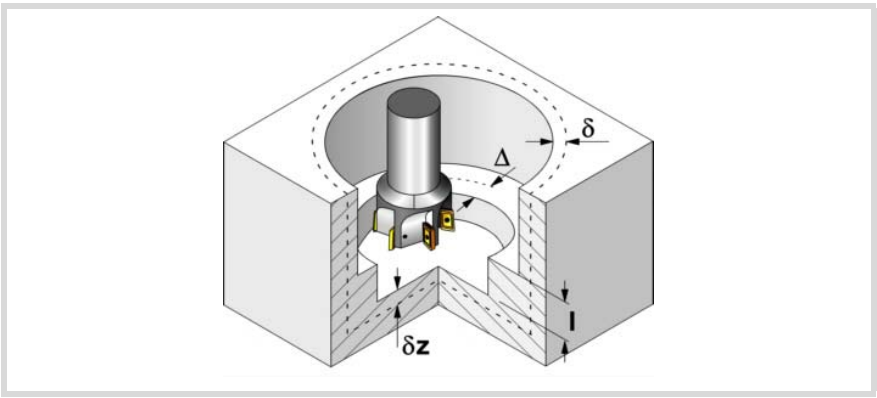
Geometric parameters:

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

Roughing parameters:

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

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



The roughing operation defining parameters are:

Δ	Maximum milling pass or width. The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.
I	Penetration step. • If programmed with a positive sign (I+), the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed. • If programmed with a negative sign (I-), the pocket is machined with the given pass (step) except the last pass that machines the rest. In either case, the cycle limits the step to the cutting length assigned to the tool in the tool table.

F Surface milling feedrate.

S Spindle speed.

T Roughing tool.

If programmed T=0, there is no roughing.

D Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Machining direction (icon).

Clockwise.

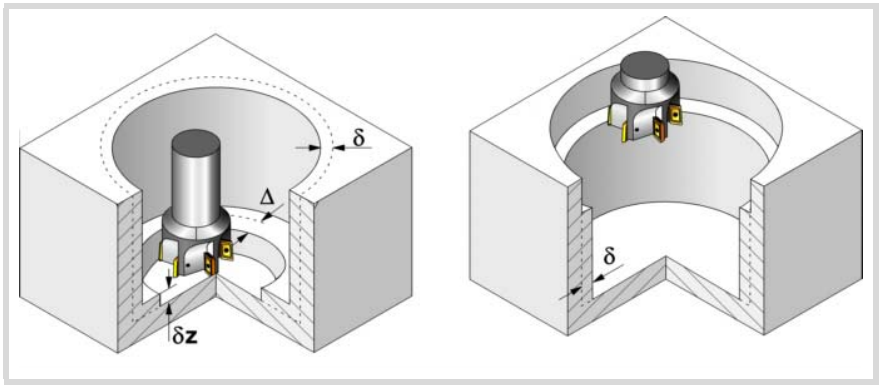
Counterclockwise.

Programming of M functions.

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

Finishing parameters:

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



The finishing operation defining parameters are:

δ Finishing stock on the side walls.

δz Finishing stock at the bottom of the pocket.

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

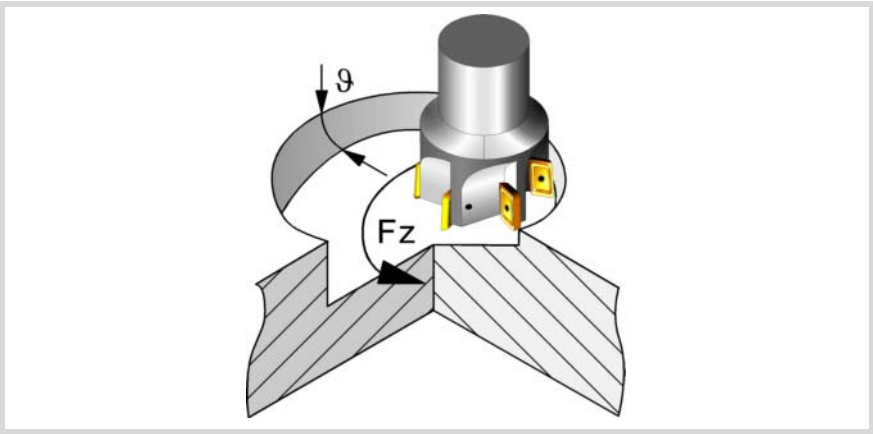
The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

N Number of penetration passes (steps) for the side finishing. If the resulting step is greater than the cutting length assigned to the table in the tool table, the step will be limited to that value.

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.



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.

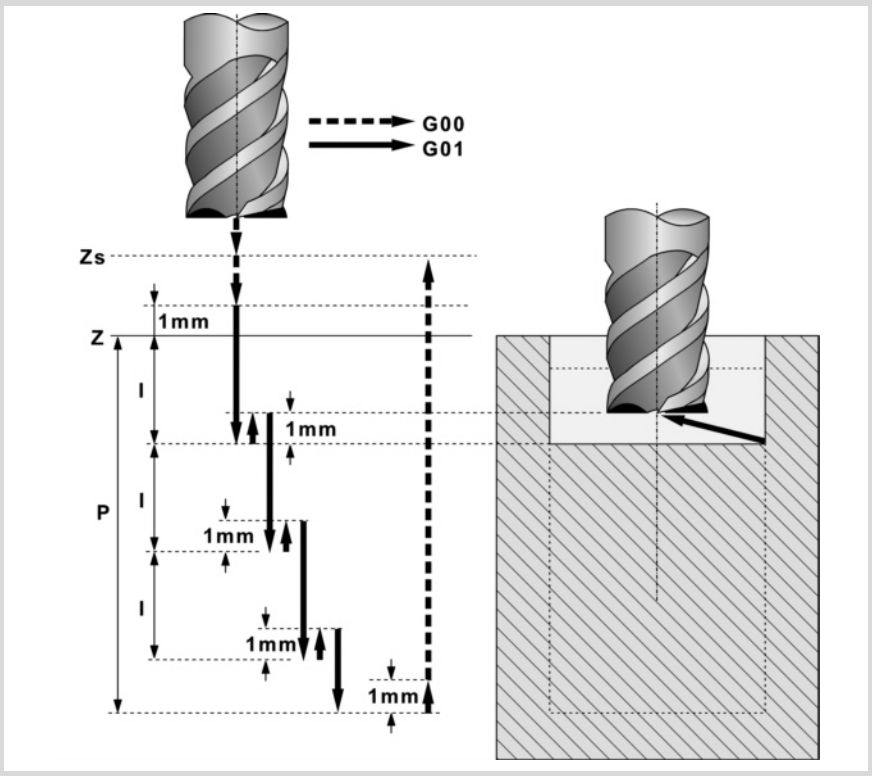

Activating or deactivating the preparation of the next tool.

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

3.14.1 Basic operation.

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

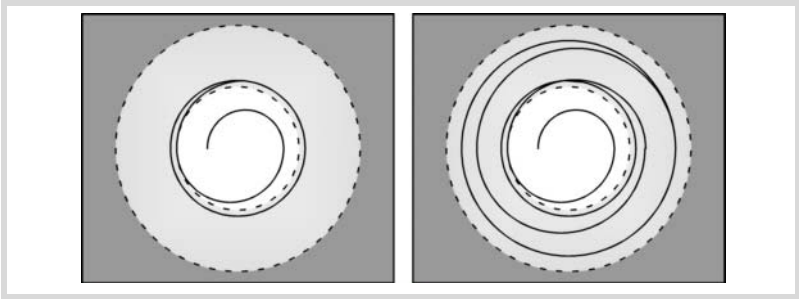
2 Rapid movement (G0) to the center of the pocket and the safety plane (Zs). Depending on the starting plane, it first moves in XY and then in Z or vice versa.



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



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

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

7 Finishing of the bottom of the pocket.

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

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

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

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

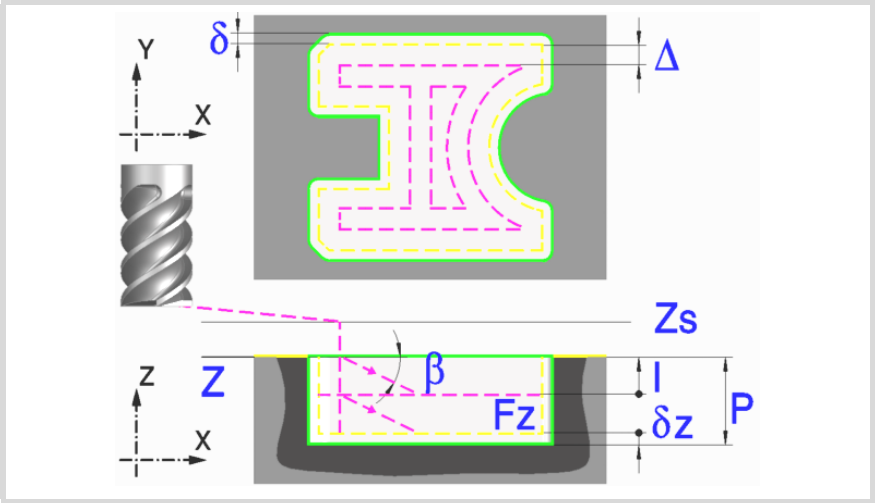
If it has a multiple machining operation associated with it, it executes the following steps as often as necessary:

11 Rapid movement (G0) to the next point.

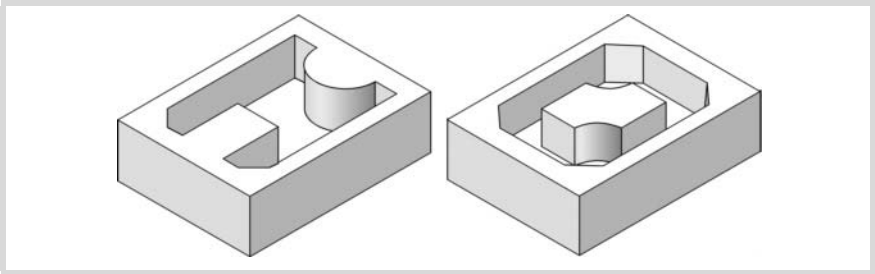
12 Repeats steps 3, 4, 5, 6, 7, 8, 9, 10.

FAGOR AUTOMATION
CNC 8060
CNC 8065
(REF: 1709)

3.15 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.

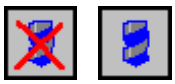
Geometric parameters:

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.

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.

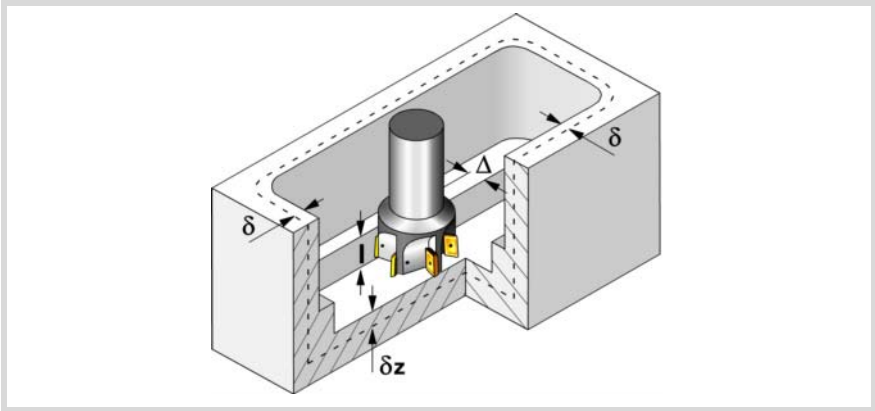
The cycle calculates the drilling point depending on the programmed profile and the roughing tool.

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.

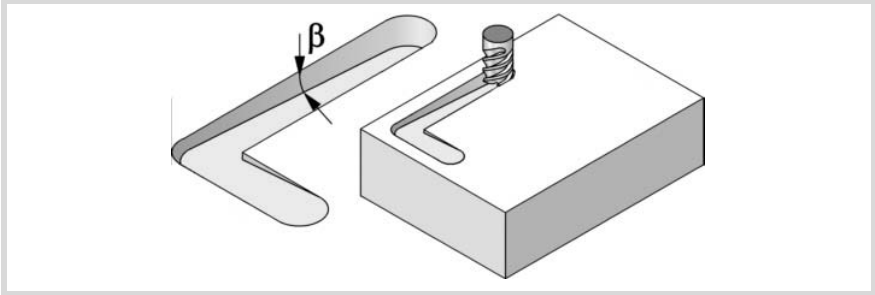
- If programmed with a positive sign (I+), the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If programmed with a negative sign (I-), the pocket is machined with the given pass (step) except the last pass that machines the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

T Roughing tool.

If programmed T=0, there is no roughing.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

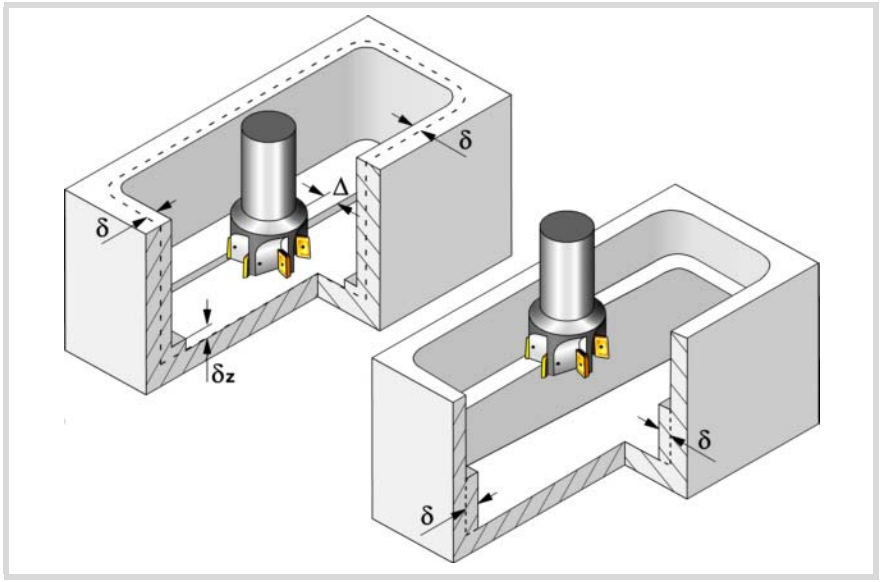
Programming of M functions.

