

**CNC**

**8060/65**

**User quick  
reference**

**Ref. 1906**



FAGOR AUTOMATION



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

The information described in this manual may be subject to changes due to technical modifications. Fagor Automation reserves the right to change the contents of this manual without prior notice.

All the trade marks appearing in the manual belong to the corresponding owners. The use of these marks by third parties for their own purpose could violate the rights of the owners.

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

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

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

# INDEX

**Screen and keys ..... 5**

**Work modes ..... 13**

**ISO language ..... 45**

**ISO Canned cycles (M) ..... 65**

**ISO canned cycles (T) ..... 83**

**Conversational mode ..... 103**

**Conversational cycles (M) ..... 119**

**Conversational cycles (T) ..... 129**

**Measuring and calibration cycles (M)... 137**

**Measuring and calibration cycles (T)... 147**

**High level language ..... 151**

This manual can be used for the PC simulator and the CNC8060/65. The keys included in this manual may vary depending on whether there is a CNC or a simulator. They can also vary depending on the available CNC model.

This manual has been updated for versions up to V05.7x / V01.7x.

## **Simulator**

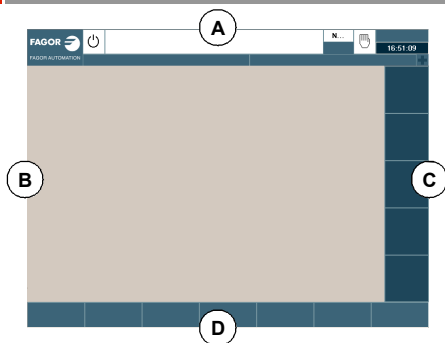
---

The CNC simulator provides the user with a complete tool to create, optimize and check programs directly at the PC. He can then transfer them directly on the machine and run them which means working faster.

If the simulator is configured with the same software version and the same configuration of the real machine (dynamics, functions M, G, etc.), it reproduces the exact behavior of the machine.

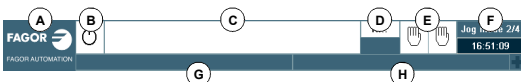
## Screen description

General screen description.



- A. General CNC-status bar.
- B. Screen for the active work mode.
- C. Vertical softkey menu.
- D. Horizontal softkey menu.

## General description of the interface.



- A. Icon (customizable) identifying the manufacturer. Clicking with the mouse or pressing on a touch-screen, the CNC shows the task window (same as pressing the keystroke sequence [CTRL]+[A]) that shows the list of the work modes, user modes, OEM modes and CNC hotkey modes.
- B. Icon showing the status of the program of the active channel:
- C. Program selected in the active channel for execution. Clicking with the mouse or pressing a touch-screen has the same effect as the [Main-Menu] key, which shows the initial screen of the CNC.
- D. Number of the last Nxxxx label executed. The bottom icon indicates whether the Single-block execution mode is active.
- E. Number of channels available and active channel (indicated in blue). Icons indicate which execution mode (Manual, Automatic or MDI), or simulated execution (theoretical, G functions, G S M and T functions, fast and fast [S=0]) is found in each channel. Clicking with the mouse or pressing a touch-screen to access the desired channel, doing it on the icon of the active channel, has the same effect as the [ESC] key.
- F. Active work mode (automatic, manual, etc.) selected screen number and total number of screens available. System clock. By clicking on the active work mode, the CNC shows the list of available pages and which ones are visible.
- G. Active CNC message. If the "+" symbol is displayed on the right hand side of the CNC message, this means that there are other CNC messages on other channels.
- H. PLC messages. If the "+" symbol appearing on the right of the PLC message is blue, this means that there is more than 1 active PLC message. To see the other active messages, click it with the mouse or tap it on the touch-screen or press [CTRL + M].

## Description of the keys

### Monitor & keyboard.

#### Function keys.

Softkeys.

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

#### Browsing keys.



NEXT key. Screen change.

Key that can be configured by the machine manufacturer (OEM).



FOCUS key.

It is used to switch between the different windows of the screen.



BACK key. (\*)

On the horizontal softkey menu, it may be used to go from a softkey submenu up to the previous menu. The ESC key performs the same function.



#### Help key.



HELP key.

Display CNC help.

#### Work modes.



Automatic mode.



Manual mode.



EDISIMU mode.



MDI/MDA mode.



User tables (zero offsets, fixtures and arithmetic parameters).



Tool and magazine table.



Utilities mode.



Configurable mode.

OEM configurable table.

(\*) The BACK key is not available on all keyboards.

## Browsing keys.



Main menu.



Multifunction key:

- Change the active icon status.
- When editing conversational cycles, a copy of the selected profile can be made.
- In conversational mode, it alternates between the standard screen and auxiliary screen.
- In ISO mode, it opens and closes dynamic percentage control (dynamic override).



The SPACE key (space bar) performs the same function.

## Keys to move the cursor.



The arrow keys move the cursor one position to the left, right, up or down.



The previous-page or next-page keys show the previous or next page at the part-program or PLC program editor.



The home and end keys move the cursor the beginning or end of the line.



The tab key moves the cursor to the next field of the active menu.

## Editing keys.



Delete.



Delete.



Insert or overwrite.

It also inserts the cycle being edited by the program called by the cycle itself, or inserts the calculator value at the call calculator point.



Escape key, to cancel the current action without assuming the changes.



Key to validate commands, data and program blocks of the editor.



It allows for the recuperation and modification of a program profile or a cycle that has already been edited.



Calculator (\*).

(\*) The calculator key is not available on all keyboards.



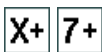
## Jog panel

### Turn the CNC off.

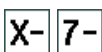


Turn the CNC off.  
To choose Shut down, Cancel or Restart.

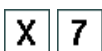
### Jog keyboard for jogging the axes.



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



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



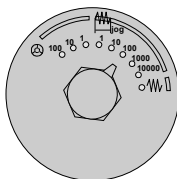
Keys to select the axes and keys to select the jogging direction. Both keys (axis and direction) must be pressed to jog the axis.



Rapid key. When pressing this key while moving an axis, the CNC applies the rapid feedrate.

### Feed selectors.

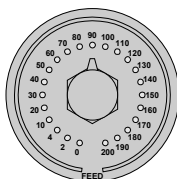
#### Selector of the manual movement type.



Continuous jog, incremental jog or handwheel.

- In handwheel mode, it selects the multiplying factor for the handwheel pulses (x1, x10 o x100).
- In incremental mode, it selects the incremental value of the axis movements.

#### Feedrate override % selector.



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

## Execution keys.



Cycle start key [START].

Execute the selected program in automatic mode, a block in MDI/MDA mode, etc.



Cycle stop key [STOP].

Interrupt the execution of the CNC.



Reset key.

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



Single block execution mode.

When selecting the "single block" execution mode, the program simulation will be interrupted at the end of each block.



Machine reference zero (home) search.

This key only works in MANUAL mode. It can be used directly, where it calls the home search subroutine for all axes, or it can be used after selecting an axis, so that it only looks for the machine reference for that axis.

## Spindle control.



Start the spindle clockwise.



Stop the spindle.



Start the spindle counterclockwise.

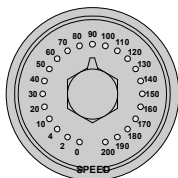


Percentage variation of the spindle speed.







Spindle orientation.

## Speed override % selector.



Selector of percentage of spindle speed override between 0% and 200%.

## Keyboard shortcuts

	<b>Browsing keys.</b>
	[CTRL]+[F1] Previous menu.
	[CTRL]+[F2] Switch window.
	[CTRL]+[F3] Switch screens. Configurable keys
	[ALT]+[B] Two-color key
	<b>Operations at the interface.</b>
	[CTRL] + [W] Minimize/Maximize the CNC. It can be configured by the machine manufacturer (OEM).
	[CTRL] + [J] Show / hide the virtual operator panel.
	[CTRL] + [M] Show / hide the PLC message list.
	[CTRL] + [O] Show / hide the CNC message list.
	[ALT] + [W] Show / hide the window for errors and warnings.
	[ALT] + [F4] Turn the CNC off.
	[ALT] + [-] Expand and collapse cycles/profiles in the editor.
	[SHIFT] + [Up / Right Arrow] Increase simulation speed.
	[SHIFT] + [Down / Left Arrow] Reduce simulation speed.

## Work modes.

[CTRL] + [A]  
To show the task window.



[CTRL] + [SHIFT] + [F1]  
Main menu.



[CTRL] + [F6]  
Automatic mode.



[CTRL] + [F7]  
Manual mode.



[CTRL] + [F9]  
EDISIMU mode, in other words editing and simulation.



[CTRL] + [F8]  
MDI/MDA mode.



[CTRL] + [F10]  
User tables.



[CTRL] + [F11]  
Tool and magazine table.



[CTRL] + [F12]  
Utilities mode.



[CTRL] + [K]  
Calculator.

## Execution keys.



[CTRL]+[S]  
Cycle start key [START].



[CTRL]+[P]  
Cycle stop key [STOP].








[CTRL]+[R]  
Reset key.



[CTRL]+[B]  
Single block execution mode.

*The shortcuts for the [START] [STOP] and [RESET] keys are only available when the CNC is installed as simulator on a PC.*

## Jog mode

Softkey	Meaning
	Description of the manual mode screen.
	Change the units for data display (mm or inches). For programming, the CNC assumes the units defined with the active function G70 or G71, or, when not programmed, the units set by the machine manufacturer (INCHES parameter).
	Setting and activating the zero offsets and the fixture offsets. This softkey shows the zero offsets and the fixture offsets of the system, either to store the active zero offset or to activate a new zero offset.
	Tool calibration / measurement (·M· model).
	Tool calibration / measurement (·T· model).
	Part centering (·M· model).

## 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.

### Keyboard.



1. Select the axis to be homed (on alphanumeric keyboard). The CNC will highlight the coordinate of that axis.



2. Press the homing key [ZERO]. The CNC will display the symbol "1" in the numeric area.



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

### Softkey menu.

1. Press the home search softkey to show the list of axes of the channel.
2. Select the axis to be homed on the softkey menu. The CNC will highlight the coordinate of that axis and will show the symbol "1" in the numeric area.
3. Press [START] to go ahead with the home search or [ESC] to cancel the operation.

## Automatic home search (with subroutine)

### Keyboard.



1. Press the homing key [ZERO]. The CNC will display the symbol “1” in the numeric area.



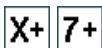
2. Press [START] to go ahead with the home search or [ESC] to cancel the operation.

### Softkey menu.

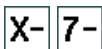
1. Press the home search softkey to show the list of axes of the channel.
2. Select the “All” option on the softkey menu.
3. Press [START] to go ahead with the home search or [ESC] to cancel the operation.

## Move the axes

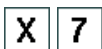
### JOG keypad.



Select an axis and move it in the positive direction.



Select an axis and move it in the negative direction.



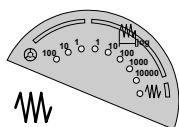
Keys to select the axes and keys to select the jogging direction. Both keys (axis and direction) must be pressed to jog the axis.



Move the axis in rapid.

### Jogging.

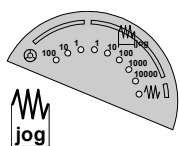
#### Movement in continuous jog.



In continuous jog, the axes keep moving while the jog keyboard is acted upon.

1. Turn the jog selector switch to the continuous JOG position.
2. In incremental mode, it selects the incremental value of the axis movements.

#### Movement in incremental jog.



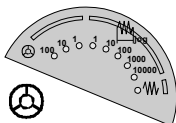
In incremental jog, the axis moves a specific distance every time the key is pressed.

1. Turn the jog selector switch to one of the incremental jog positions.
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.



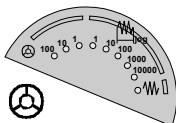
Jogging  
(via handwheel).