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:

δ Finishing stock on the side walls.

δz Finishing stock at the bottom of the pocket.

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

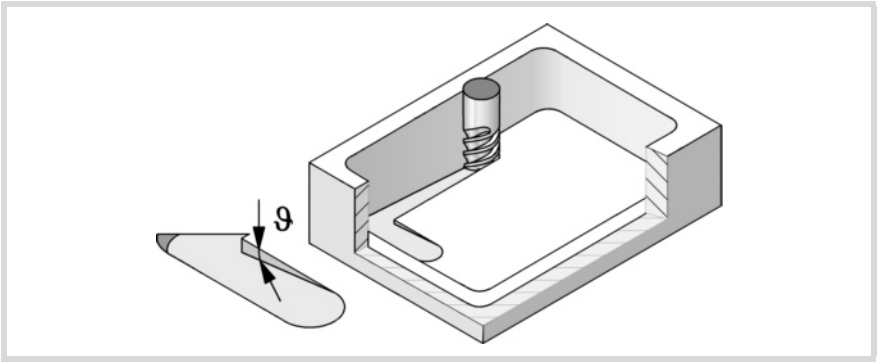
The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

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

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

θ Penetrating angle.

The penetration is carried out at the feedrate set by roughing parameter "Fz" maintaining this angle until reaching the corresponding depth. If defined with a value greater than the one assigned to the tool in the tool table, it assumes the table value.



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.

3.15.1 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 P.2D) and the extension C2D. These files must not be deleted, moved to another location or tampered with in any way. If when executing or simulating the pocket, the CNC cannot find these files, it will generate them.

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

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

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

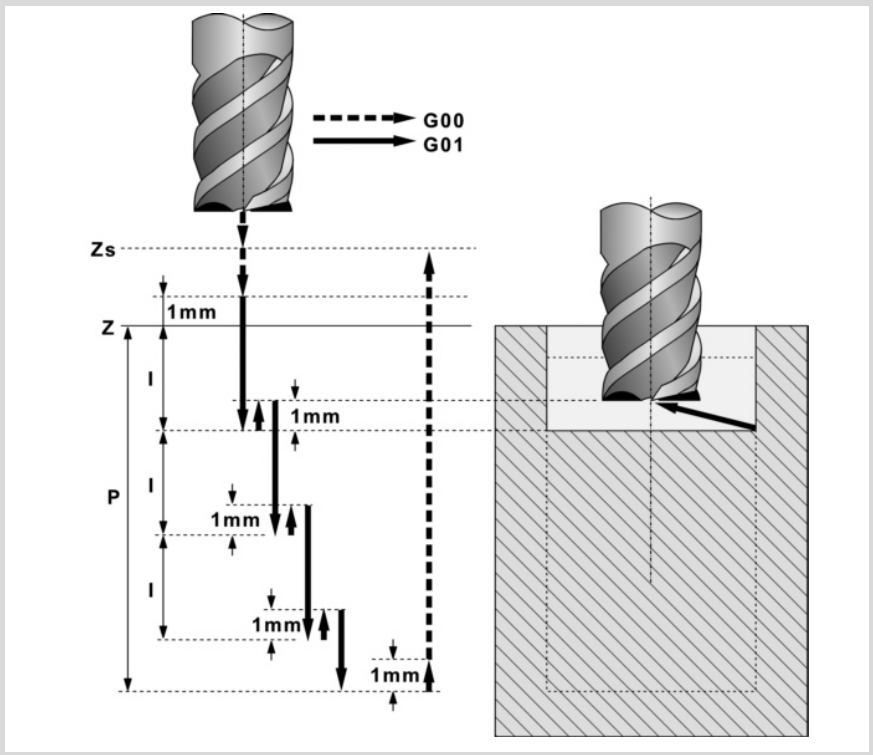
3.15.2 Basic operation.

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.

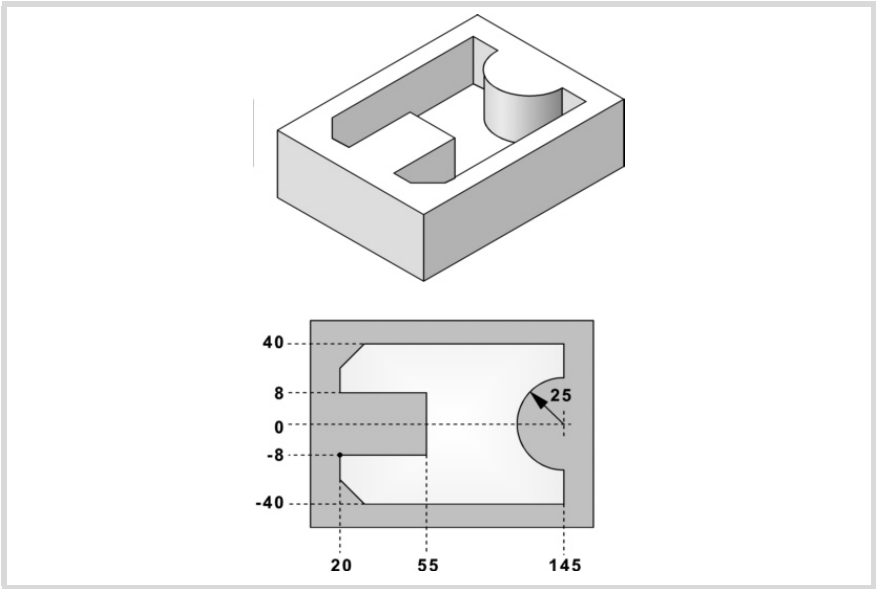
The pocket is machined following paths concentric to the profile, in the same direction as the outside profile was defined. The islands are machined in the opposite direction.

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).

3.15.3 Examples of how to define 2D profiles



Profile P.XY	FAGOR 101	[RECALL]

Configuration:

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

Profile:

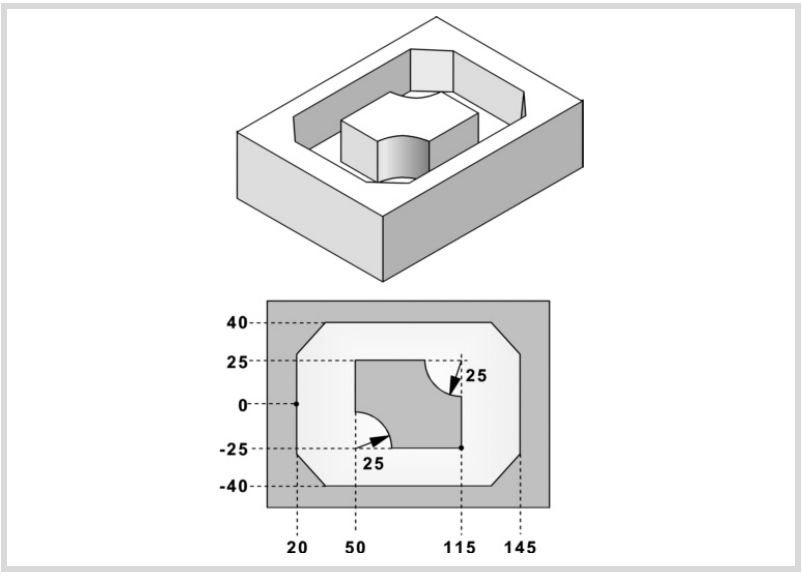
Starting point	X 20	Y -8		Validate
Straight	X 20	Y -40		Validate
Straight	X 145	Y -40		Validate
Straight	X 145	Y -25		Validate
Clockwise arc	Xf 145	Yf 25	R 25	Validate
Straight	X 145	Y 40		Validate
Straight	X 20	Y 40		Validate
Straight	X 20	Y 8		Validate
Straight	X 55	Y 8		Validate
Straight	X 55	Y -8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



Profile P.XY	FAGOR 102	[RECALL]

Configuration:

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

Profile (outside profile):

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

Corners

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

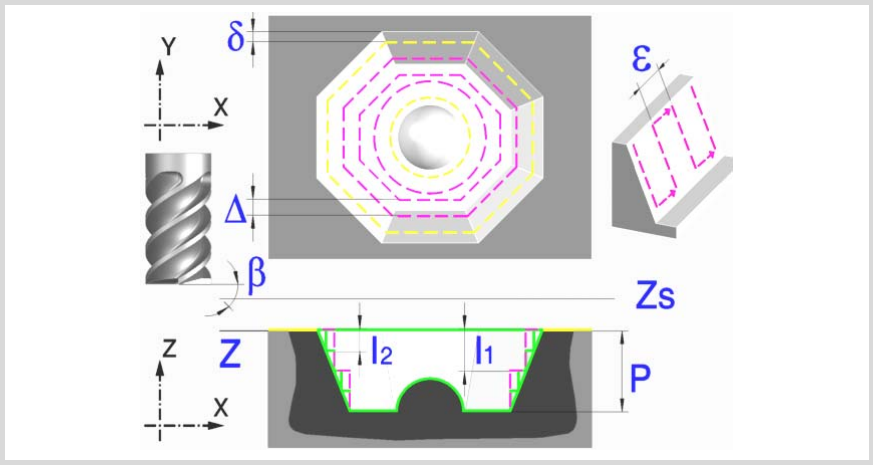
New profile (island):

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

End:

Save profile

3.16 3D profile pocket with islands.

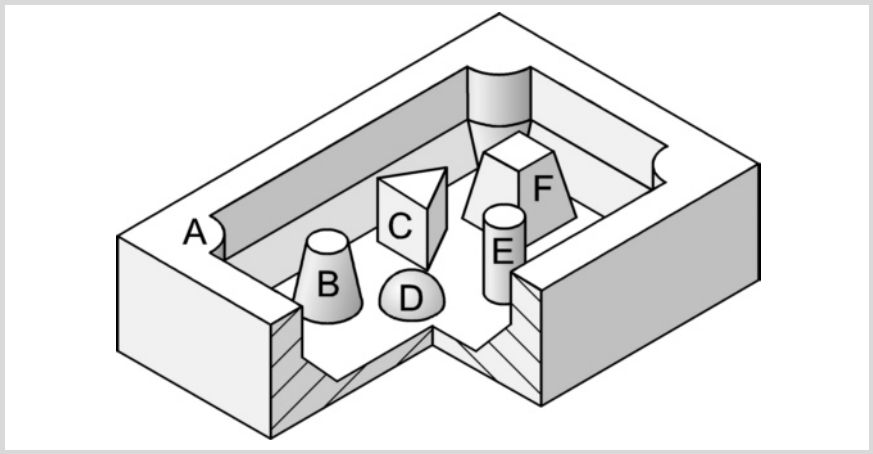


A pocket consists of an outside contour and a number of inside contours called islands.

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.

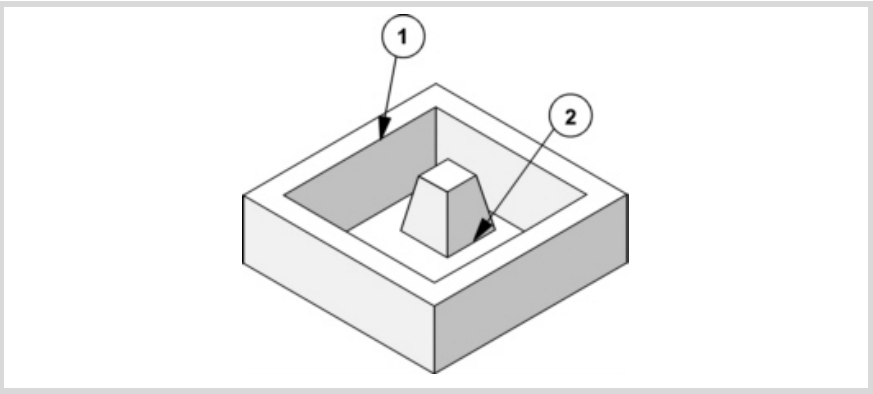
pocket.P3D	Pocket composition.
profile.PXY	Plane profile.
profile.PXZ	Depth profile.

P.3D Name of the 3D pocket.

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

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

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



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

Name of the depth profiles.

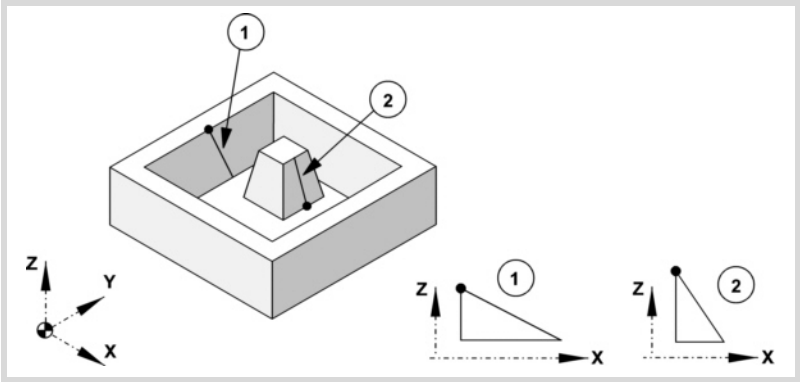
They corresponds to the first 4 contours defined in the surface profile, the number indicates the order.

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

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

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

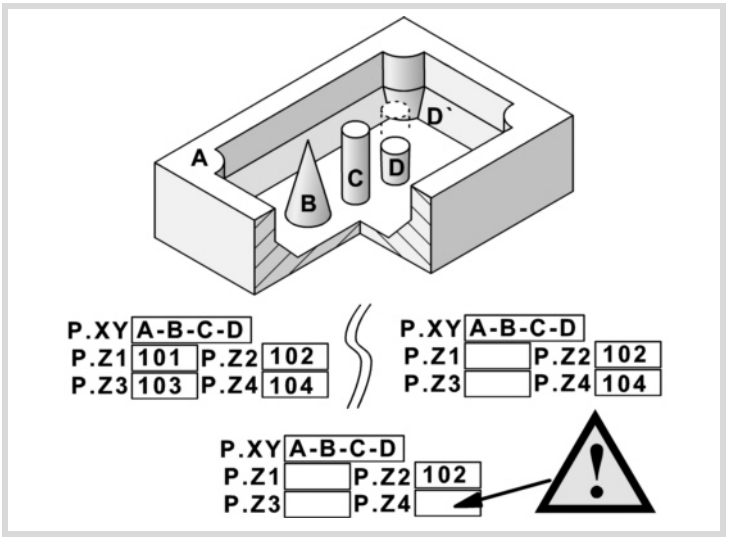
For the islands, one for the base (2).



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

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

The figure shows three programming examples.



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

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

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

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

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

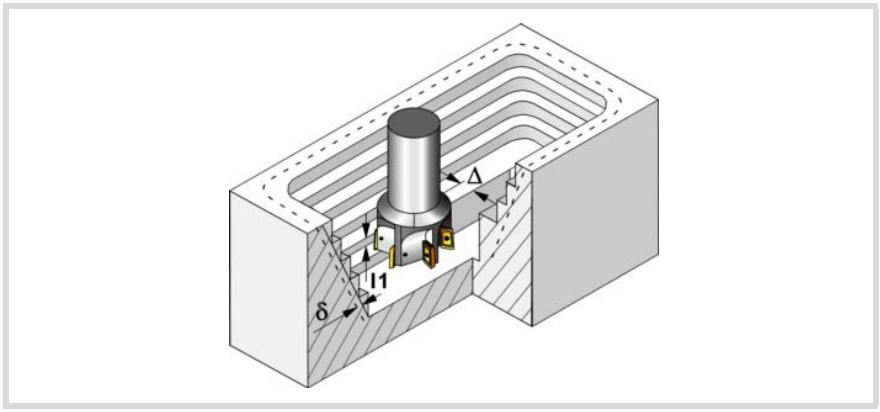
Z Part surface coordinate.

Zs Safety plane coordinate.

P Total depth.

Roughing parameters:

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



The roughing operation defining parameters are:

Δ Maximum milling pass or width.

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

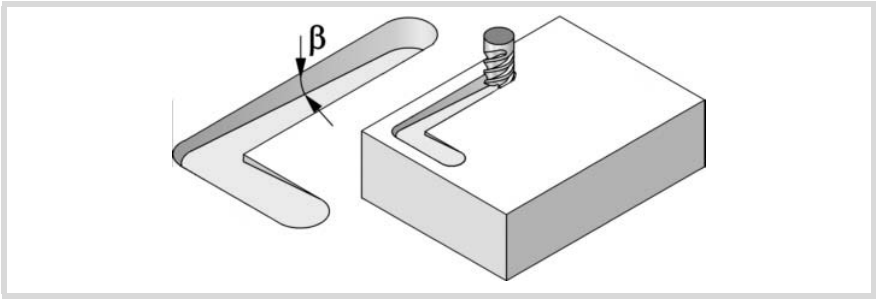
I1 Penetration step.

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

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.

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

D Tool offset.

Spindle turning direction (icon).


Clockwise.

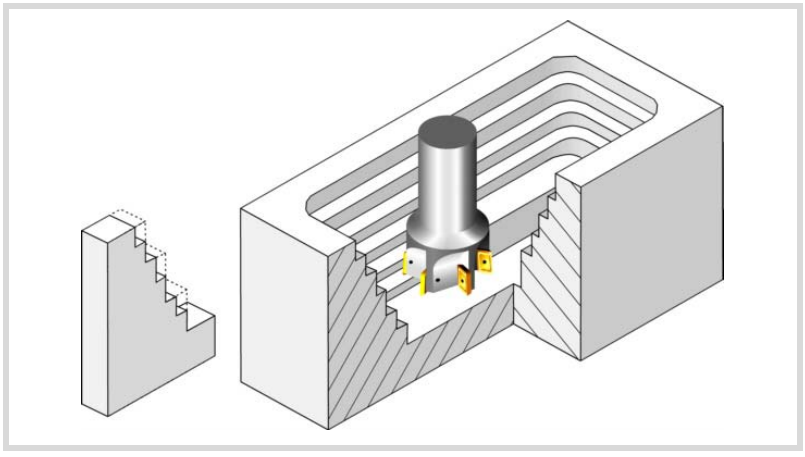

Counterclockwise.

Programming of M functions.


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

Pre-finishing parameters:

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



The pre-finishing operation defining parameters are:

I2 Penetration step.

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

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

F Milling feedrate.

S Spindle speed.

T Pre-finishing tool.

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

D Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

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

Finishing parameters:

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

δ Finishing stock on the side walls.

ε Milling pass or width for the side walls.

Machining direction for the side walls (icon).

Always downwards.

Always upwards.

In zig-zag.

F Milling feedrate.

S Spindle speed.

T Finishing tool.

If programmed T=0, there is no finishing.

D Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

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

Next tool.

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

3.16.1 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.

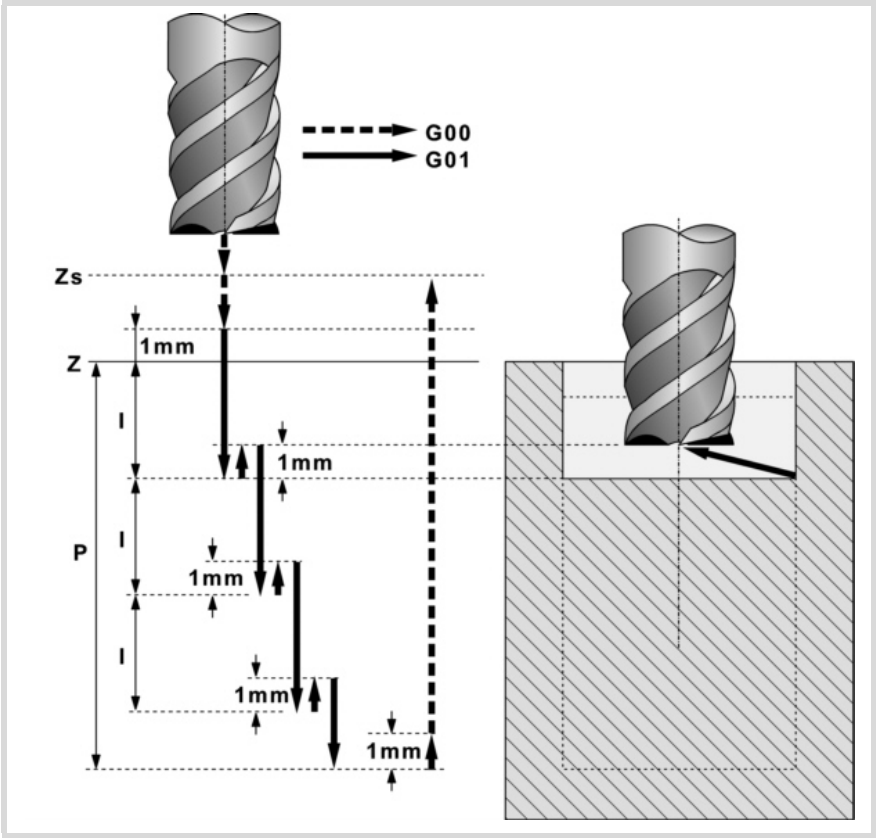
The executable file is also updated after a software update and when executing or simulating a pocket,

3.16.2 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.

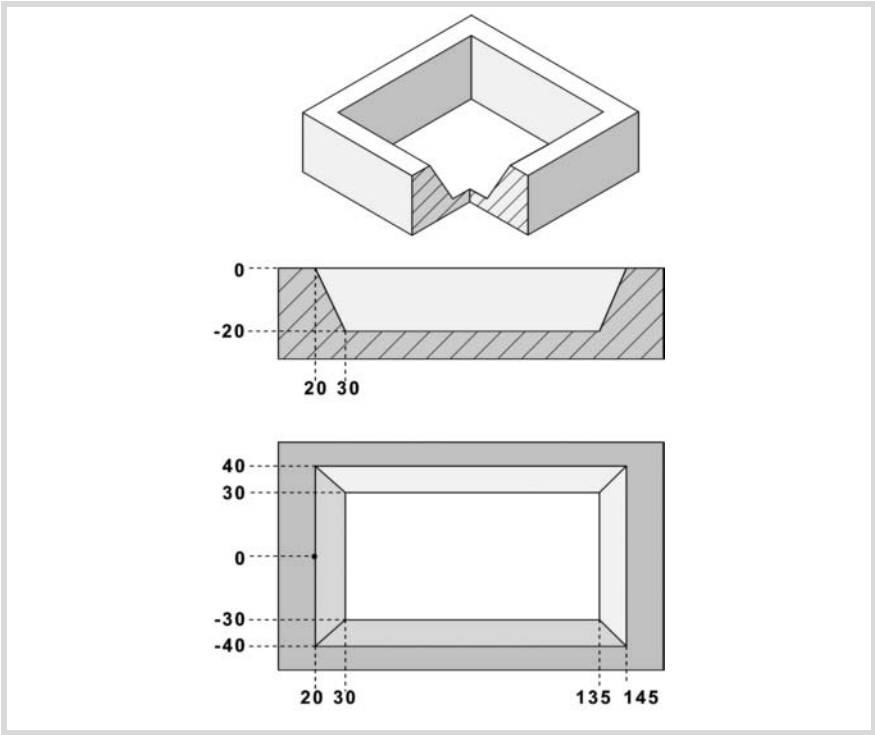
8 Rapid withdrawal (G0) up to the approach plane.

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 "ε" and direction indicated by the icon.

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

3.16.3 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	Validate

Profile (outside profile):

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

End:

Save profile

Profile P.Z1	FAGOR 211	Recall

Configuration:

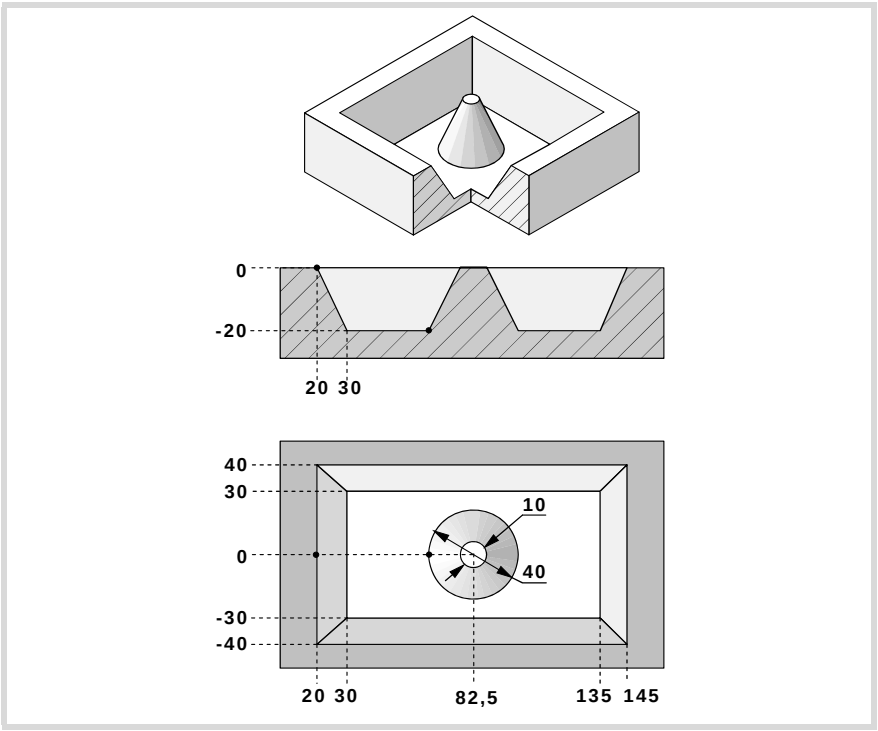
Abscissa axis: X	Ordinate axis: Z
Autozoom: Yes	Validate

Profile (depth profile):

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

End:

Save profile



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

Configuration:

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

Profile (outside profile):

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

New profile (island):

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

End:

Save profile

Profile P.Z1	FAGOR 221	[RECALL]

Configuration:

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

Profile (outside depth profile):

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

End:

Save profile

Profile P.Z2	FAGOR 222	Recall

Configuration:

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

Profile (island depth profile):

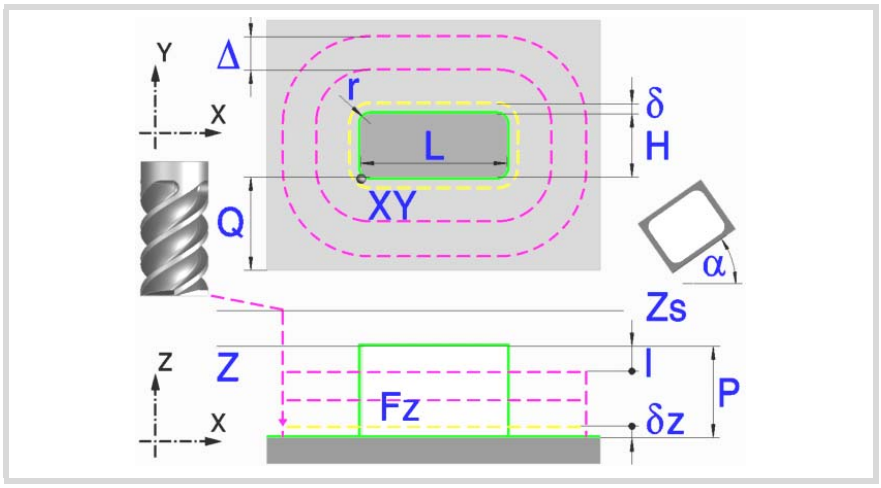
Starting point	X 62,5	Z-20	Validate
Straight	X 77,5	Z0	Validate

End:

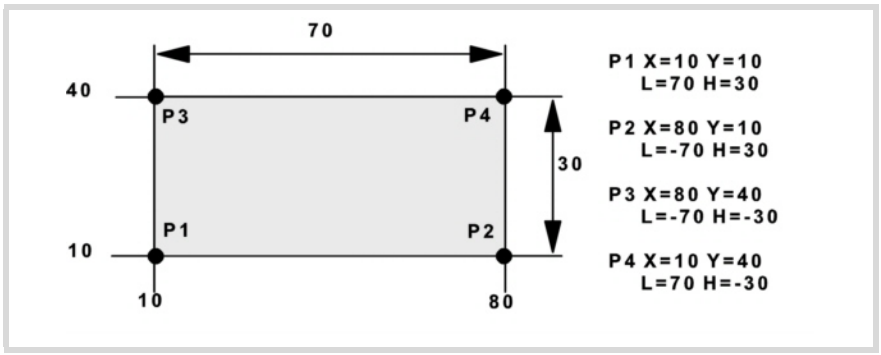
Save profile

3.17 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:



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

Type of corner (icon).