**General handwheel (it may be used to jog any axis of the machine).**



1. Turn the jog selector switch to one of the handwheel positions.
2. Select the axis or axes to be jogged on the jog keyboard. The CNC will highlight the selected axes.
3. 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.

**Individual handwheel (it is associated with a particular axis).**



1. Turn the jog selector switch to one of the handwheel positions.
2. 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.

## Coordinate preset (temporary offset)

The [ESC] key may be used to cancel the operation at any time.

### Keyboard.



1. Select the axis to be preset (on alphanumeric keyboard). The CNC will highlight the coordinate of that axis.

2. Key in the desired preset value.



3. Press [ENTER] to assume the entered value.

### Keyboard (using the calculator).

1. Select the axis to be preset (on alphanumeric keyboard). The CNC will highlight the coordinate of that axis.

2. Open the calculator using the [CALC] key or by pressing [CTRL] + [K]. The value shown by the axis goes directly into the calculator and any operation can be performed (for example: /2 [ENTER]).

3. Press the [INSERT] key to directly enter the value of the calculator for the selected axis.

4. Press [ENTER] to assume the entered value.

### Softkey menu.

1. Press the softkey for presetting axes to show the list of axes of the channel and select an axis. The CNC will highlight the coordinate of that axis.

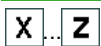
2. Key in the desired preset value.

3. Press [ENTER] to assume the entered value.

## Move an axis to a particular position

The [ESC] key may be used to cancel the operation at any time.

### Keyboard.



1. Select the axis to be moved (on alphanumeric keyboard). The CNC will highlight the coordinate of that axis.

2. Enter the coordinate of the target point.



3. Press [START] to make the move.

### Keyboard (using the calculator).

1. Select the axis to be moved (on alphanumeric keyboard). The CNC will highlight the coordinate of that axis.

2. Open the calculator using the [CALC] key or by pressing [CTRL] + [K]. The value shown by the axis goes directly into the calculator and any operation can be performed (for example: /2 [ENTER]).

3. Press the [INSERT] key to directly enter the value of the calculator for the axis to be moved.

4. Press [START] to make the move.

### Softkey menu.

1. Press the softkey for presetting axes to show the list of axes of the channel and select an axis. The CNC will highlight the coordinate of that axis.

2. Enter the coordinate of the target point.

3. Press [START] to make the move.

## Set the feedrate, speed or tool

### Feedrate.

**F**

1. Press [F] at the alphanumeric keyboard.
2. Enter the new feedrate directly or with the calculator if a calculated value is to be used.
3. Press [START] to assume the entered value or [ESC] to cancel the operation.



### Speed.

**S**

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.



### Tool.





**T**

1. Press [T] on the alphanumeric keyboard.
2. Enter the tool to be selected.
3. Press [START] to assume the entered value or [ESC] to cancel the operation.




*Note: this command initiates a tool change that may require the movement of axes and mechanical parts of the machine.*

## Master spindle control

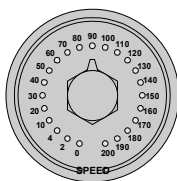
	Keyboard.
	Start the spindle counterclockwise (same as the function M03) at the active speed.
	Start the spindle clockwise (same as the function M04) at the active speed.
	Stop the spindle (same as M05 function).
	Orient the spindle (same as M19 function).

## Vary the speed override from the operator panel.

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





	Keyboard.
	<b>JOG keypad.</b> 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.

### Switch.



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

## Tool calibration

Softkey	Meaning
	Description of the softkeys of the tool calibration.
	Tool calibration in a mill model.
	Tool calibration in a lathe model.
	If there is no tabletop probe, only manual calibration is available. All types of calibration are available when using a table-top probe. The different calibration methods may be selected from the vertical softkey menu.
	
	<p><b>Manual calibration (without a probe).</b></p> <p>In this mode, only the active tool can be calibrated. Since there is no probe, a reference part is required to calibrate the tool. All the movements are carried out manually.</p> <p><b>Semi-automatic calibration (with a probe).</b></p> <p>The positioning movements are carried out manually and the CNC executes the probing movements.</p> <p><b>Semi-automatic calibration (with a probe).</b></p> <p>The CNC executes all the movements using the calibration canned cycle #PROBE.</p>
	Probe selection. (*)

The CNC uses the active probe for calibration. The active probe may be changed via part-program or MDI using the instruction #SELECT PROBE.

#SELECT PROBE [1]      #SELECT PROBE [2]

(\*) Depending on the software version and when the manufacturer has configured it, the CNC is capable of assigning the correct probe for each operation.

**Manual calibration without a probe.**

All the movements are carried out manually. 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. In this mode, only the active tool can be calibrated.

**Milling model.**

Calibrate the length of the endmills and the offsets of the lathe tools.

**Lathe model (plane).**

Calibrate the offset of any tool.

**Lathe model (trihedron).**

Calibrate the length or offsets of the endmills and the offsets of the lathe tools.

Tool calibration steps.

1. Define the dimensions of the reference part being used in the calibration.
2. Define the tool and the offset to be calibrated and press [START] to execute the tool change (if [ENTER] is pressed, the CNC will only show the tool data).
3. Calibrate the tool. Approach the tool manually until touching the part and then validate the calibration using the softkey menu. After validating the calibration, it updates the values and initializes the wear value to zero. Then, the new values are saved in the tool table.
4. Press [START] for the CNC to assume the new values of the offset.

### Semi-automatic calibration with a probe.

The positioning movements are carried out manually and the CNC executes the probing movements. The CNC will move the tool on the selected axis until it touches the probe and validates the calibration only on that axis. In this mode, only the active tool can be calibrated.

#### **Milling model.**

Calibrate the length or radius of the endmills and the offsets of the lathe tools.

#### **Lathe model.**

Calibrate the offset of any tool.

Tool calibration 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. Define the tool and the offset to be calibrated and press [START] to execute the tool change (if [ENTER] is pressed, the CNC will only show the tool data).
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 tool 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. 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.



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

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. This mode may be used to calibrate any tool.

**Milling model.**

Calibrate the length, the radius or offsets of the endmills and the offsets of the lathe tools.

**Lathe model (plane).**

Calibrate the offset of any tool.

**Lathe model (trihedron).**

Calibrate the length, the radius or offsets of the endmills and the offsets of the lathe tools.

Tool calibration steps.

1. Select the tool and the offset to be calibrated.
2. Define the data defining the calibration.
3. Press the [CYCLE START] key to start the calibration.  
The CNC calibrates the tool making all the necessary movements; there is no need to manually approach the tool. If necessary, the CNC makes the tool change.
4. After the calibration It updates the tool table data.  
Also, the CNC assumes the new values.

## Part centering (mill model)

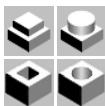
Part centering is available in the jog mode. This option is only available at the mill model. To quit the part centering mode and return to jog mode, press the [ESC] key.

### Accessing part centering.



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

### Type of part to be centered.



- Rectangular boss.
- Circular boss.
- Rectangular pocket.
- Circular pocket.

### Cycle equivalent to part centering.

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

## MDI / MDA mode

### Edit new blocks.

- In MDI mode, the edit line is always visible.
- In MDA mode, one must select the "new block" option from the softkey menu.

### Modify a block from the block history.

- In MDI mode, use the [↑][↓] keys to display the history and scroll through it. The [ENTER] key restores from the history the block selected with the cursor and insert it in the edit line.
- In MDA mode, use the [↑][↓] keys, select a block from the history and use the "modify" option from the softkey menu (or the [ENTER] key) to copy it into the edit line.

### Execute blocks.






- The [START] key executes the block currently displayed on the editing line. Once the block has been executed is saved in the block history. The [START] key also directly executes the line where the cursor is located on the history, if this line is not e be modified.
- The [STOP] key interrupts the execution of the block. Press [START] again to resume execution from where it was interrupted.

Cancel block



- While the execution is interrupted, the "CANCEL" softkey cancels the execution of the block while keeping the programmed machining conditions (it does not do a general reset of the CNC).
- The [RESET] key cancels the execution of the block and resets the CNC to its initial conditions.

## Automatic mode

Softkey	Meaning
	Description of the softkeys of the automatic mode.
	Select a program for execution.
	Begin tool inspection. Tool inspection is only available when program execution is interrupted.
	End simulated execution and start executing the program
	Select the program that is being edited.

## Program execution

### Select a program.

Each channel executes the program selected in it. To select a program, press one of the following softkeys of the vertical menu.



This softkey opens a browser that displays the programs that are in the CNC memory.



This softkey directly selects the last program opened in EDISIMU mode.

### Execute a program.

The name of the program selected in the channel for execution appears on the general status bar. If not indicated otherwise, the program execution will begin from the first block of the program to the execution of one of the end-of-program functions "M02" or "M30". As an option, it is possible to define the first and last blocks of the execution.



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



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



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



Single block execution mode. The program may be executed in –single block– or –automatic– mode; the mode may be selected even while executing the program.

## Execute blocks separately.

Press the [EXBLK] softkey of the horizontal menu. Being this option active, every time the [START] key is pressed, it only executes the block selected in the active program. Once that block is executed, another block may be executed by selecting it with the cursor and pressing [START] again and so on. Blocks executed like this change the history of the M and G functions.

*Note: the [EXBLK] softkey is an activation/deactivation softkey. In order to start the execution of the program, it must be deactivated.*

## Simulated execution of a program.

With simulated execution, it is possible to simulate a program, interrupt it at a point and start execution from that point on. Depending on the type of simulation selected, it can involve movement of axes, spindle, etc.

	Path	Axis movement	Spindle control	Send M-H-S-T to the PLC	G04	M00 M01
Theoretical path	Programmed tool Path	No	No	No	Yes	Yes
G functions	Tool center	No	No	No	Yes	Yes
Functions G M S T (*)	Tool center	No	No	Yes	Yes	Yes
Main plane (*)	Tool center	Yes (plane)	Yes	Yes	No	Yes
Rapid	Tool center	Yes	Yes	Yes	No	Yes
Rapid [S=0]	Tool center	Yes	No	Yes	No	Yes

(\*) These can cause collisions, depending on the type of machine.

### Start the simulation of the program.

1. On the horizontal softkey menu, select the desired type of simulation.
2. If necessary, set the desired simulation conditions (first and last block)
3. Press the [START] key to start the simulation. The program may be simulated in "single block" or "continuous" mode; the mode may be selected even while simulating the program.

**End simulation and start executing the program.**

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

2. When switching to execution mode (after pressing the softkey), the CNC goes into tool inspection to reposition the axes, modify program conditions, etc. To complete the tool inspection and before starting the execution of the program, the spindle turning direction must be restored and the axes repositioned. The vertical softkey menu offers two options.



- Repositioning the axes at the interruption point.



- Repositioning the axes at the starting point of the interrupted block.

3. Press the [START] key to start the execution.

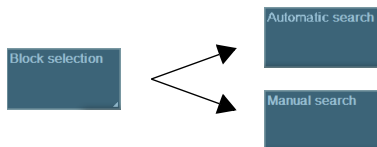
## Block search

Using block search, it is possible to restore the program history up to a particular block in such way that if the program is executed from that block on, it will do so under the same conditions as if it were executed from the beginning.

- The automatic block search may be used to recover the program history up to the block where the previous execution was canceled. The CNC remembers in which block the execution was interrupted, thus not being necessary to set the stop block.
- The manual block search may be used to recover the program history up to a particular block of the program or of the subroutine, set by the operator.

### Executing block search.

1. Selecting the type of search: automatic or manual.
2. Selecting the stop block. In the automatic block search, there is no need to select the stop block; by default, the CNC runs the search up to the block where the program was interrupted.
3. Selecting the starting block for the search. If the first block is not selected, the block search starts at the beginning of the program.
4. Press the [CYCLE START] key to start the block search.
5. Depending on how the treatment of functions M, H, F, S is configured, it may be necessary to decide which ones are sent out to the PLC.
6. Reposition the axes to the point to resume execution.
7. Tool inspection may be accessed to change the machining conditions.
8. Press [START] to execute the program.





### Repositioning the axes.

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

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

### Changing the machining conditions.

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

## Tool inspection

### Begin tool inspection.



Tool inspection may be accessed from the vertical softkey menu only when the execution of the program has been interrupted ([STOP] key) and when the program is not being executed. After activating tool inspection, it is possible to jog the axes using the jog keyboard, act upon the master spindle of the channel from the operator panel and execute blocks from the MDI/MDA mode.

### Execute blocks in MDI/MDA mode.

Any program block may be executed in MDI/MDA mode. The conditions when entering the MDI/MDA mode will be those of the interruption point; i.e. the CNC maintains the history of active G and M functions, feedrate, spindle speed, tool and other commands that were programmed. However, the CNC treats certain functions and commands (type of movements, radius compensation, etc.) differently. Refer to the operating manual.

In general, all the changes made in MDI/MDA mode are kept active when resuming the program after tool inspection except the following functions that are restored at the time of interruption; type of interpolation (G00, G01, G02, G03, G33 or G63), G90/G91 function or #MCS function.

### Repositioning the axes and the spindle.

To complete the tool inspection and before resuming the execution of the program, the spindle turning direction must be restored and the axes repositioned.



Repositioning the axes at the interruption point.



Repositioning the axes at the starting point of the interrupted block.



Canceling repositioning.

### Repositioning the axes.

The CNC allows repositioning the axes either one by one or in groups. Use the vertical softkeys to select the axes to be repositioned and press [START]. The CNC will reposition the axes at the selected point (according to the softkey selected earlier) at the feedrate set by the machine manufacturer. Once one axis has reached its position, the repositioning will no longer be available.

### Repositioning the master spindle.









If the status of the master spindle has changed during the inspection, the softkeys will show the M3, M4, M5 or M19 function to restore. The spindle turning status may be restored either together with the repositioning of the axes or separately. If the spindle was interrupted in a positioning with M19, repositioning will complete that positioning.

### Resuming the execution of the program.

Once all the axes are repositioned or after canceling repositioning, press [START] to resume program execution.

- If tool inspection has ended by repositioning all the axes, when pressing [START], the CNC completes the interrupted path and goes on with the rest of the program.
- If tool inspection has ended after canceling the repositioning of the axes, when pressing [START], the axes move from their current position to the end point of the interrupted path and then the CNC goes on with the rest of the program.

## Edisimu mode (editing and simulation)

Softkey	Meaning
	Description of the softkeys of the edisimu mode.
	START (simulation). Starts program simulation or resume it if it was interrupted.
	STOP (simulation). Interrupt program simulation. Simulation will resume by pressing the START softkey.
	RESET (simulation). Cancel program simulation. If an error occurs during simulation, reset eliminates the error status and returns the simulation mode to its initial conditions.
	Change the channel being displayed for editing and simulation. It does not affect the active channel at the CNC. (This icon will only be available when the CNC has channels).
	Select the "single block" or "continuous" execution mode. When "single block" mode is active (the icon will appear pressed), program simulation will be interrupted at the end of each block.
	Analyze the program looking for syntax errors. The syntax check is not available for programs written in 8055 CNC language.
	Offer an estimate of the total execution time at 100% of the programmed feedrate. This softkey is only accessible from the statistics screen and the result is displayed on the same screen.
	Configuring simulation options.

## Program simulation

### Select a program.

The "Open program" softkey is used to select a program in EDISIMU mode and may be a new program or an existing one. A different program may be edited and executed in each channel. When selecting this option, the CNC shows a list of the available programs.

1. Select the folder that contains the program. If it is a new program, it will be saved in this folder.
2. Select the program from the list or write its name in the bottom window. To edit a new program, write the name of the program in the lower window and the CNC will open an empty program or a predefined template depending on how the editor is configured.

*Note: depending on the name of the program and the configuration of the editor, editing is directly activated in 8055 mode.*

3. Press [ENTER] to accept the selection and open the program or [ESC] to cancel it and close the program listing.

### Simulating a program.

1. Choose the type of graphic representation, its dimensions and the point of view. This data may also be modified during the simulation of the program.



2. Activate the desired simulation options using the softkey menu.



3. Press the START softkey to start the simulation. The simulation may be interrupted with the STOP softkey or canceled using the RESET softkey.



The simulation of the program starts at the first block of the program and ends after executing one of the end-of-program functions "M02" or "M30". As an option, it is possible to define the first and last blocks of the simulation. To simulate the program, the CNC assumes the real configuration of the spindles of the channel and the configuration of the machine parameters.



"Single block" simulation. The program may be simulated in –single block– or –automatic– mode; the mode may be selected even while simulating the program.

The horizontal menu that appears during a graphical simulation is dependent upon whether the cursor is in the graphic section (options for handling graphics, measurement, etc.) or in the program section (choose initial block, stop block, etc.).

## Simulation options.

**Radius compensation.**

Activate or cancel tool radius compensation to simulate the program.

**Block skip.**

Option to simulate the external "block skip" switch. Being this option active, the CNC does not simulate the blocks containing the block-skip character "/".

**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.**

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).

**Software limits.**

Activate or deactivate the software limits for program simulation.









**Conditional stop during simulation.**

Option to simulate the external "conditional stop" switch. Being this option active, the CNC interrupts the simulation in the blocks where the "M01" function is programmed.

**Cancel channel synchronization.**

There is one icon per channel that cancels the wait for synchronism with the channel during simulation.

## User tables

Softkey	Meaning
	Description of the softkeys from the user tables.
	Change the units for data display (mm or inches). For programming, the CNC assumes the units defined with the active function G70 or G71, or, when not programmed, the units set by the machine manufacturer (INCHES parameter).
	Initialize the table. Reset all the table data to "0".
	Search a text or a value in the table.
	Accessing the tables of other channels. With some tables, only the data of the active channel are displayed, this softkey is used to show the tables of the other channels. This softkey will only be available when using channels.
	Select the axes to be displayed in the tables. When using several channels, only those axes assigned to the active channel may be accessed.
	Save the values of the table into a file.
	Restore the values of the table previously saved into a file.
	Print the table in the pre-determined printer or save it as a file (prn format) at the CNC.



### Zero offset tables.

This table contains the absolute zero offsets, the incremental zero offset (G158), and the PLC offset of all the axes and spindles that may be activated as C axis. The zero offsets associated with the possible C axes are always visible, even when the C axis is not active.

- PLC offset (PLC offset). The PLC offset may not be set directly in the table, its values are set from the PLC or via part-program using variables. The CNC always adds the PLC offset to the selected zero offset.
- The incremental zero offset is defined and activated from the part-program by executing the command G158.
- Absolute zero offsets besides being set directly in the table may also be set from the PLC or via part-program using variables.

The zero offsets are used to place the part zero at different positions of the machine. To apply an absolute zero offset, it must be activated via program using the relevant function.

### Fixture table.

This table stores the clamp offsets for each axis.

The active fixture offset is added to the zero offset (absolute + incremental) and the PLC offset.

The fixture offset besides being set directly in the table may also be set from the PLC or via part-program using variables.

The clamp offsets are used to set the position of the clamping system of the machine. To apply a clamp offset it must be activated from the program using the variable (V.)[ch].G.FIX.









### Arithmetic parameter tables.

There are the following arithmetic parameter tables:

- Common parameters. The table is common to all the channels.
- Global parameters. There is a table for each channel.
- Local parameters. There are seven tables for each channel, one table per nesting level (7 levels).

The parameter values may be set directly in the table or from the PLC or via part-program. In this case, the table values are updated after carrying out the operations indicated in the block being executed.

## Utilities mode

Softkey	Meaning
	Description of the softkeys of the utilities mode.
Cut 	Cut the selected files onto the clipboard. With this option, when pasting the files to their new location, they are erased from the current folder.
Copy 	Copy the selected files onto the clipboard.
Paste 	Paste the files from the clipboard into the selected folder. If the files were placed using the "Cut" option, they will be removed from their original location.
Rename 	Rename the selected folder or file.
Modifia... 	Change the "modifiable" attribute of the selected files. The attributes column shows the letter -M- indicating that the program may be modified. This attribute is used to protect the files so they can't be modified from the EDISIMU mode.
Hidden 	Change the "hidden" attribute of the selected files. The attributes column shows the letter -H- indicating that the program will be hidden (not visible). This attribute allows protecting the files so they are not displayed when selecting a program to be edited or executed.
	Encrypt files. Encrypting may be used to protect any file (part-program, subroutine, etc.) making it illegible so it cannot be used by anyone else. A general password to be set for encrypting files.
Remove 	Delete the selected folder or files. The folders can only be deleted if they are empty.



## Programming commands

Command	Meaning	Format
<b>/</b>	Block skip condition.	
<b>#</b>	Programming instructions.	
<b>\$</b>	Flow controlling instructions.	
<b>%</b>	Program header.	14 characters. (1)
<b>%Ln</b>	Definition of the local subroutine (included in the program).	
<b>%XX</b>	Beginning of program with local subroutines.	
<b>;</b>	Block comment.	
<b>(</b>	Block comment.	
<b>)</b>	Block comment. The piece found between "(" and ")".	
<b>[ ]</b>	Text type block label.	14 characters.
<b>N</b>	Number type block label.	0 - 4294967295
<b>G</b>	Preparatory functions.	1 - 999
<b>X, Y, Z, A, B, C, U, V, W</b>	Position of the axes. If XvalueI or Xn=valueI, the programmed position is incremental.	±99999.9999 mm ±9999.99999 inches
<b>Xn=, Yn=, Zn=, An=, Bn=, Cn=, Un=, Vn=, Wn=</b>	Position of the axes, where n=1 to 9 and is part of the axis identifier.	
<b>R, Q</b>	Position of the axis for the polar coordinates. R=Radius, Q=Degrees. If RvalueI and QvalueI, it is programmed incrementally.	
<b>F</b>	Feedrate of the axes.	
<b>S</b>	Spindle speed. Sn= Spindle speed. Sn (n=1 to 9).	
<b>T</b>	Tool number.	0 - 4294967295
<b>D</b>	Tool offset number.	
<b>M</b>	Auxiliary functions.	0 - 65535
<b>H</b>	Auxiliary functions.	0 - 65535
<b>NR</b>	Number of block repetitions.	

(1) The format admits upper and lower case letters and numbers (no blank spaces).

## Technological functions

Function	Meaning
<b>F</b>	<p><b>Machining feedrate.</b></p> <p>The machining feedrate may be selected by programmed using the "F" code which remains active until another value is programmed. The programming units depend on the active work mode (G93, G94 or G95) and the type of axis being moved (linear or rotary).</p>
<b>S</b>	<p><b>Spindle speed.</b></p> <p>The spindle speed is selected by program using the spindle name followed by the desired speed. The speeds of all the spindles of the channel may be programmed in the same block. The programmed speed stays active until another value is programmed. The programming units will be rpm unless selected otherwise. If G96 is active, the programming units will be m/min.</p>
<b>T</b>	<p><b>Tool number.</b></p> <p>The "T" code identifies the tool to be selected. The tools may be in a magazine managed by the CNC or in a manual magazine (referred to as ground tools).</p>
<b>D</b>	<p><b>Tool offset number.</b></p> <p>The tool offset contains the tool dimensions. Each tool may have several offsets associated with it. To activate an offset, it must be previously defined. To do that, the CNC offers a section of the tool table where several offsets may be defined.</p>
<b>[..]</b>	<p><b>Optional parameters.</b></p> <p>The parameters indicated between angular brackets are optional.</p>

## M functions

Function	Meaning
<b>M00</b>	Program stop.
<b>M01</b>	Conditional program stop.
<b>M02 / M30</b>	End of program.
<b>M03</b>	Start the spindle clockwise. (1)
<b>M04</b>	Start the spindle counterclockwise. (1)
<b>M05</b>	Stop the spindle. (1)
<b>M06</b>	Tool change.
<b>M08</b>	Activate coolant.
<b>M09</b>	Deactivate coolant.
<b>M17 / M29</b>	End of a global or local subroutine.
<b>M19</b>	Spindle orientation. (2)
<b>M41 - M44</b>	Gear change. (3)

(1) For spindles  $Sn$  ( $n=1$  to 9),  $M3.Sn$ ,  $M4.Sn$  and  $M5.Sn$  are programmed.