Square corner with icon.

Rounded corner with icon.

Chamfered corner with icon.

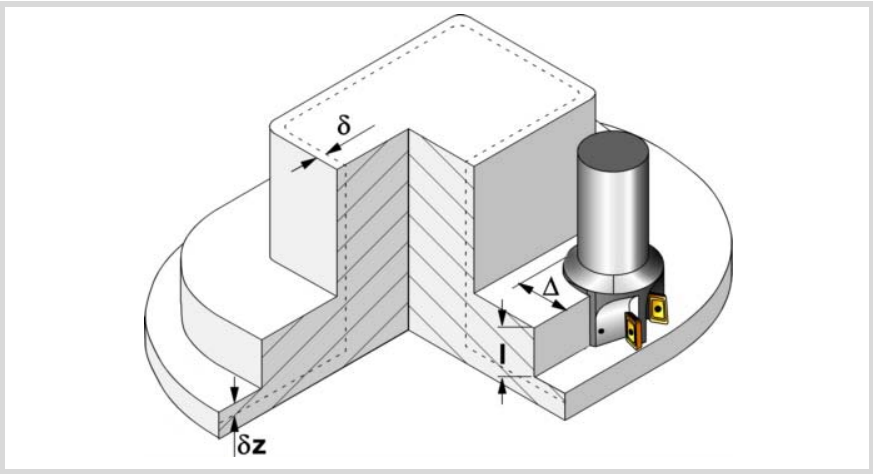
r	Rounding radius or chamfer size.

Roughing parameters:

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

δ Finishing stock on the side walls.

δz Finishing stock at the base of the boss.



The roughing operation defining parameters are:

Δ Maximum milling pass or width.

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

I Penetration step.

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

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.

If programmed T=0, there is no roughing.

D Tool offset.

Spindle turning direction (icon).


Clockwise.


Counterclockwise.

Machining direction (icon).


Clockwise.

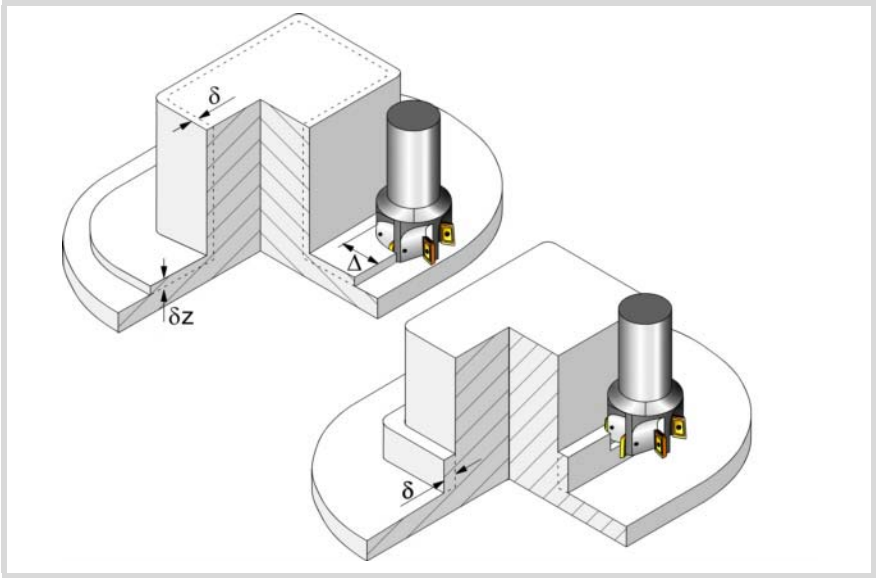

Counterclockwise.

Programming of M functions.

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

Finishing parameters:

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



The finishing operation defining parameters are:

δ	Finishing stock on the side walls.
δz	Finishing stock at the base of the boss.
N	Number of penetration passes (steps) for the side finishing. If the resulting step is greater than the cutting length assigned to the table in the tool table, the step will be limited to that value.
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.

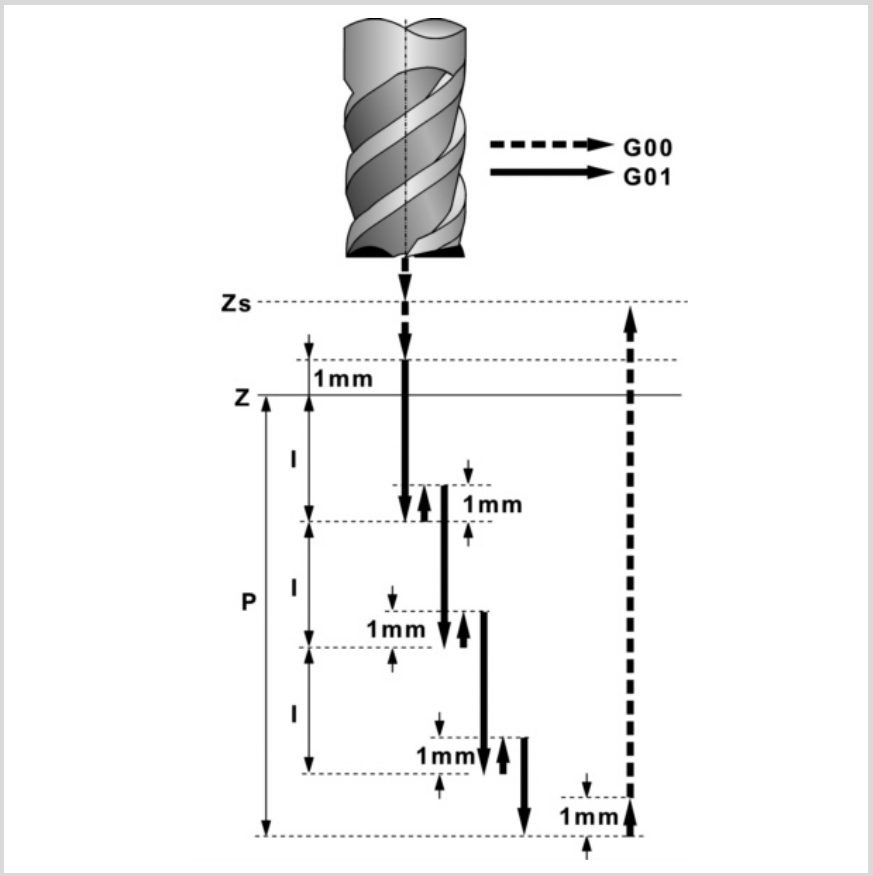
Activating or deactivating the preparation of the next tool.
The editor will only show this option if the user has configured the editor to allow programming the next tool.

3.17.1 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 "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.

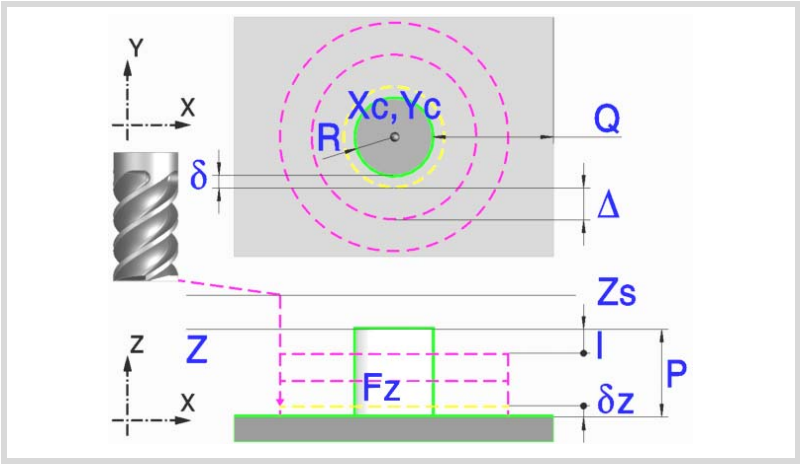
10 Rapid withdrawal (G0) up to the safety plane (Zs).

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.

3.18 Circular boss.



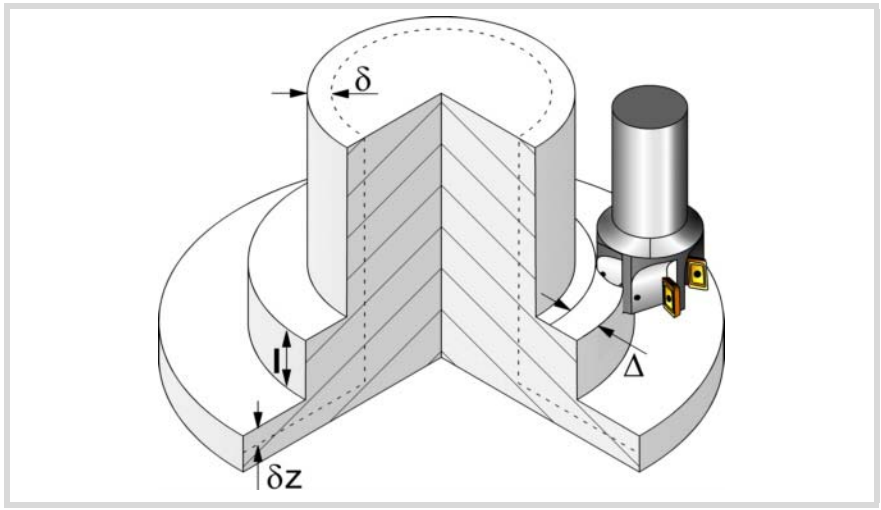
Geometric parameters:

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.

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.

Fz 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.

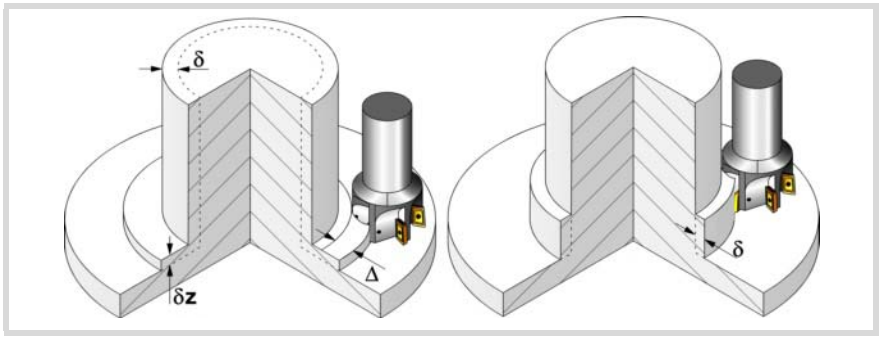
Counterclockwise.

Programming of M functions.

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

Finishing parameters:

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



The finishing operation defining parameters are:

δ Finishing stock on the side walls.

δz Finishing stock at the base of the boss.

N Number of penetration passes (steps) for the side finishing. If the resulting step is greater than the cutting length assigned to the table in the tool table, the step will be limited to that value.

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.