(2) For spindles  $Sn$  ( $n=1$  to 9),  $M9.Sn$  is programmed.  $Sn=$ Value in degrees.

(3) For spindles  $Sn$  ( $n=1$  to 9),  $M41.Sn$ , ...  $M44.Sn$ .

## Common G Functions (M and T models)

- M** Modal function. Those cases indicated with "!", mean the function remains active even after an M02, M30 or a reset and after the CNC is powered off and back on.
- D** Default function. Those cases indicated with "?" mean that the default quality of the function depends on the settings of the CNC machine parameters.
- V** This function is displayed in the G-code history.

Function	M	D	V	Meaning
<b>G00</b>	*	?	*	Rapid positioning.
<b>G00 X...[I] Y...[I] Z...[I] ... R...[I] Q...[I]</b> <b>X...Y...Z...A...B...C...U...V...W...:</b> cartesian coordinates <b>R...:</b> polar coordinates (Radius) <b>Q...:</b> polar coordinates (Angle) <b>I:</b> incremental				
<b>G01</b>	*	?	*	Linear interpolation.
<b>G01 X...[I] Y...[I] Z...[I] ... R...[I] Q...[I]</b> <b>X...Y...Z...A...B...C...U...V...W...:</b> cartesian coordinates <b>R...:</b> polar coordinates (Radius) <b>Q...:</b> polar coordinates (Angle) <b>I:</b> incremental				



Function	M	D	V	Meaning
<b>G02</b>	*		*	Clockwise circular (helical) interpolation.
See: G06 G261 G262	<b>G02 X...[I] Y...[I] I...J...[Z...] (G17)</b> <b>G02 X...[I] Z...[I] I...K...[Y...] (G18)</b> <b>G02 Y...[I] Z...[I] J...K...[X...] (G19)</b> <b>G02 X...[I] Y...[I] R...</b> <b>X...Y...Z...:</b> coordinates of the end point of the interpolation <b>I...J...K...:</b> coordinates of the arc center (I=X, J=Y, K=Z) <b>R...:</b> radius (alternative to I, J, K) <b>I:</b> incremental <i>Note: I, J, K are programmed using function G06 or after activating function G261.</i>			
<b>G03</b>	*		*	Counterclockwise circular (helical) interpolation.
See: G06 G261 G262	<b>G03 X...[I] Y...[I] I...J...[Z...] (G17)</b> <b>G03 X...[I] Z...[I] I...K...[Y...] (G18)</b> <b>G03 Y...[I] Z...[I] J...K...[X...] (G19)</b> <b>G03 X...[I] Y...[I] R...</b> <b>X...Y...Z...:</b> coordinates of the end point of the interpolation <b>I...J...K...:</b> coordinates of the arc center (I=X, J=Y, K=Z) <b>R...:</b> radius (alternative to I, J, K) <b>I:</b> incremental <i>Note: I, J, K are programmed using function G06 or after activating function G261.</i>			
<b>G04</b>			*	Dwell.
See: #TIME	<b>G04 K...</b> <b>K...:</b> dwell measured in seconds			
<b>G05</b>	*	?	*	Controlled corner rounding (modal).
See: G61 #ROUNDPAR	<b>G05</b>			

Function	M	D	V	Meaning
<b>G06</b>			*	Arc center in absolute coordinates (not modal).
See: <b>G261</b> <b>G262</b>	<b>G06 G02 ...</b> <b>G06 G03 ...</b>			
<b>G07</b>	*	?	*	Square corner (modal).
See: <b>G60</b>	<b>G07</b>			
<b>G08</b>			*	Arc tangent to previous path.
	<b>G08 X...Y...[R...Q...]</b> <b>X...Y...:</b> cartesian coordinates of the end point of the arc <b>R...Q...:</b> polar coordinates of the final point of the arc			
<b>G09</b>			*	Arc defined by three points.
	<b>G09 X...Y...[R...Q...] I...J...</b> <b>X...Y...:</b> cartesian coordinates of the end point of the arc <b>R...Q...:</b> polar coordinates of the final point of the arc <b>I...J...:</b> coordinates of the intermediate point of the arc (for both cartesian points and polar points)			
<b>G10</b>	*	*		Cancel mirror image on all the axes.
	<b>G10</b>			
<b>G11</b>	*		*	Mirror image on the abscissa axis.
	<b>G11</b> <i>Note: if it has been programmed along with another mirror image, for example: G11 G12: mirror image on X and Y.</i>			
<b>G12</b>	*		*	Mirror image on the ordinate axis.
	<b>G12</b> <i>Note: if it has been programmed along with another mirror image, for example: G11 G12: mirror image on X and Y.</i>			

Function	M	D	V	Meaning
<b>G13</b>	*		*	Mirror image on the axis perpendicular to the plane.

**G13**

*Note: if it has been programmed along with another mirror image, for example: G11 G13: mirror image on X and Z.*

<b>G14</b>	*		*	Activate or cancel mirror image on a axis.
------------	---	--	---	--

**G14 X+/-1 Y+/-1 ..etc...**

Axis with negative value: enable mirror image  
 Axis with positive value: disable mirror image

<b>G17</b>	*	?	*	Main plane formed by the first axis (abscissa), second (ordinate) and third axis (perpendicular) of the channel.
------------	---	---	---	--

**G17**

<b>G18</b>	*	?	*	Main plane formed by the third axis (abscissa), first axis (ordinate) and second axis (perpendicular) of the channel.
------------	---	---	---	---

**G18**

<b>G19</b>	*		*	Main plane formed by the second axis (abscissa), third axis (ordinate) and first axis (perpendicular) of the channel.
------------	---	--	---	---

**G19**

<b>G20</b>	*		*	Select any work plane formed by the first three axes of the channel.
------------	---	--	---	--

See:

#SET AX

**G20 Axis1 1 Axis2 2 Axis3 3 ...****Axis1 1:** axis defined as abscissa**Axis2 2:** axis defined as ordinates**Eje3 3:** perpendicular axis

Etc...

Example: G20 X1 Z2 Y3

X abscissa; Z ordinates; Y perpendicular.

Function	M	D	V	Meaning
<b>G30</b>			*	Polar origin preset. <b>G30 I...J...</b> <b>I</b> : polar origin abscissa <b>J</b> : polar origin ordinate
<b>G31</b>			*	Temporary polar origin shift to the center of arc. <b>G31</b>
<b>G33</b>	*		*	Electronic threading with constant pitch. <b>G33 X...Y...Z...I...J...K...[Q1=...]</b> <b>X...Y...Z...</b> : coordinate of the end point of the thread <b>I...J...K...</b> : thread pitch, respectively for abscissa, ordinates and perpendicular plane <b>Q1=...</b> : Spindle angular position for the starting point of the thread (default Q1=0)
<b>G36</b>			*	Automatic radius blend. <b>G36 I...</b> <b>I...</b> : value of the rounding radius in the corner
<b>G37</b>			*	Tangential entry. <b>G37 I...</b> <b>I...</b> : value of the entry radius
<b>G38</b>			*	Tangential exit. <b>G38 I...</b> <b>I...</b> : value of the exit radius
<b>G39</b>			*	Automatic chamfer blend. <b>G39 I...</b> <b>I...</b> : value of the chamfer size
<b>G40</b>	*	*		Cancel tool radius compensation. <b>G40</b>
<b>G41</b>	*		*	Left-hand tool radius compensation. <b>G41</b>

Function	M	D	V	Meaning
----------	---	---	---	---------

<b>G42</b>	*		*	Right-hand tool radius compensation.
------------	---	--	---	--------------------------------------

**G42**

<b>G45</b>				Turn tangential control on and off.
------------	--	--	--	-------------------------------------

See:

#TANGCTRL

**G45 X...Y...Z...A...B...C...U...V...W...****X...~W...:** axis upon which tangential control is applied and angular position with respect to the tool path.The angle is defined in degrees ( $\pm 359.9999$ ).

<b>G50</b>	*	?		Semi-rounded corner.
------------	---	---	--	----------------------

**G50**

<b>G53</b>	*			Cancel zero offset.
------------	---	--	--	---------------------

See:

#MCS

**G53**

<b>G54</b>	!		*	Zero offset.
------------	---	--	---	--------------

See:

**G159****G54:** absolute zero offset 1

<b>G55</b>	!		*	Zero offset.
------------	---	--	---	--------------

**G55:** absolute zero offset 2

<b>G56</b>	!		*	Zero offset.
------------	---	--	---	--------------

**G56:** absolute zero offset 3

<b>G57</b>	!		*	Zero offset.
------------	---	--	---	--------------

**G57:** absolute zero offset 4

<b>G58</b>	!		*	Zero offset.
------------	---	--	---	--------------

**G58:** absolute zero offset 5

<b>G59</b>	!		*	Zero offset.
------------	---	--	---	--------------

**G59:** absolute zero offset 6

<b>G60</b>			*	Square corner (not modal).
------------	--	--	---	----------------------------

See:

**G07****G60**

Function	M	D	V	Meaning
<b>G61</b>			*	Controlled corner rounding (not modal).
See: <b>G05</b> <b>#ROUNDPAR</b>	<b>G61</b>			
<b>G63</b>	*		*	Rigid tapping.
	<b>G63 X...Y...Z...S...</b> <b>X...Y...Z...:</b> coordinate of the end point of the thread <b>S...:</b> turning speed (positive or negative) <i>Note: the thread pitch will be F/S.</i>			
<b>G70</b>	*	?	*	Programming in inches.
	<b>G70</b>			
<b>G71</b>	*	?		Programming in millimeters.
	<b>G71</b>			
<b>G72</b>	*		*	Scaling factor.
See: <b>#SCALE</b>	<b>G72 S...</b> <b>S...: scaling factor</b> <i>Note: it is annulled by only programming G72.</i>			
<b>G73</b>	*		*	Rotation of the coordinate system.
	<b>G73 Q...I...J...</b> <b>Q...: rotation angle (incremental)</b> <b>I...J...: abscissa and ordinate of the rotational center (optional)</b> <i>Note: it is annulled by only programming G73.</i>			
<b>G74</b>			*	Machine reference zero (home) search.
	<b>G74 Axis1 1 Axis2 2 Axis3 3 ...</b> <b>Axis1 1 ~ Axis n:</b> machine reference (home) order Example: G74 X1 Y2 Z3 ...: first look for the zero for X, then for Y and then for Z. <i>Note: the active zero offset is maintained.</i>			
<b>G90</b>	*	?		Programming in absolute coordinates.
	<b>G90</b>			

Function	M	D	V	Meaning
<b>G91</b>	*	?	*	Programming in incremental coordinates.
<b>G91</b> <i>Note: G91 affects all the coordinates in the block. XvalueI affects the non-modal programmed axis and it is non-modal.</i>				
<b>G92</b>	!		*	Zero offset. Coordinate preset.
<b>G92 X...Y...Z...A...B...C...U...V...W...</b> <b>X... ~ W ...:</b> zeroing of the axis in the position where the machine is located (the programmed coordinate may be 0 or another value)				
<b>G93</b>	*		*	Feedrate in seconds/block.
<b>G93 F...</b> <b>F ...:</b> time in seconds for the programmed movement				
<b>G94</b>	*	?		Feedrate in millimeters/minute (inches/minute).
<b>G94 F...</b> <b>F...:</b> feedrate in millimeters/minute or inches/minute.				
<b>G95</b>	*	?	*	Feedrate in millimeters/revolution (inches/revolution).
<b>G95 F...</b> <b>F...:</b> feedrate in millimeters/revolution or inches/revolution.				
<b>G96</b>	*		*	Constant surface speed.
<b>G96 S...</b> <b>S...:</b> constant surface speed (CSS) in meters/minute				
<b>G97</b>	*	*		Constant turning speed.
<b>G97 S...</b> <b>S...:</b> constant surface speed of spindle in in revolutions/minute.				

Function	M	D	V	Meaning
<b>G100</b>			*	Probing until making contact. <b>G100 Axis1... axis2... ... Axis n... F...</b> <b>axis1... ~ Axis n:</b> position value of probing point <b>F...:</b> movement speed (optional)
<b>G101</b>	*			Include probe offset. <b>G101 Axis1... axis2... ... Axis n...</b> <b>axis1... ~ Axis n:</b> axis whose theoretical position value is included in the resulting offset of the measurement
<b>G102</b>	*			Exclude probe offset. <b>G102 Axis1... axis2... ... Axis n...</b> <b>axis1... ~ Axis n:</b> optional, axis whose theoretical position value is not included in the resulting offset of the measurement
<b>G103</b>			*	Probing until not making contact. <b>G103 Axis1... axis2... ... Axis n... F...</b> <b>axis1... ~ Axis n:</b> position value of probing point <b>F...:</b> movement speed (optional)
<b>G104</b>				Probe movement up to the programmed position. <b>G104</b> <i>Note: Function G104 must be programmed together with a G100 or G103 probe movement; otherwise, it will be ignored.</i> Examples: G100 G104 Z23.45 G103 G104 Z1 F20
<b>G108</b>	*	*		Blend the feedrate at the beginning of the block. <b>G108</b> See: <b>G193</b>
<b>G109</b>			*	Blend the feedrate at the end of the block. <b>G109</b> See: <b>G193</b>



Function	M	D	V	Meaning
<b>G112</b>	*			Changing of parameter range of an axis. <b>G112 X1...4 Y1...4 Z1...4 S1...4</b> <b>X1...4~W1...4</b> : name of the axis and parameter set (between 1 and 4)
<b>G130</b>	*		*	Percentage of acceleration to be applied per axis or spindle.  See: <b>#SLOPE</b> <b>G130 X...Y...Z...A...B...C...U...V...W...S...</b> <b>X...~W...</b> : acceleration percentage to be applied per axis or spindle (S for spindle) Example: G130 X50 Y75 Z100 X acceleration at 50%, Y at 75% and Z at 100%
<b>G131</b>	*		*	Percentage of acceleration to be applied, global.  See: <b>#SLOPE</b> <b>G131 ...</b> <b>...:</b> acceleration percentage to be applied for all axes and spindle Example: G131 100 all axes and spindles at 100% of the selected acceleration
<b>G132</b>	*		*	Percentage of jerk to be applied per axis or spindle.  See: <b>#SLOPE</b> <b>G132 X...Y...Z...A...B...C...U...V...W...S...</b> <b>X...~W...</b> :jerk percentage to be applied per axis or spindle (S for spindle) Example: G132 X50 Y75 Z100 X jerk at 50%, Y at 75% and Z at 100%
<b>G133</b>	*		*	Percentage of jerk to be applied, global.  See: <b>#SLOPE</b> <b>G133 ...</b> <b>...:</b> jerk percentage to be applied for all axes and spindle Example: G133 100 all axes and spindles at 100% of the selected jerk

Function	M	D	V	Meaning
<b>G134</b>	*		*	Percentage of feed-forward to be applied. <b>G134 X...Y...Z...A...B...C...U...V...W...</b> <b>X... ~ W...:</b> feed-forward percentage to be applied per axis (maximum 120%) Example: G134 X55 Y85 Z120 X feed-forward at 55%, Y at 85% and Z at 120%
<b>G135</b>	*		*	Percentage of AC-Forward to be applied. <b>G135 X...Y...Z...A...B...C...U...V...W...</b> <b>X... ~ W...:</b> AC-forward percentage to be applied per axis (maximum 120%) Example: G135 X50 Y75 Z110 X AC-forward at 50%, Y at 75% and Z at 110%
<b>G136</b>	*		*	Circular transition between blocks. <b>G136</b>
<b>G137</b>	*	*		Linear transition between blocks. <b>G137</b>
<b>G138</b>	*		*	Direct activation/cancellation of tool compensation. <b>G138</b>
<b>G139</b>	*	*		Indirect activation/cancellation of tool compensation. <b>G139</b>
<b>G145</b>				Freeze tangential control. See: <b>G145 K... X ~ W</b> <b>#TANGCTRL K...:</b> K0 Freeze (suspend) tangential control. K1 restores previously frozen tangential axis <b>X... ~ W...:</b> axis where tangential control is frozen (optional).

Function	M	D	V	Meaning
<b>G151</b>	*	*	*	Programming the face axis in diameter.
<b>G151</b> <i>Note: with axes configured using machine parameter DIAMPROG=YES.</i>				
<b>G152</b>	*			Programming the face axis in radius.
<b>G152</b> <i>Note: with axes configured using machine parameter DIAMPROG=YES.</i>				
<b>G157</b>	*		*	Excluding axes in the absolute zero offset.
See: <b>G158</b>	<b>G157 X ~ W</b> <b>X ~ W:</b> axes where the zero offset is not applicable Example: G157 Y G55 (the second zero offset is applicable, except on the Y axis, which retains the previous movement)			
<b>G158</b>	*		*	Zero offset. Incremental zero offset.
<b>G158 X...Y...Z...A...B...C...U...V...W...</b> <b>X... ~ W...:</b> value of zero offset. It is annulled by only programming G158.				
<b>G159</b>	!		*	Zero offset. Absolute zero offset.
See: <b>G54-G59</b>	<b>G159=...</b> <b>...:</b> number of the zero offset to be activated (from 1 to 99)			
<b>G170</b>	*			Cancel Hirth axes.
<b>G170 A...B...C...</b> <b>A...B...C...:</b> Hirth axis to deactivate and deactivation order number Example: G170 A1 B2: first deactivate axis A and then axis B				

Function	M	D	V	Meaning
<b>G171</b>	*	*		Activate Hirth axes. <b>G171 A...B...C...</b> <b>A...B...C...:</b> Hirth axis to activate and activation order number Example: G171 C1 B2: first activate axis C and then axis B
<b>G174</b>	*			Set the machine coordinate. <b>G174 X...Y...Z...A...B...C...U...V...W...S...</b> <b>X... ~ W...:</b> set the machine coordinate for the axis (the position is programmed for a single axis) <b>S...:</b> set the machine coordinate for the spindle
<b>G180</b> <b>G189</b>			*	Execution of OEM subroutines. See: <b>G380-G399</b> <b>G180 ~ G189 P0=... P1=... ~ P99=...</b> <b>G180 ~ G189 A... B... ~ Z...</b> <b>P0=... ~ Pn=...:</b> local parameters of the subroutine, optional Example: G183 P1=12.3 P2=6 <i>Note: these are functions that the manufacturer can use for its specific functions; it is possible that they are not present.</i>
<b>G192</b>	*		*	Turning speed limitation. <b>G192 Sn=...</b> <b>G192 S...</b> For multiple spindles, n=spindle number. The limit is programmed in rpm.
<b>G193</b>			*	Interpolate feedrate during the block. See: <b>G108</b> <b>G109</b> <b>G193</b> <i>Note: when programming function G193, the adaptation to the new feedrate is interpolated linearly during the movement programmed in the block.</i>

Function	M	D	V	Meaning
----------	---	---	---	---------

<b>G196</b>	*		*	Constant tangential feedrate.
-------------	---	--	---	-------------------------------

**G196**

*Note: after executing function G196, the CNC interprets that the programmed "F" corresponds to the contact point between the tool and the part.*

<b>G197</b>	*	*		Constant feedrate of the tool center.
-------------	---	---	--	---------------------------------------

**G197**

*Note: after executing function G197, the CNC interprets that the programmed "F" corresponds to the tool center. This means that the feedrate at the cutting point increases on inside arcs and decreases on outside arcs.*

<b>G198</b>				Set lower software travel limits.
-------------	--	--	--	-----------------------------------

**G198 X... ~ W...**

**X... ~ W...:** lower software limit for the movement of the axes

<b>G199</b>				Set upper software travel limits.
-------------	--	--	--	-----------------------------------

**G199 X... ~ W...**

**X... ~ W...:** upper software limit for the movement of the axes

<b>G200</b>				Exclusive manual intervention.
-------------	--	--	--	--------------------------------

**G200**

*Note: interrupts the program execution to activate the manual mode for all axes.*

<b>G201</b>	*			Activate additive manual intervention.
-------------	---	--	--	--

**G201 #AXIS [X ~ W, ... ]**

Example: G201 #AXIS [X, Z]

*Note: jogging can be performed while the programmed movements are executed.*

Function	M	D	V	Meaning
<b>G202</b>	*	*		Cancellation of additive manual intervention.

**G202 #AXIS [X ~ W, ... ]**

Example: G202 #AXIS [X]

*Note: cancels the additive manual intervention of the selected axis; if only the function G202 is programmed it will cancel the manual intervention for all axes.*

<b>G261</b>	*		*	Arc center in absolute coordinates (modal).
-------------	---	--	---	---

See:

**G06**

**G262**

**G261**

*Note: when this function is active, the CNC interprets that the coordinates of the arc center are defined by the origin of active reference system.*

<b>G262</b>	*	*		Arc center referred to starting point.
-------------	---	---	--	--

See:

**G06**

**G261**

**G262**

*Note: when this function is active, the CNC interprets that the coordinates of the arc center are referred to the starting point of the arc.*

<b>G263</b>	*		*	Arc radius programming.
-------------	---	--	---	-------------------------

See:

**G02/G03**

**G263=...**

...: Arc radius

*Note: the arc radius is defined with the letter "R" or by using assignments "R1=<radius>" or "G263=<radius>".*

<b>G264</b>	*		*	Cancel arc center correction.
-------------	---	--	---	-------------------------------

**G264**

*Note: if the difference between the initial radius and the final radius exceeds the tolerance, then the corresponding error will be displayed.*

<b>G265</b>	*	*		Activate arc center correction.
-------------	---	---	--	---------------------------------

**G265**

*Note: the arc center is recalculated to make the initial radius and the final radius the same.*

Function	M	D	V	Meaning
----------	---	---	---	---------

<b>G266</b>			*	Set feedrate percentage at 100%.
-------------	--	--	---	----------------------------------

**G266**

*Note: this sets the feedrate override at 100%, which cannot be changed by selector switch on the operator panel or via PLC. G66 is only valid for the block in which it is programmed.*

<b>G380 G399</b>			*	OEM subroutine execution.
----------------------	--	--	---	---------------------------

See:

G180-G189

**G380 ~ G399 P0=... P1=... ~ P99=...****G380 ~ G399 A... B... ~ Z...**

**P0=... ~ Pn=...:** local parameters of the subroutine, optional

Example: G388 A12.3 B45.3 P10=6

*Note: these are functions that the manufacturer can use for its specific functions; it is possible that they are not present.*

<b>G500 G599</b>			*	Generic user subroutines.
----------------------	--	--	---	---------------------------

**G500 ~ G599 P0=... P1=... ~ P99=...****G500 ~ G599 A... B... ~ Z...**

**P0=... ~ Pn=...:** local parameters of the subroutine, optional

Example: G500

G583 P1=12.3 P2=6

G588 A12.3 B45.3 P10=6

*Note: when calling the subroutine associated with G... the subroutine is executed non-modally. When calling it with MG ... it is executed modally.*





# ISO CANNED CYCLES (M)

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

[G functions]	G8x	[Machining point]	Parameters of the cycle	[F S T D M]
---------------	-----	-------------------	-------------------------	-------------

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

```
G99 G1 G81 X60 Y0 Z2 I-20 F1000 S2000 M4
G99 G1 X60 Y0 F1000 S2000 M4 G81 Z2 I-2
```

Function	M	D	V	Meaning
<b>G80</b>	*			Cancel the canned cycle.