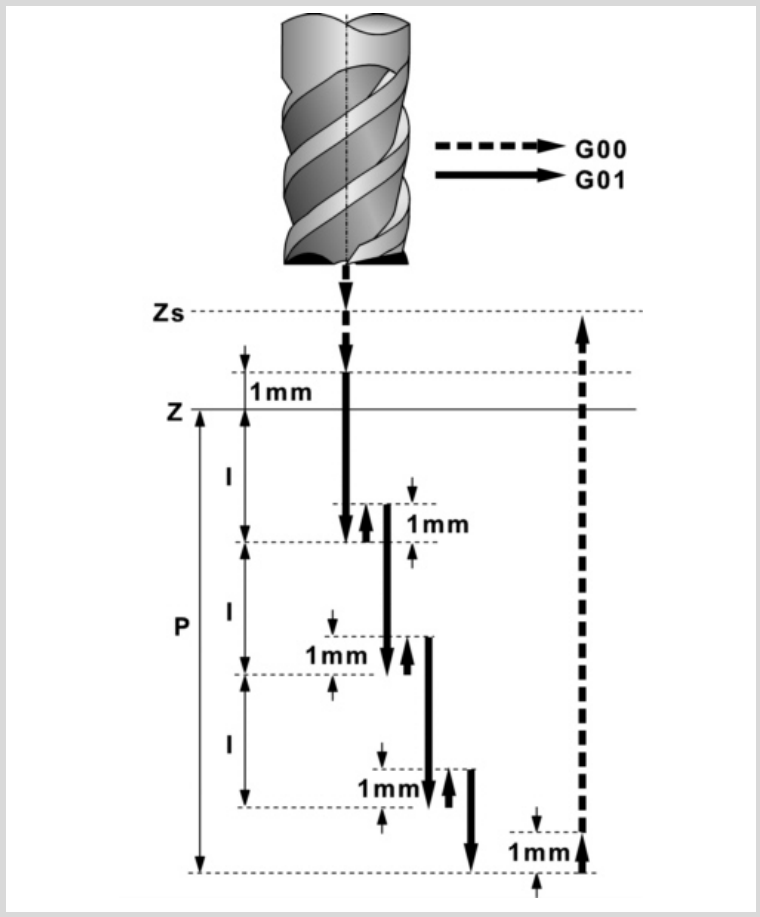
Activating or deactivating the preparation of the next tool.

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

3.18.1 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 "δz".

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

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

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

7 Finishing of the base of the boss.

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

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

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

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

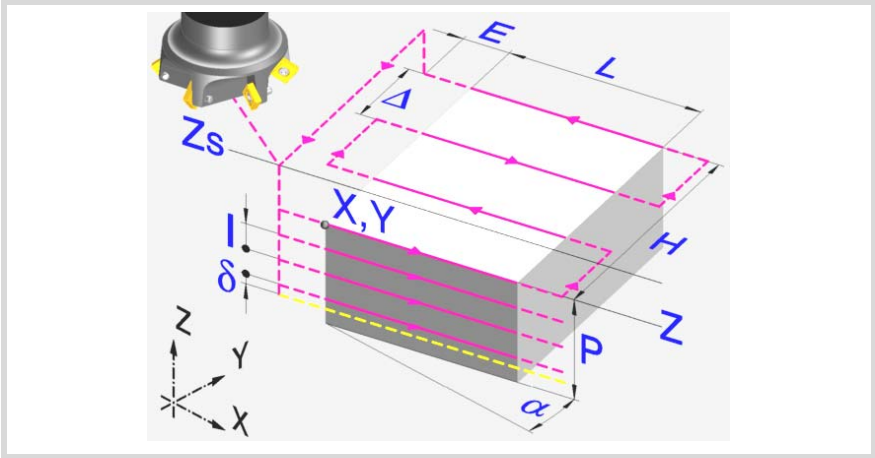
If it has a multiple machining operation associated with it, it executes the following steps as often as necessary:

11 Rapid movement (G0) to the next point.

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

3.19 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.

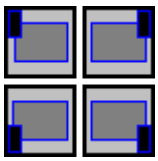
Unidirectional machining along the abscissa axis.

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).



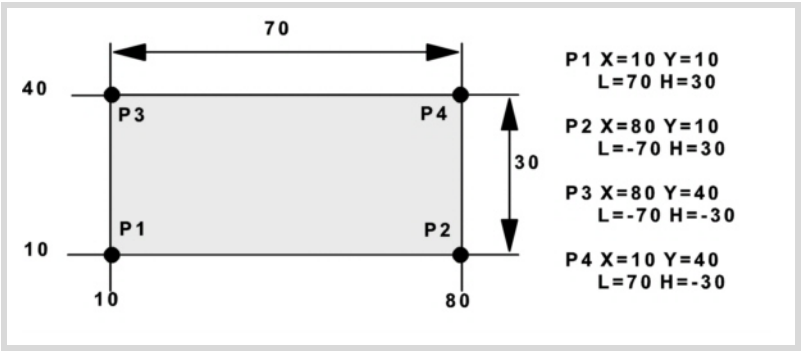
Any of the 4 corners may be selected.

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.



Z Part surface coordinate.

Zs Safety plane coordinate.

P Total depth.

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

Roughing parameters:

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

Δ Maximum milling pass or width.

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

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

Fz Penetration feedrate.

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:

δz Finishing stock.

Δ Maximum milling pass or width.

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

F Surface milling feedrate.

S Spindle speed.

T Roughing tool.

If programmed T=0, there is no roughing.

D Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

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

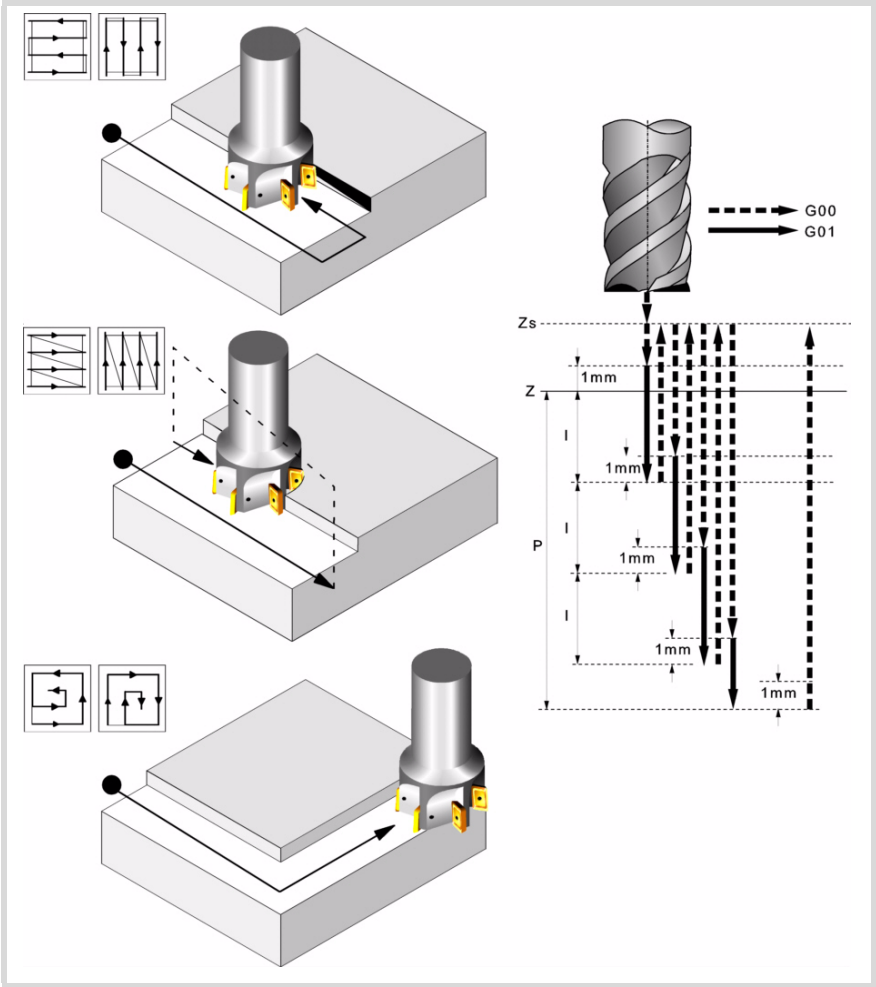
Next tool.

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

3.19.1 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 "δz".

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

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

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

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

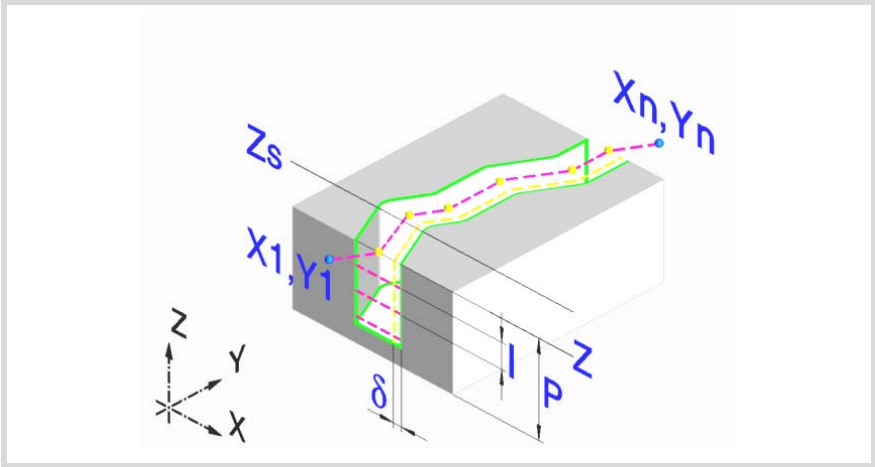
6 Finishing.

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

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

3.20 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.

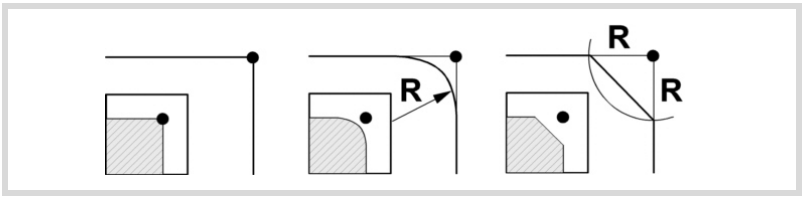


Geometric parameters:

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.

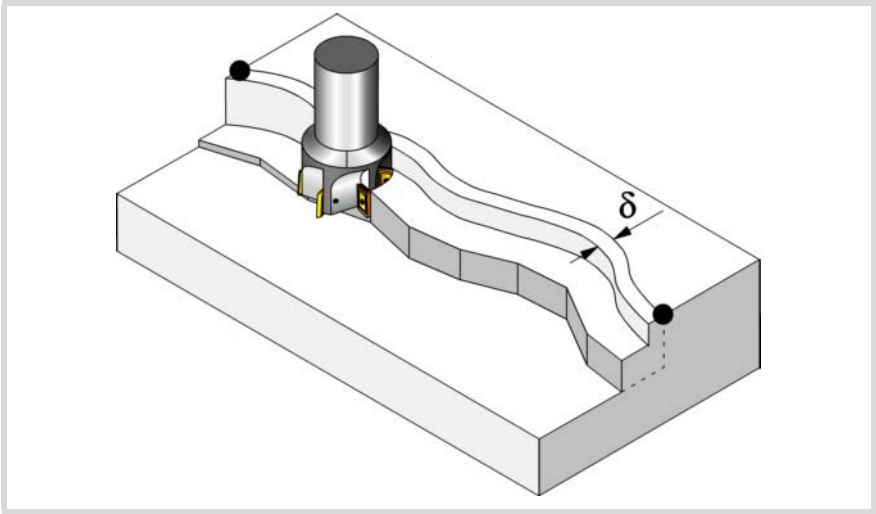


Delete all the points of the profile. Select this icon and press [DEL] to delete all the points of the table.

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

Roughing parameters:

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



The roughing operation defining parameters are:

Fz Penetration feedrate.

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.

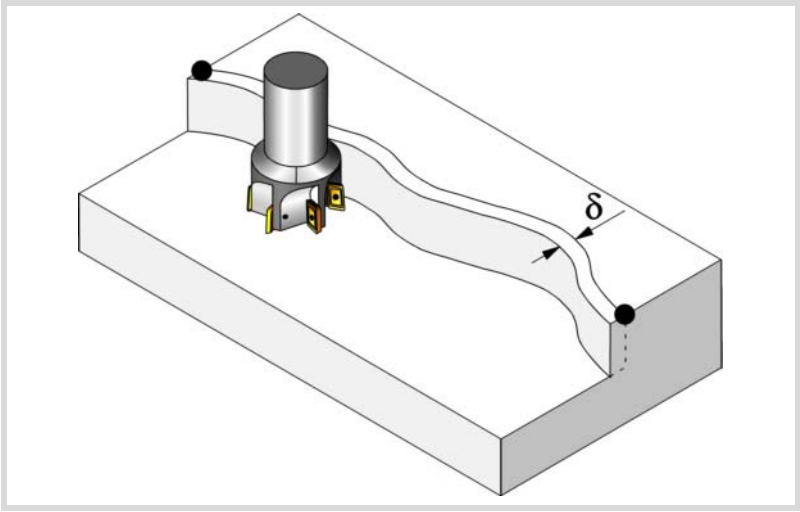
Right-hand compensation.

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).



The roughing operation defining parameters are:

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

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

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

Next tool.