**G80**

<b>G81</b>	*		*	Drilling canned cycle.
------------	---	--	---	------------------------

## **G81 [Z] I [K] [A]**

Z: Reference plane

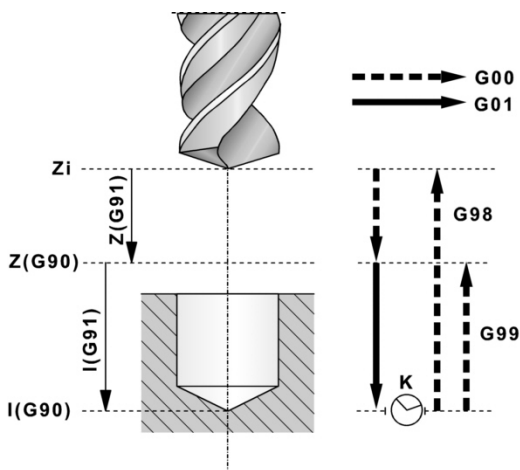
I: Drilling depth

K: Dwell in seconds at the bottom of the hole

A: Spindle behavior when entering and exiting the hole.

A0: The tool enters the hole and exits while turning (default value)

A1: The tool goes into the hole turning and comes out stopped



Function M D V Meaning

**G82**

\* \* \* Drilling canned cycle with a variable peck.

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

Z: Reference plane

I: Drilling depth

D: Distance between the reference plane and the part surface (default = 0)

B: Drilling peck (step)

H: Rapid feedrate withdrawal (J>0) or withdrawal coordinate (J=0) distance after each drilling peck

C: Approach coordinate

J: Number of drilling steps for rapid feedrate withdrawal (G0). J0: it returns to the coordinate value H after each step; J1: it returns to the coordinate value Z; J>1: after each step, it returns the distance indicated by H and every J step to the reference plane Z.

K: Dwell in seconds at the bottom of the hole

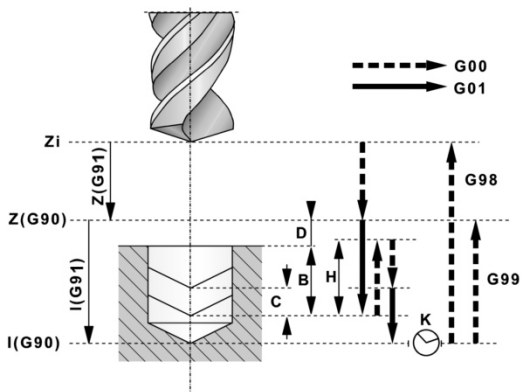
R: Factor that increases or reduces the drilling peck (step) B

L: Minimum value allowed for the drilling peck

A: Spindle behavior when entering and exiting the hole.

A0: The tool enters the hole and exits while turning (default value)

A1: The tool goes into the hole turning and comes out stopped



# ISO canned cycles (M)

Function M D V Meaning

**G83**

\*

\*

Deep-hole drilling canned cycle with constant peck.

**G83 [Z] I J [B] [K]**

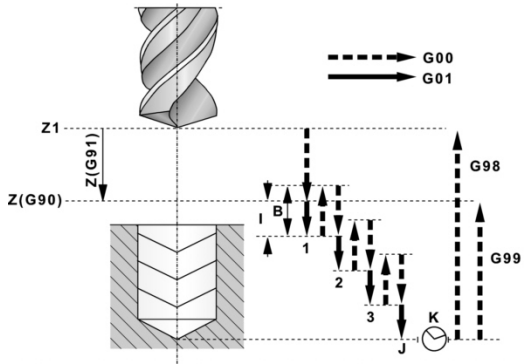
Z: Reference plane

I: Drilling peck (step)

J: Number of drilling pecks

B: Rapid withdraw (G0) distance after each drilling step

K: Dwell in seconds at the bottom of the hole



Function	M	D	V	Meaning
----------	---	---	---	---------

<b>G84</b>	*		*	Tapping canned cycle.
------------	---	--	---	-----------------------

**G84 [Z] I [K] R [J] B H**

Z: Reference plane

I: Tap depth

K: Dwell in seconds at the bottom of the hole (default =0)

R: Type of threading

R0: Normal tapping (only for spindles that don't have the spindle positioning option).

R1: Rigid tapping

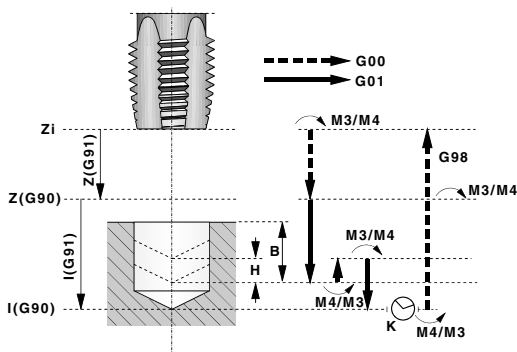
J: Withdrawal feedrate factor. When rigid tapping, the returning feedrate will be J times the tapping feedrate. When not programmed or programmed J1, they will both be the same

B: Penetration step when tapping with chip removal (only for R1)

H: Withdrawal distance after each penetration step (only for R1)

*Note: If G94 (F in mm/min) it is programmed with F and S. For S, the value of F will be:*

*F (mm/min) = S (rpm) \* Pitch (mm/rev)*



Function	M	D	V	Meaning
----------	---	---	---	---------

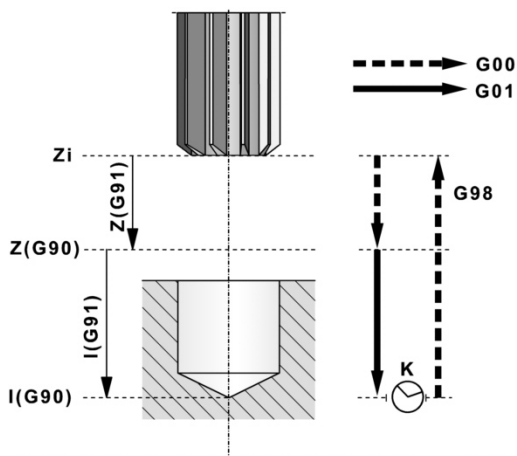
<b>G85</b>	*		*	Reaming canned cycle.
------------	---	--	---	-----------------------

**G85 [Z] I [K]**

Z: Reference plane

I: Reaming depth

K: Dwell in seconds at the bottom of the hole (default =0)



Function M D V Meaning

**G86**

\* \* Boring canned cycle.

**G86 [Z] I [K] [R] [A] Q D E**

Z: Reference plane

I: Boring depth

K: Dwell in seconds at the bottom of the hole (default =0)

R: Type of withdrawal, when parameter A is not programmed. R0: Rapid feedrate withdrawal (G0) with the spindle stopped (default value); R1: Withdraw at work feedrate (G1)

A: Spindle behavior when going into the hole and coming out of it

A0: The tool goes into the hole turning and comes out stopped

A1: The tool goes into the hole stopped and comes out turning

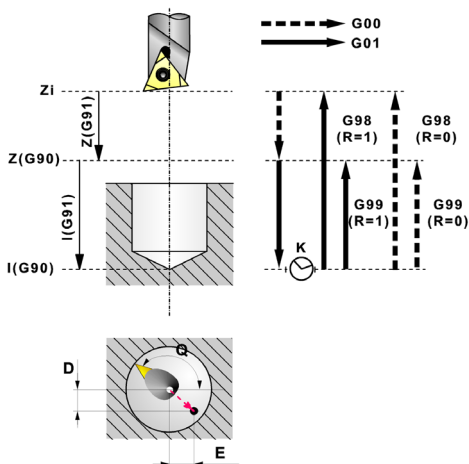
When not programmed, the tool goes into the hole and comes out turning. When programming A0 or A1, define the spindle stop with parameters Q, D and E.

Q: Spindle position, in degrees, to separate the cutter from the wall of the hole

When programming A0 or A1, this parameter sets the tool orientation and parameters D and E set the distance the tool withdraws off (away from) the walls of the hole.

D: Distance to withdraw the cutter off the wall of the hole along the abscissa axis

E: Distance to withdraw the cutter off the wall of the hole along the ordinate axis



**Function M D V Meaning**
**G87** \* \* \* Rectangular pocket canned cycle.

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

Z: Reference plane

I: Pocket depth

D: Distance between the reference plane and the part surface

A: Angle, in degrees, between the pocket and the abscissa axis (default value = 0).

J: Half length of the pocket

The sign indicates the pocket machining direction: (J+) clockwise, (J-) counterclockwise

K: Half width of the pocket

M: Corner type (0) square, (1) rounded, (2) chamfered (default value = 0).

Q: Rounding radius or chamfer size

B: Depth of pass

C: Milling pass or width

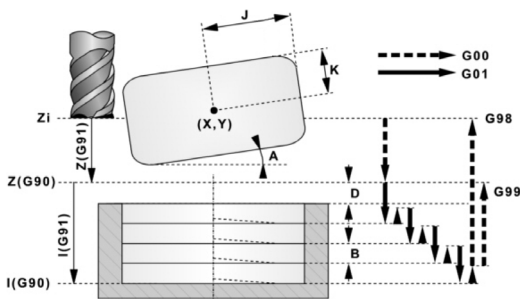
If not programmed or programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool

L: Finishing pass

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

H: Feedrate for the finishing pass

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



Function M D V Meaning

**G88** \* \* \* Circular pocket canned cycle.

**G88 [Z] I [D] J B [C] [L] [H] [V]**

Z: Reference plane

I: Pocket depth

D: Distance between the reference plane and the part surface (default = 1mm)

J: Pocket radius; the sign indicates the pocket machining direction: (J+) clockwise, (J-) counterclockwise

B: Depth of pass. If  $B > 0$ , the cycle recalculates the step so that all penetrations are the same; if  $B < 0$ , the pocket is machined with the set step, except the last step that machines the rest.

C: Milling pass or width

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

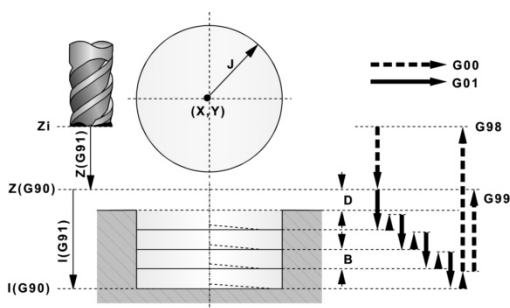
L: Finishing pass

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

H: Feedrate for the finishing pass

V: Tool penetrating feedrate

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



**G98** \* \* \* Withdrawal to the starting plane.

**G98**

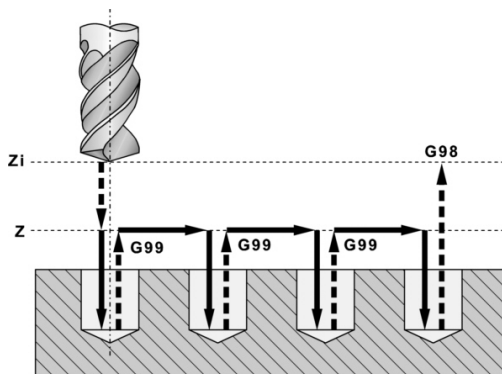


Function M D V Meaning

<b>G99</b>	*		*	Withdrawal to the reference plane at the end of the canned cycle.
------------	---	--	---	---

**G99**

*Note: use G98 to avoid obstacles (fixtures).*



Function M D V Meaning

**G210** \* \* Bore milling canned cycle.

**G210 [Z] [D] I [J] [K] B**

Z: Reference plane

If not programmed, it assumes as reference plane the current position of the tool

D: Safety distance (default = 0)

I: Machining depth

It may be programmed either in absolute or incremental coordinates, in which case it will be referred to the reference plane

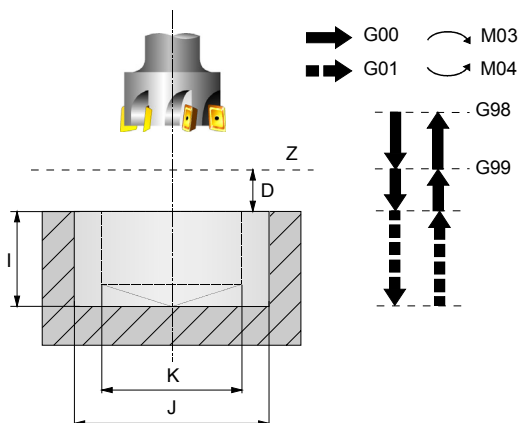
J: Hole diameter

K: Pre-drilling diameter

B: Profiles

*Note: the tool must meet the following conditions:*

- The tool radius must be smaller than  $K/2$ .
- The tool radius must be equal to or larger than  $(J-K)/4$ .



## Function M D V Meaning

**G211**

\* \* \* Inside thread milling cycle.

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

Z: Reference plane

If not programmed, it assumes as reference plane the current position of the tool

D: Safety distance (default = 0)

I: Machining depth

It may be programmed either in absolute or incremental coordinates, in which case it will be referred to the reference plane

J: Thread diameter

K: Thread depth

B: Thread pitch

C: Type of threading

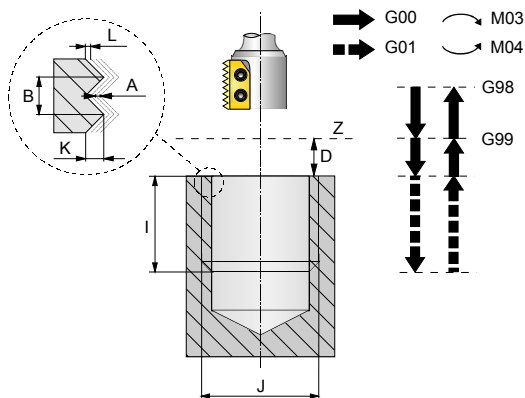
- When programming C = 0, the threading will be done in a single pass
- When programming C = 1, it will make one thread per each pass (single-edge cutter)
- When programming C = n (where n is the number of cutting edges of the cutter), it will make n threads per pass (default = 1)

L: Finishing stock allowance for finishing (default = 0).

A: Maximum penetration pass

E: Approach distance

Q: Thread entry angle (default = 0)



**Function M D V Meaning**
**G212**

\* \* **Outside thread milling cycle.**
**G212 [Z] [D] I J K B [C] [L] [A] E [Q]**
**Z:** Reference plane

If not programmed, it assumes as reference plane the current position of the tool

**D:** Safety distance (default = 0)

**I:** Machining depth

It may be programmed either in absolute or incremental coordinates, in which case it will be referred to the reference plane

**J:** Thread diameter

**K:** Thread depth

**B:** Thread pitch

**B>0:** the direction of the thread pitch is from the surface of the part to the bottom.

**B<0:** the direction of the thread pitch is from the bottom to the surface of the part.

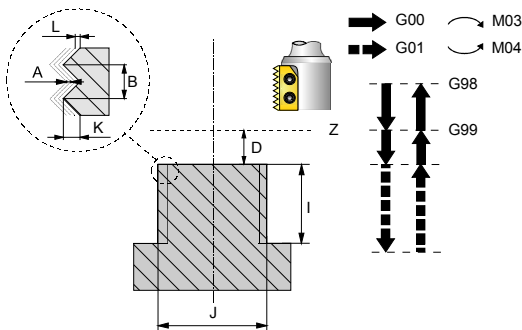
**C:** Type of threading

- When programming  $C=0$ , the threading will be done in a single pass
- When programming  $C=1$ , it will make one thread per each pass (single-edge cutter)
- When programming  $C=n$  (where  $n$  is the number of cutting edges of the cutter), it will make  $n$  threads per pass (default = 1)

**L:** Finishing stock allowance for finishing (default = 0).

**A:** Maximum penetration pass

**E:** Approach distance

**Q:** Thread entry angle (default = 0)


## Multiple machining (M model)

Parameters P, Q, R, S, T, U and V are optional parameters that may be used in any type of multiple positioning. Thus, programming "P7" means that no machining operation takes place at point 7. Programming "Q10.013" means that no machining takes place at points 10, 11, 12 and 13. If these parameters are not programmed, the CNC understands that it must perform machining at all the points along the programmed path.

Function	M	D	V	Meaning
----------	---	---	---	---------

<b>G160</b>			*	Multiple machining in a straight line.
-------------	--	--	---	--

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

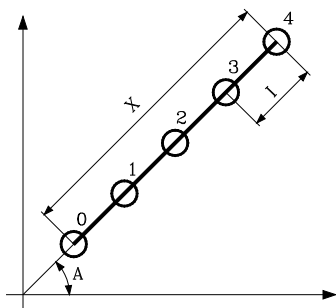
A: Angle, in degrees of the tool path with respect to the abscissa axis (default = 0).

X: Length of the machining path

I: Step between machining operations

K: Total number of machining operations in the section, including that of the machining definition point

*Note: when defining the machining operation, only two of parameters X, I and K are required.*



Function M D V Meaning

**G161**

\* Multiple machining in rectangular pattern.

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

A: Angle, in degrees of the tool path with respect to the abscissa axis (default = 0).

B: Angle between both tool paths (default value = 90)

X: Length of the parallelogram

I: Step between machining operations along the path

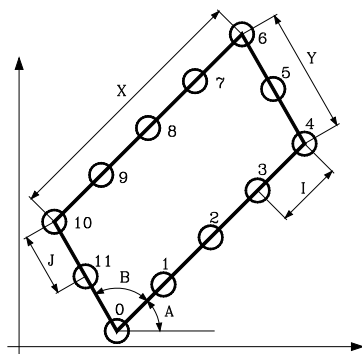
K: Total number of machining operations along the path, including that of the machining definition point

Y: Width of the parallelogram

J: Step between machining operations along the path

D: Total number of machining operations along the path, including that of the machining definition point

*Note: when defining the machining operation, only two of parameters X, I and K are required. When defining the machining operation, only two of parameters X, J and D are required.*



## Function M D V Meaning

**G162**

\* Multiple machining in a grid pattern.

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

A: Angle, in degrees of the tool path with respect to the abscissa axis (default = 0).

B: Angle between both tool paths (default value = 90)

X: Length of the grid

I: Step between machining operations along the path

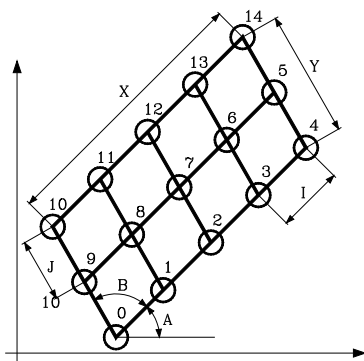
K: Total number of machining operations along the path, including that of the machining definition point

Y: Width of the grid

J: Step between machining operations along the path

D: Total number of machining operations along the path, including that of the machining definition point

*Note: when defining the machining operation, only two of parameters X, I and K are required. When defining the machining operation, only two of parameters X, J and D are required.*



Function M D V Meaning

**G163**

\* Multiple machining in a circular pattern.

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

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

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

I: Angular increment between machining operations

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

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

C: Type of movement between machining points (default value = 0)

C=0: In rapid (G00)

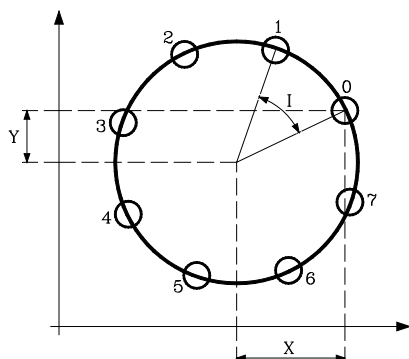
C=1: Linear interpolation (G01)

C=2: In clockwise circular interpolation (G02)

C=3: In counterclockwise circular interpolation (G03)

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

*Note: when defining the machining operation, only one of parameters I and K is required.*





**Function M D V Meaning**
**G164**

\* Multiple machining in an arc.

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

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

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

B: Angular distance in degrees of the machining path

I: Angular increment between machining operations

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

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

C: Type of movement between machining points (default value = 0)

C=0: In rapid (G00)

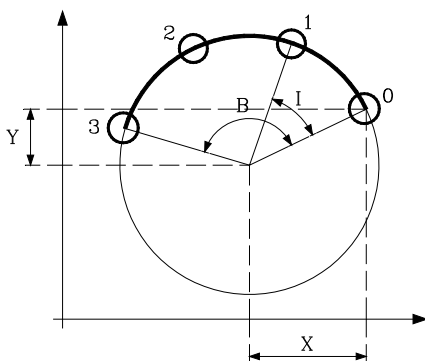
C=1: Linear interpolation (G01)

C=2: In clockwise circular interpolation (G02)

C=3: In counterclockwise circular interpolation (G03)

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

*Note: when defining the machining operation, only one of parameters I and K is required.*



Function	M	D	V	Meaning
<b>G165</b>			*	Machining programmed with an arc-chord.

### **G165 X Y A I C F**

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

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

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

I: Length of the chord

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

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

C: Type of movement between machining points (default value = 0)

C=0: In rapid (G00)

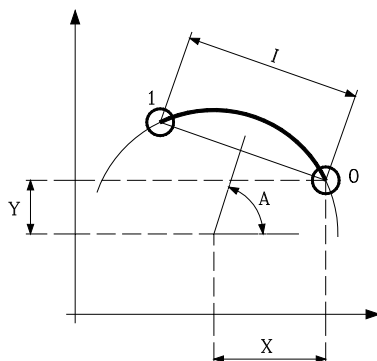
C=1: Linear interpolation (G01)

C=2: In clockwise circular interpolation (G02)

C=3: In counterclockwise circular interpolation (G03)

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

*Note: when defining the machining operation, only one of parameters A or I is required.*



# ISO CANNED CYCLES (T)

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

[G functions]	G6x	[Machining point]	Parameters of the cycle	[F S T D M]
---------------	-----	-------------------	-------------------------	-------------

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

```
G99 G1 G66 X60 Z0 I-20 F1000 S2000 M4
G99 G1 X60 Z0 F1000 S2000 M4 G68 Z2 I-2
```

Function	M	D	V	Meaning
----------	---	---	---	---------

<b>G66</b>			*	Pattern repeat canned cycle.
------------	--	--	---	------------------------------

## **G66 X Z I C A [L] [M] [J] [H] S E P Q**

X: X coordinate of the profile's starting point

Z: Z coordinate of the profile's starting point

I: Excess material (in radius)

C: Machining pass (in radius)

A: Machining main axis

A=0 main axis Z; A=1 main axis X

L: Finishing stock on the X axis (in radius)

M: Finishing stock on the Z axis (If not programmed, it will be that indicated for parameter L)

J: Maximum travel in each section on the machining axis

H: Finishing feedrate

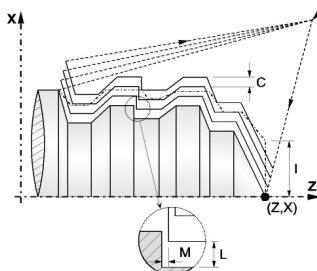
S: Label number of the first block describing the geometry of the profile

E: Label number of the last block describing the geometry of the profile

P: Number of the subroutine where that defines the profile

Q: Name of the global subroutine, that contains the definition of the profile (parameters E and S) or of the program that contains the local subroutine that contains the profile (parameter P)

See: *Optimizing the machining operation (Page 101).*



Function M D V Meaning

**G68**

\* Stock removal cycle along X axis.