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

3.20.1 Basic operation.

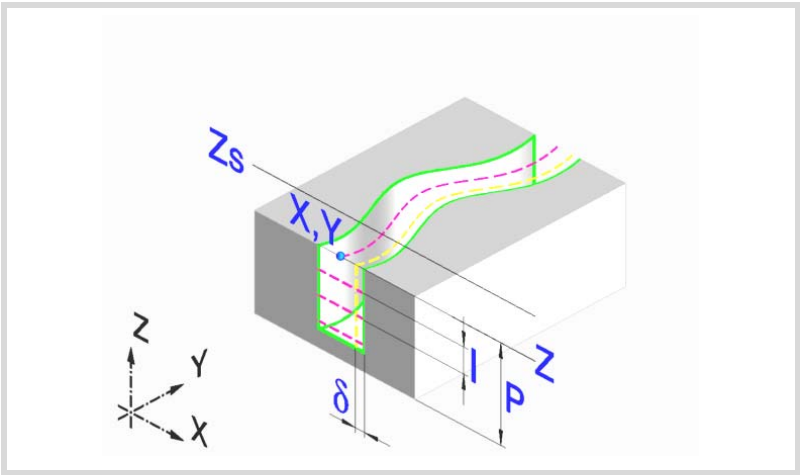
1. It selects the roughing tool and starts the spindle in the requested direction.
2. Rapid movement (G0), up to the XY point and the safety plane (Zs). Depending on the starting plane, it first moves in XY and then in Z or vice versa.
3. Rapid movement (G0) up to the approach plane.
4. Roughing operation.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

3.21 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:

X, Y	Profile entry point

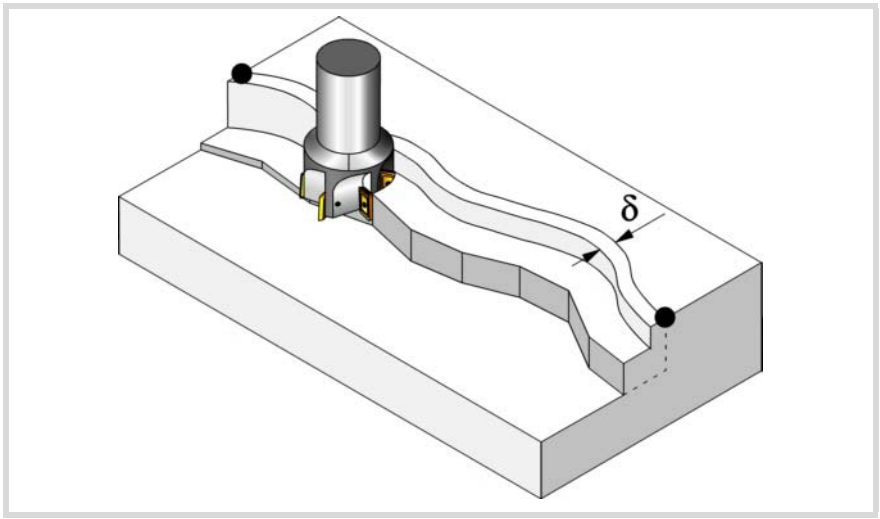
Name of the profile.

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

Z	Part surface coordinate.
Zs	Safety plane coordinate.
P	Total depth.

Roughing parameters:

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



The roughing operation defining parameters are:

Fz	Penetration feedrate.
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.

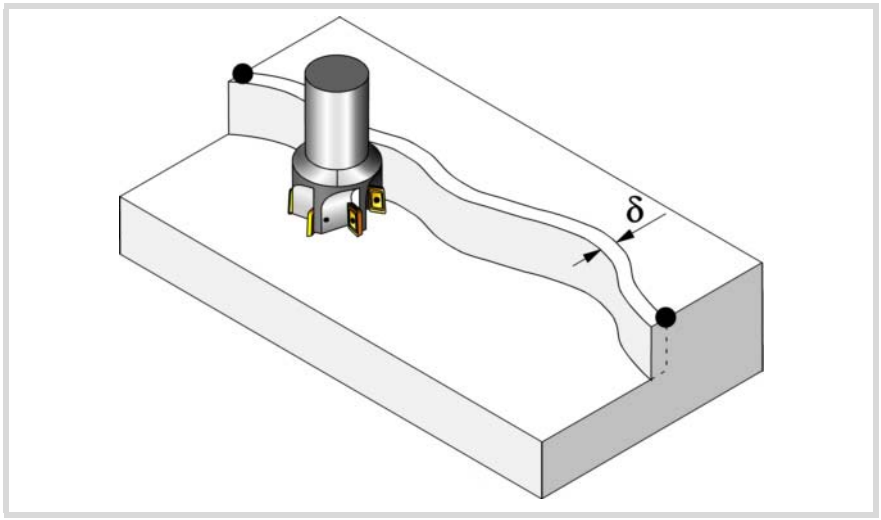
Right-hand compensation.

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. This operation removes the finishing stock (δ).



The roughing operation defining parameters are:

δ Finishing stock on the side walls.

When working without tool radius compensation, the stock (δ) is ignored . In this case, the tool center travel is the same when roughing as when finishing.

F Milling feedrate.

S Spindle speed.

T Finishing tool.

If programmed T=0, there is no finishing.

D Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

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

Next tool.

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

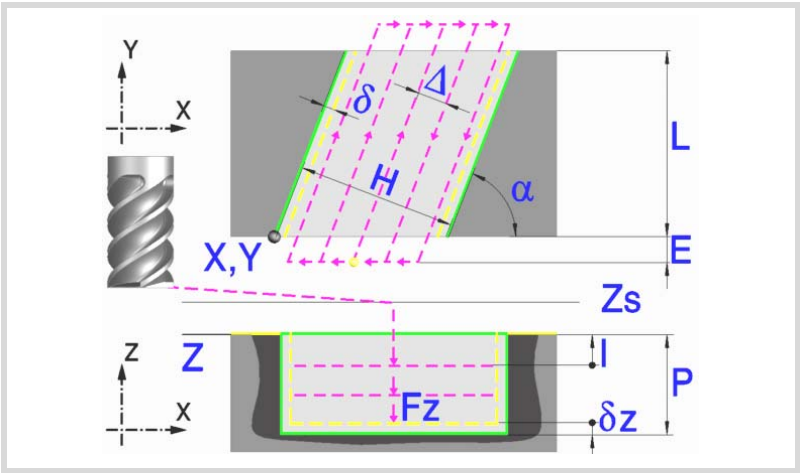
3.21.1 Basic operation.

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).

3.22 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:

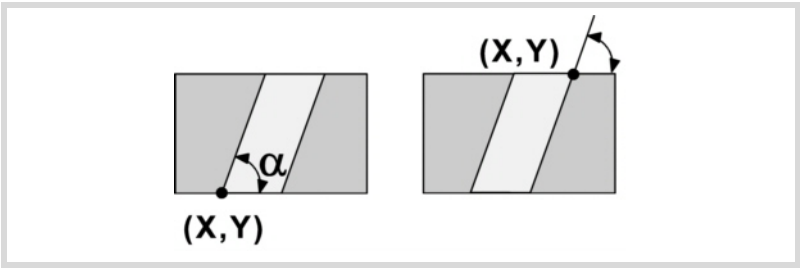
Type of slot milling (icon).

There are 6 possible types.

4 for slot mill each corner of the part.

2 for milling a slot across the part.

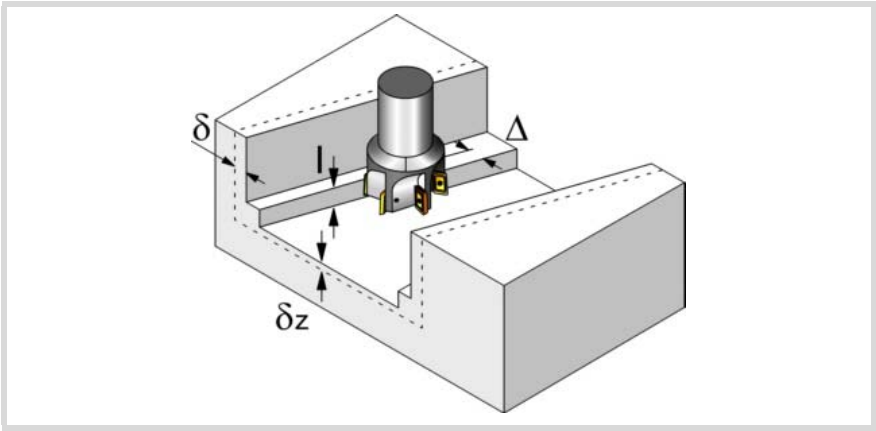
X, Y	Corner where the slot is to be milled.
L, H	Slot dimensions. The sign indicates the orientation referred to the XY point.
Z	Part surface coordinate.
Zs	Safety plane coordinate.
P	Total depth.
α	Angle, in degrees, between the slot and the abscissa axis. The turn is carried out on the defined corner,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

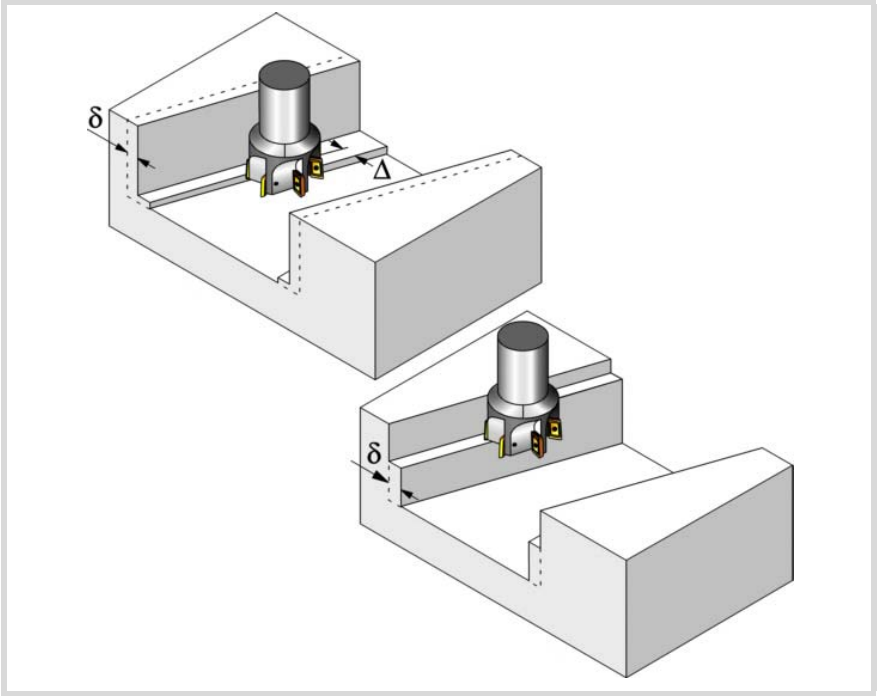
Programming of M functions.


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:

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

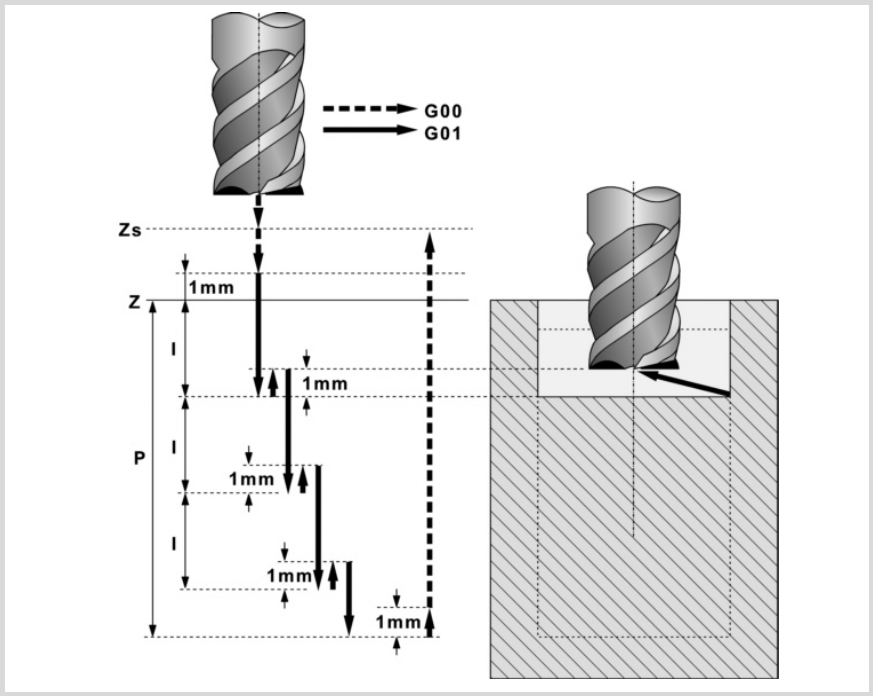
Activating or deactivating the preparation of the next tool.
The editor will only show this option if the user has configured the editor to allow programming the next tool.

3.22.1 Basic operation.

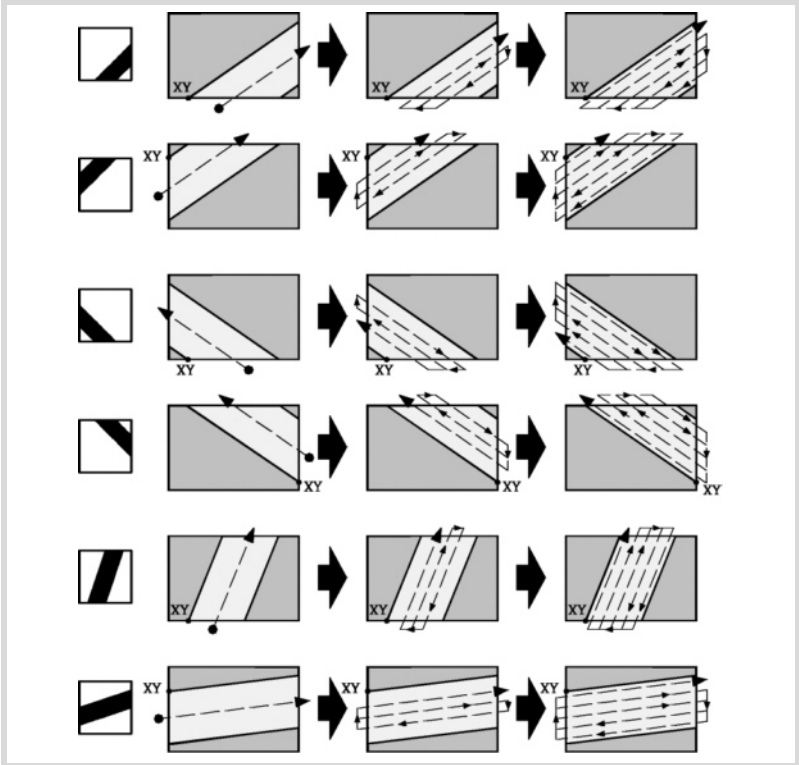
1 It selects the roughing tool and starts the spindle in the requested direction.

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.

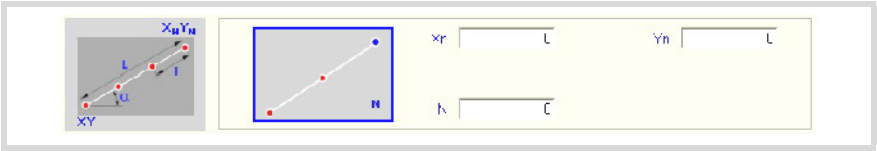
Penetration at feedrate "Fz". Milling of the bottom of the slot up to a distance "δ" from the pocket. It is carried out at finishing feedrate "F" and, if necessary, it recalculates the finishing pass (Δ) so all the passes are identical.

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).

3.23 Multiple machining in straight line.



Definition format (icon).

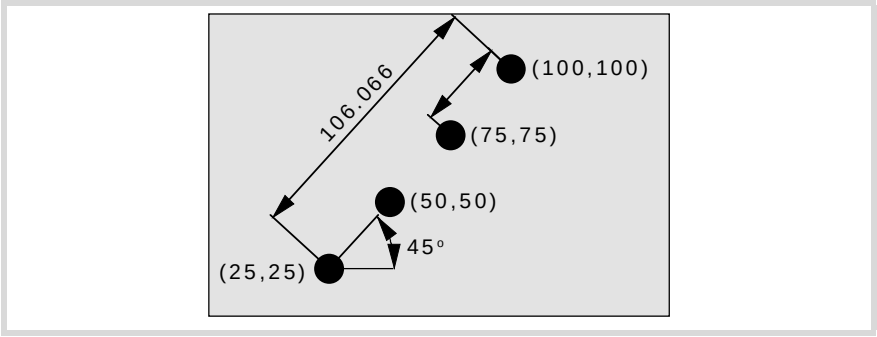
There are 5 different ways to define the machining operation.

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:

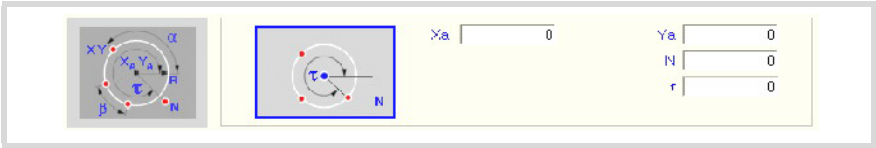
The canned cycle defined at point X25, Y25 is to be repeated at the rest of the points.



We now show the 5 possible ways to define it.

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

3.24 Multiple machining in arc pattern.



Definition format (icon).

There are 9 different ways to define the machining operation.

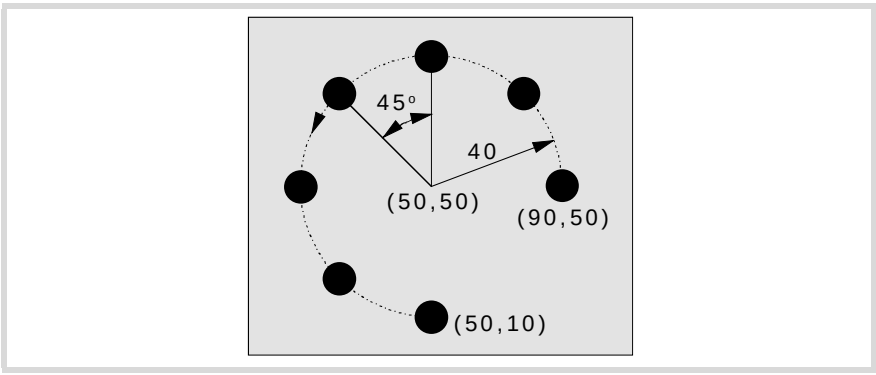
To select the desired one, place the cursor on the icon and press the space bar.

The movement in arc is made counterclockwise. To do it clockwise, define the angular distance between machining operations β 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.



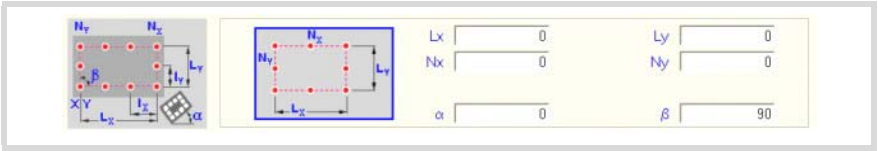
We now show the 9 possible ways to define it.

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

FAGOR AUTOMATION
CNC 8060
CNC 8065
(REF: 1709)

3.25 Multiple machining in rectangular pattern.



Definition format (icon).

There are 3 different ways to define the machining operation.

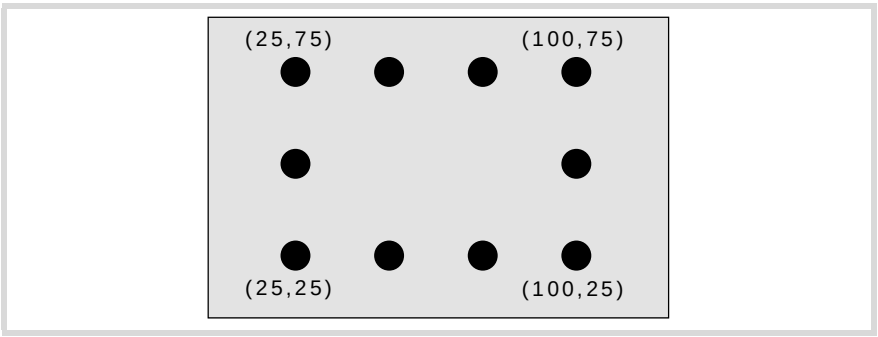
To select the desired one, place the cursor on the icon and press the space bar.

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

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

Programming example:

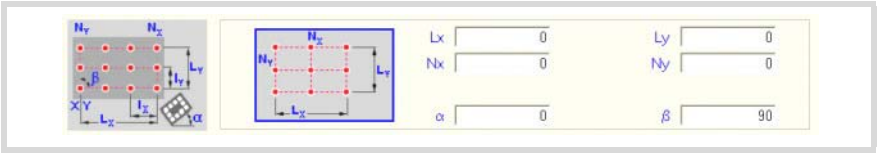
The canned cycle defined at point X25, Y25 is to be repeated at the rest of the points.



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	α 0
	Angle between paths	β 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	α 0
	Angle between paths	β 90
3)	Lengths in X, Y	Lx 75, Ly 50
	Distance between machining operations in X and Y	Ix 25, Iy 25
	Rotation angle	α 0
	Angle between paths	β 90

3.26 Multiple machining in grid pattern.



Definition format (icon).

There are 3 different ways to define the machining operation.

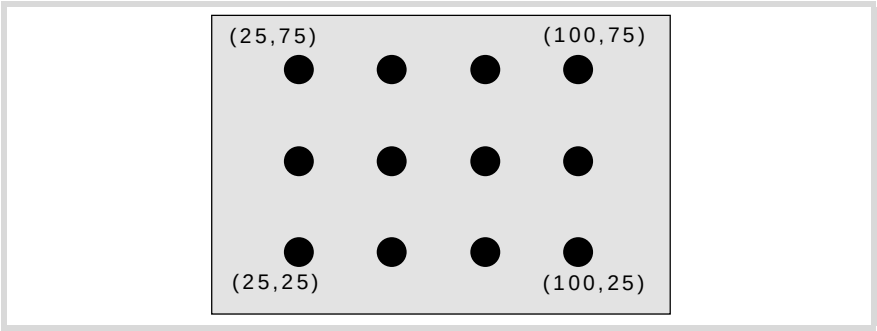
To select the desired one, place the cursor on the icon and press the space bar.

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

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

Programming example:

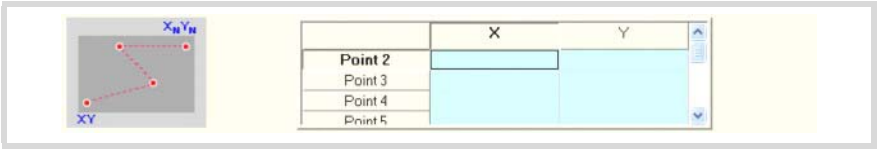
The canned cycle defined at point X25, Y25 is to be repeated at the rest of the points.



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	α 0
	Angle between paths	β 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	α 0
	Angle between paths	β 90
3)	Lengths in X, Y	Lx 75, Ly 50
	Distance between machining operations in X and Y	Ix 25, Iy 25
	Rotation angle	α 0
	Angle between paths	β 90

3.27 Random multiple machining.



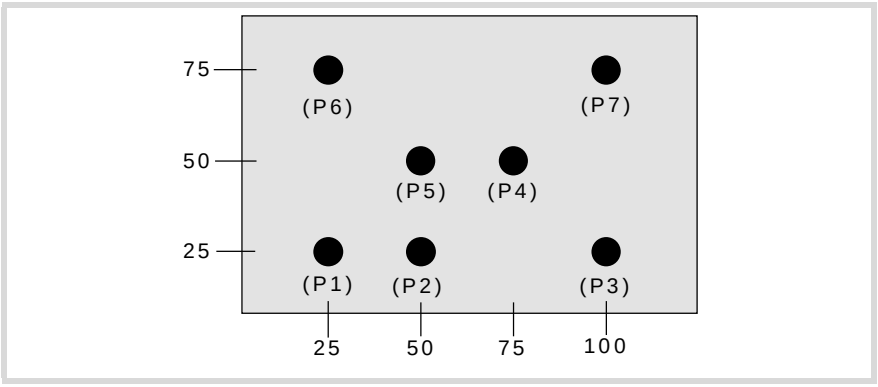
The starting point is the cycle defining point.

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

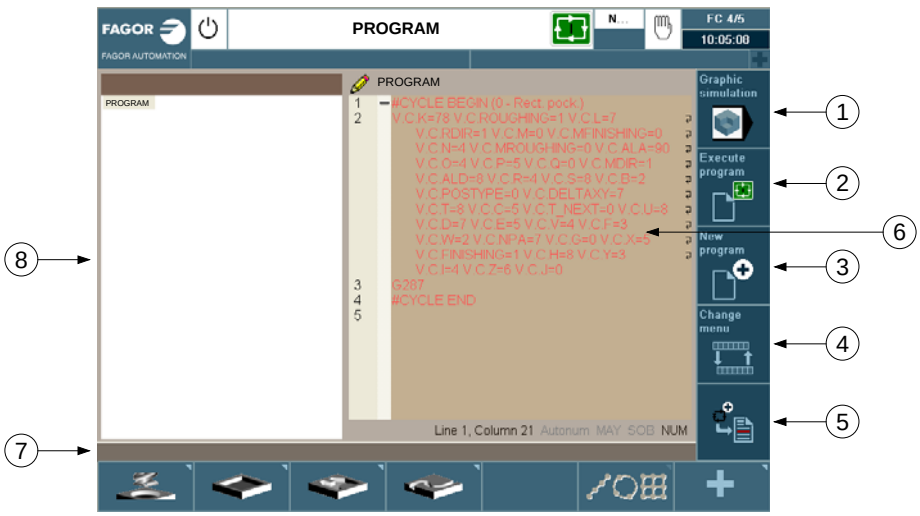
SAVING PROGRAMS

4

Part-programs may be edited, saved, simulated and executed. Each one of these programs is made up by concatenating simple operations or single cycles.

4.1 List of saved programs

Press [EDIT] to access the list of saved part-programs. The CNC will display the following screen:



1 Softkey to start simulating the selected program.

2 Softkey to start executing the selected program.

3 Softkey to create a new program.

4 Softkey that changes the horizontal softkey menu. The sodtkeys appearing when pressing this softkey are:

- Open program.
- Operations with blocks.
- Undo/Redo.
- Insert the cycle being edited.

5 Replace or insert the edited cycle.

6 Window that shows the cycles and ISO-coded blocks that make up the selected part. To edit a cycle, put the cursor on the cycle press [RECALL].

7 Message bar.

8 This softkey shows the list of saved programs. In in this window, it is possible to move through the list of programs. When selecting a program, the right window will show its contents. To change the working folder, press [RECALL] while the focus is on the list and a folder explorer will open to choose the new folder.

To toggle the focus between the list of programs and the part editor, press [CTRL][F2].

4.2 Edit a new part-program

To edit a new program, proceed as follows:

1. Press [EDIT] to access the list of saved part-programs.
2. Press the vertical softkey "New program". The following options will appear:
   - New: Entering a name in the dialog box and pressing the New button will create a new empty file with that name.
   - Copy: Entering a name in the dialog box and pressing the Copy button will create a file with that name and all the cycles and ISO-coded blocks of the program selected at the time on the list of programs will be copied into it.
   - Cancel: To cancel the creation of a new program.

Keyboard shortcuts [CTRL][C] and [CTRL][V] may be used on the programs list to copy and paste a program.

4.3 Delete a new part program

To delete a part-program, proceed as follows:

1. Press [EDIT] to access the list of saved part-programs.
2. Select, with the pointer, the part-program to be deleted from the left column. A program cannot be deleted if it is being executed or simulated.
3. Press the [DEL] key.

The bottom of the screen will show a message requesting confirmation of the deleting operation.

- If [ENTER] is pressed, the CNC will delete the selected program and will update the list of saved programs.
- If [ESC] is pressed, the program will not be deleted and it will quit deleting operation.

4.4 Inserting comments in any part-program.

The block or cycle may be added at the end of the program, after the last operation or insert it between 2 existing operations or replace it with one of the cycles of the program. To save the cycle, proceed as follows:

1. Define the desired cycle assigning the relevant data to it.
2. Press the softkey "Save cycle" to access the list of part-programs.
3. Select the desired program in the left window. The right window shows the contents of the part-program.
4. In the right window, place the cursor at the position where the edited cycle is to be inserted or replaced.
5. Press the vertical softkey to replace or insert the edited cycle and select the desired option.

4.5 Inserting a cycle in the current part-program.

The editor lets insert the cycle being edited directly into the current program in the cursor position.

1. Define the desired cycle assigning the relevant data to it.
2. Press the [INS] key to insert the cycle into the current program in the cursor position.

EXECUTION AND SIMULATION

5

The simulation may be used to graphically show a part-program or an operation with the data used to define it. This way, the simulation may be used to check the part-program or the operation before executing it or saving it and, therefore, correct or modify its data.

It is possible to execute or simulate a part-program or any operation. The simulation or execution may be done from beginning to end or step by step by pressing the [SINGLE] key.

5.1 Execute a part-program

Proceed as follows to execute a part-program:

1. Press [EDIT] to access the list of part-programs stored.
2. Select on the left column the program to be executed.
3. Press the vertical softkey "Execute program".

   After pressing this softkey, the CNC shows the standard screen for the MC work mode. When pressing the two-color key, the CNC shows the auxiliary screen for the MC mode. When pressing the vertical softkey "Graphics", the CNC shows the graphics screen in execution.
4. Once the desired screen has been selected, press [START].

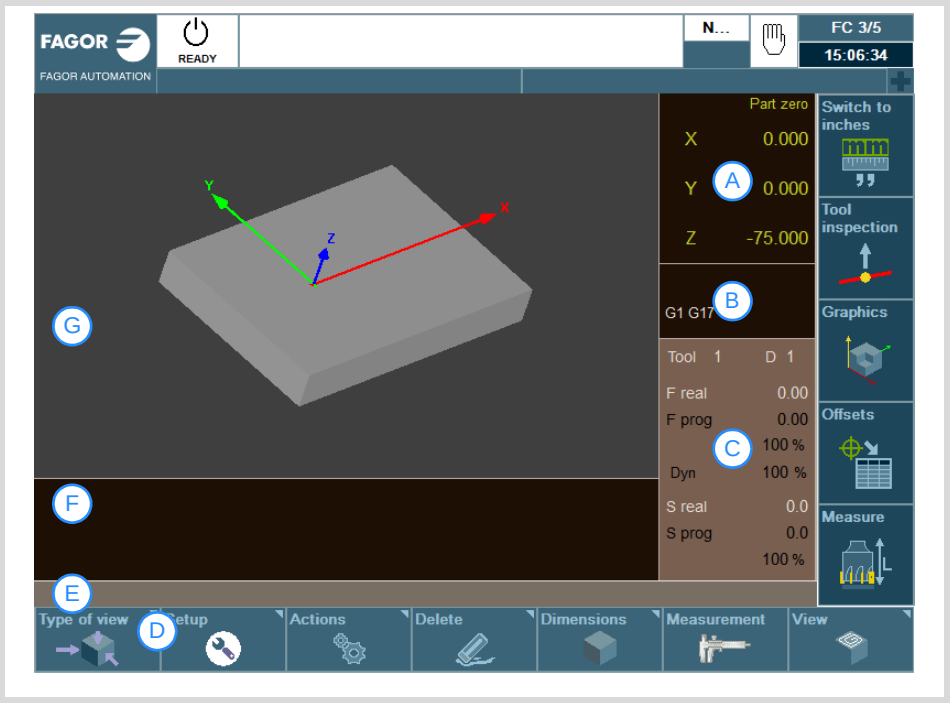
5.1.1 Execute a portion of a part-program

Proceed as follows to execute a portion of a part-program:

1. Press [EDIT] to access the list of part-programs stored.
2. Select the program from the left column and the operation (on the right column) from which to begin executing the part program.
3. Press the vertical softkey "Execute program".

   After pressing this softkey, the CNC shows the standard screen for the MC work mode. When pressing the two-color key, the CNC shows the auxiliary screen for the MC mode. When pressing the vertical softkey "Graphics", the CNC shows the graphics screen in execution.
4. Once the desired screen has been selected, press [START].

5.1.2 Graphics screen in execution.



A Window that informs about the position of the axes during the execution of the program.

B Window that shows the status of the G functions.

C Window that shows the selected tool T, the tool offset D, the real F and the programmed F as well as the real S and the programmed S.

D Softkeys with graphic display options.

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

E Message bar.

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

G Window that shows the lines of the program being executed through graphics.

5.2 Simulating a part-program

Proceed as follows to simulate a part-program:

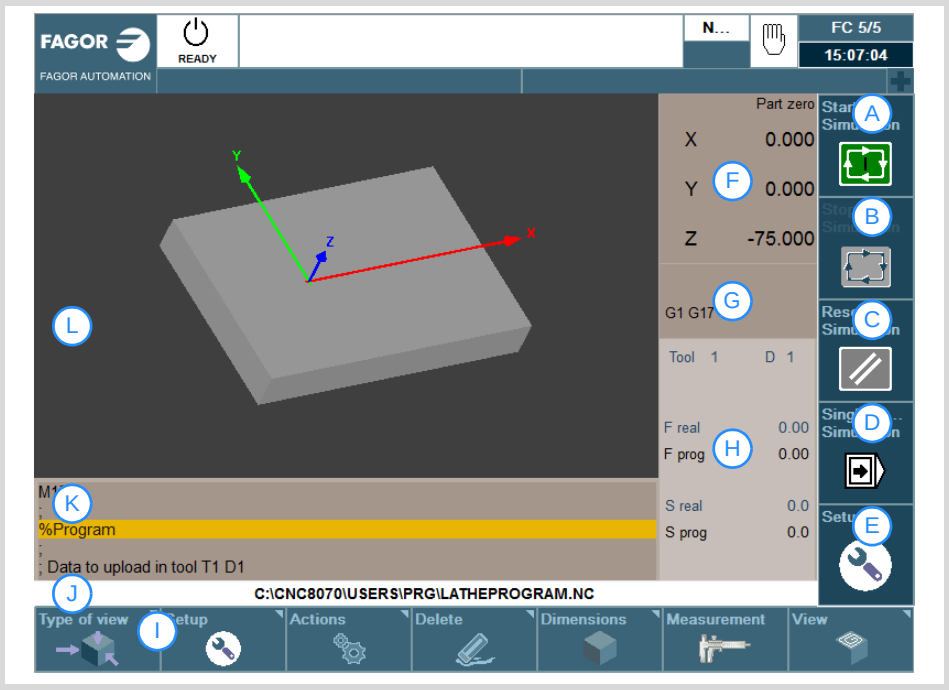
1. Press [EDIT] to access the list of part-programs stored.
2. Select on the left column the program to be simulated.
3. Press the vertical softkey "Graphic simulation". After pressing this softkey, the CNC shows the graphics screen in simulation.
4. Press the vertical softkey "Start simulation".

5.2.1 Simulate a portion of a part-program

Proceed as follows to simulate a portion of a part-program:

1. Press [EDIT] to access the list of part-programs stored.
2. Select the program from the left column and the operation (on the right column) from which to begin executing the part program.
3. Press the vertical softkey "Graphic simulation". After pressing this softkey, the CNC shows the graphics screen in simulation.
4. Press the vertical softkey "Start simulation".

5.2.2 Graphics screen in simulation.

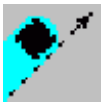


Description of the page items:

- A Softkey to start simulating the selected program.
- B Softkey to stop simulating the selected program.
- C Softkey to reset the execution of the selected program.
- D Softkey to simulate the program block by block.
- E Simulation options.
- F Window that informs about the position of the axes during the execution of the program.
- G Window that shows the status of the G functions.
- H Window that shows the selected tool T, the tool offset D, the real F and the programmed F as well as the real S and the programmed S.
- I Softkeys with graphic display options.
- J Message bar.
- K Window that shows the lines of the program being executed.
- L Window that shows the lines of the program being executed through graphics.

5.2.3 Simulation options.

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



Tool radius compensation

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

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

Conditional stop during simulation

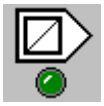
It simulates the external "conditional stop" switch.

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

Software and work zone limits.

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

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



Block jump

It simulates the external "block skip" switch.

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

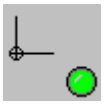
Cancel channel synchronization

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

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

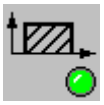
Synchronizing spindles.

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



Assume the active origins for execution.

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



Assume the active zones in execution.

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

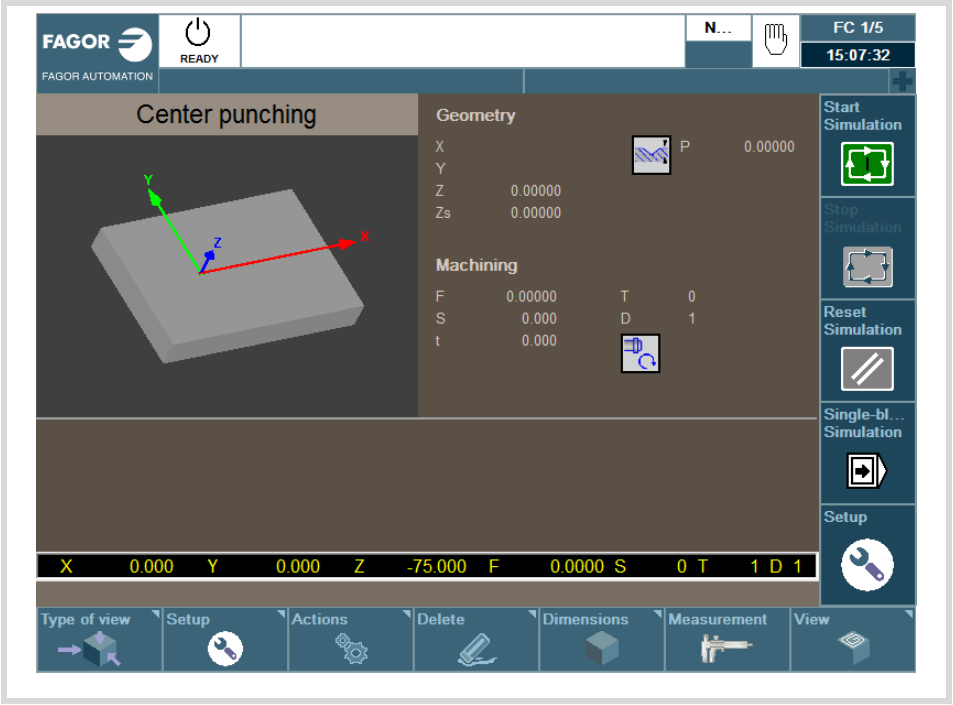
5.3 Simulating or executing an operation that has been saved

Proceed as follows to simulate or execute an operation that has been saved as a part of program:

1. Press [EDIT] to access the list of part-programs stored.
2. Select the program that contains it from the left column and the operation to be simulated or executed from the right column.
3. Press the [RECALL] key.
4. Press the vertical softkey "Simulate cycle" to simulate the operation and the vertical softkey "Execute cycle" to execute it.

5.3.1 Simulating a cycle

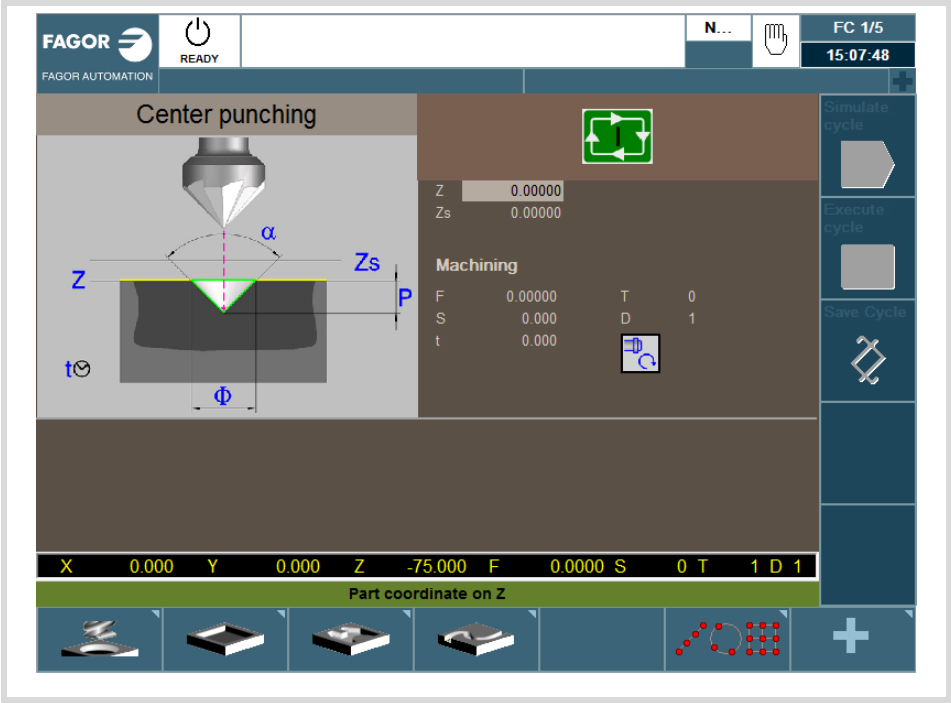
To simulate the edited cycle, press the vertical softkey "Simulate cycle".



For further information on simulating cycles, see the chapter *"2.1.4 Cycle simulation"*.

5.3.2 Cycle execution

To execute the edited cycle, press the vertical softkey "Execute cycle".



For further information on executing cycles, see the chapter *"2.1.5 Cycle execution"*.

User notes:

User notes:

FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

User notes:

FAGOR AUTOMATION

Fagor Automation S. Coop.
Bº San Andrés, 19 - Apdo. 144
E-20500 Arrasate-Mondragón, Spain
Tel: +34 943 719 200
+34 943 039 800
Fax: +34 943 791 712
E-mail: info@fagorautomation.es
www.fagorautomation.com