**G68 X Z C [D] [L] [M] [J] [K] [F] [H] S E P Q**

X: X coordinate of the profile's starting point

Z: Z coordinate of the profile's starting point

C: Machining pass (in radius)

D: Withdrawal distance after each pass

D=0: the tool exit path is the same as the entry path. D<>0: retracts at 45 degrees; D not programmed: retracts while following the profile to the previous step

L: Finishing stock on the X axis (in radius)

M: Finishing stock on the Z axis (If not programmed, it will be that indicated for parameter L)

J: Maximum travel in each section on the machining axis

K: Penetration feedrate in the roots

F: Feedrate for the last roughing pass. If not programmed or programmed as F0, it does not run the last roughing pass.

H: Finishing feedrate. If not programmed or programmed as H0, it does not run the finishing pass.

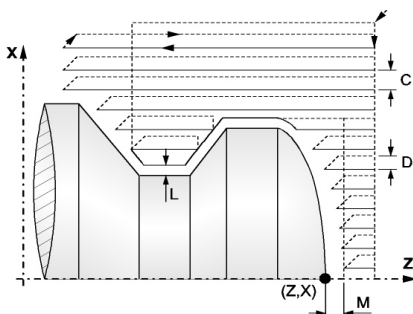
S: Label number of the first block describing the geometry of the profile

E: Label number of the last block describing the geometry of the profile

P: Number of the subroutine where that defines the profile

Q: Name of the global subroutine, that contains the definition of the profile (parameters E and S) or of the program that contains the local subroutine that contains the profile (parameter P)

See: *Optimizing the machining operation* (Page 101).



## Function M D V Meaning

<b>G69</b>			*	Stock removal canned cycle along Z axis.
------------	--	--	---	--

**G69 X Z C [D] [L] [M] [K] [F] [H] S E P Q**

X: X coordinate of the profile's starting point

Z: Z coordinate of the profile's starting point

C: Machining pass (in radius)

D: Withdrawal distance after each pass

D=0: the tool exit path is the same as the entry path

D&lt;&gt;0: retracts at 45 degrees

D not programmed: retracts while following the profile to the previous step

L: Finishing stock on the X axis (in radius)

M: Finishing stock on the Z axis (If not programmed, it will be that indicated for parameter L)

K: Penetration feedrate in the roots

F: Feedrate for the last roughing pass. If not programmed or programmed as F0, it does not run the last roughing pass.

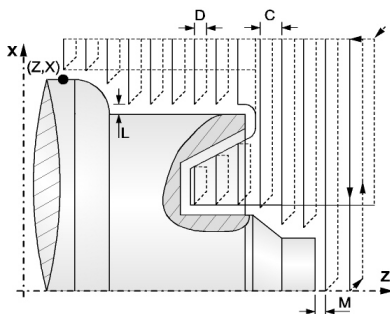
H: Finishing feedrate. If not programmed or programmed as H0, it does not run the finishing pass.

S: Label number of the first block describing the geometry of the profile

E: Label number of the last block describing the geometry of the profile

P: Number of the subroutine where that defines the profile

Q: Name of the global subroutine, that contains the definition of the profile (parameters E and S) or of the program that contains the local subroutine that contains the profile (parameter P)

*See: Optimizing the machining operation (Page 101).*

Function	M	D	V	Meaning
<b>G81</b>			*	Turning canned cycle for straight sections.

**G81 X Z Q R C D [L] [M] [F] [H]**

X: X coordinate of the profile's starting point

Z: Z coordinate of the profile's starting point

Q: X coordinate of the profile's last point

R: Z coordinate of the profile's last point

C: Machining pass (in radius)

D: Withdrawal distance after each pass

D=0: the tool exit path is the same as the entry path

D<>0: retracts at 45 degrees

D not programmed: retracts while following the profile to the previous step

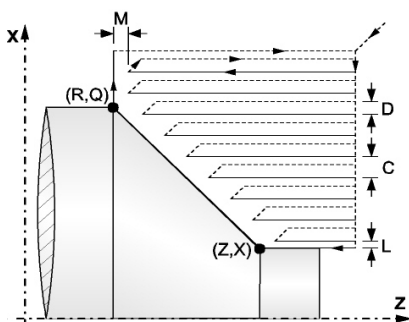
L: Finishing stock on the X axis (in radius)

M: Defines the finishing pass along the Z axis (if not programmed, a value of 0 will be assumed)

F: Feedrate for the last roughing pass. If not programmed or programmed as F0, it does not run the last roughing pass.

H: Finishing feedrate. If not programmed or programmed as H0, it does not run the finishing pass.

*Note: The radius compensation must be G41/G42 before activating G81.*



## Function M D V Meaning

<b>G82</b>			*	Facing canned cycle for straight sections.
------------	--	--	---	--

**G82 X Z Q R C D [L] [M] [F] [H]**

X: X coordinate of the profile's starting point

Z: Z coordinate of the profile's starting point

Q: X coordinate of the profile's last point

R: Z coordinate of the profile's last point

C: Machining pass (in radius)

D: Withdrawal distance after each pass

D=0: the tool exit path is the same as the entry path

D&lt;&gt;0: retracts at 45 degrees

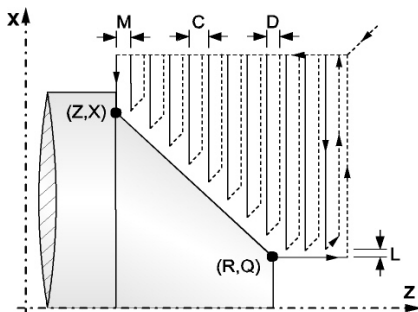
D not programmed: retracts while following the profile to the previous step

L: Finishing stock on the X axis (in radius)

M: Defines the finishing pass along the Z axis (if not programmed, a value of 0 will be assumed)

F: Feedrate for the last roughing pass. If not programmed or programmed as F0, it does not run the last roughing pass.

H: Finishing feedrate. If not programmed or programmed as H0, it does not run the finishing pass.

*Note: The radius compensation must be G41/G42 before activating G82.*

Function M D V Meaning

**G83**

\* Drilling / tapping canned cycle.

**G83 X Z I B [D] [K] [H] [C] [R] (axial drilling)**

**G83 X Z I B0 [D] [K] [R] (axial tapping)**

X: X coordinate of the starting point

Z: Z coordinate of the starting point

I: Machining depth

B: Type of machining

B=0: tapping

B>0; drilling (where B indicates the drilling peck)

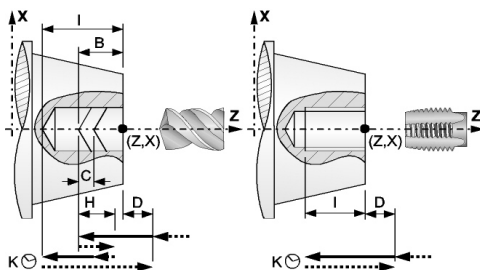
D: Safety distance

K: Dwell at the bottom of the hole (in hundredths of a second)

H: Rapid withdrawal (G0) distance after each drilling step

C: Approach distance to the previous drilling step in rapid (G00)

R: In the drilling cycle, factor that reduces the drilling peck B. In the threading cycle, type of threading.





## Function M D V Meaning

**G84**

\* Turning canned cycle for curved sections.

**G84 X Z Q R C D [L] [M] [F] [H] I K**

X: X coordinate of the profile's starting point

Z: Z coordinate of the profile's starting point

Q: X coordinate of the profile's last point

R: Z coordinate of the profile's last point

C: Machining pass (in radius)

D: Withdrawal distance after each pass

D=0: the tool exit path is the same as the entry path

D&lt;&gt;0: retracts at 45 degrees

D not programmed: retracts while following the profile to the previous step

L: Finishing stock on the X axis (in radius)

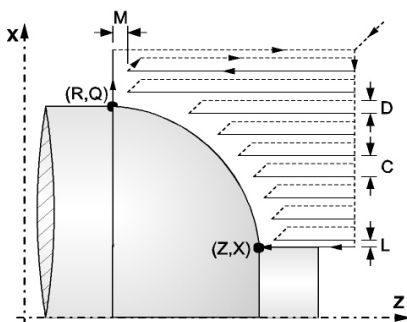
M: Defines the finishing pass along the Z axis (if not programmed, a value of 0 will be assumed)

F: Feedrate for the last roughing pass. If not programmed or programmed as F0, it does not run the last roughing pass.

H: Finishing feedrate. If not programmed or programmed as H0, it does not run the finishing pass.

I: Distance from the starting point to the arc center along the X axis

K: Distance from the starting point to the arc center along the Z axis

*Note: The radius compensation must be G41/G42 before activating G84.*

Function M D V Meaning

**G85**

**\* Facing canned cycle for curved sections.**

**G85 X Z Q R C D [L] [M] [F] [H] I K**

X: X coordinate of the profile's starting point

Z: Z coordinate of the profile's starting point

Q: X coordinate of the profile's last point

R: Z coordinate of the profile's last point

C: Machining pass (in radius)

D: Withdrawal distance after each pass

D=0: the tool exit path is the same as the entry path

D<>0: retracts at 45 degrees

D not programmed: retracts while following the profile to the previous step

L: Finishing stock on the X axis (in radius)

M: Defines the finishing pass along the Z axis (if not programmed, a value of 0 will be assumed)

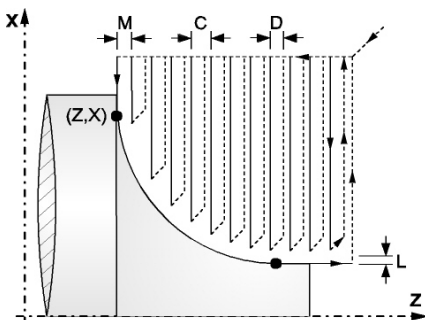
F: Feedrate for the last roughing pass. If not programmed or programmed as F0, it does not run the last roughing pass.

H: Finishing feedrate. If not programmed or programmed as H0, it does not run the finishing pass.

I: Distance from the starting point to the arc center along the X axis

K: Distance from the starting point to the arc center along the Z axis

*Note: The radius compensation must be G41/G42 before activating G85.*



Function	M	D	V	Meaning
<b>G86</b>			*	Longitudinal threading or thread repair canned cycle.

**G86 X Z Q R [K] I B [E] [D] L C [J] [A] [W]  
[V] [M] [H] [U] N**

X: X coordinate of the thread's starting point

Z: Z coordinate of the thread's starting point

Q: X coordinate of the thread's end point

R: Z coordinate of the thread's end point

K: Z coordinate of the point where the thread is measured; It is used with parameter W for thread repair

I: Thread depth (in radius)

I>0: External threading

I<0: Internal threading

B: Depth of the passes

B<0: Constant pitch increment

B>0: Decreasing pitch increment

E: Minimum value that the penetration pass can reach when B>0

D: X axis safety distance (in radius)

L: Finishing stock (in radius)

C: Thread pitch

C>0: Pitch according to the taper.

C<0: Pitch according to the associated axis

J: Exit from the thread. Z distance from the final point (R) to the point where the withdrawal from the thread begins

A: Tool penetration angle according to the X axis.

A=0: Radial penetration

A<0: Penetration in zigzag

If not programmed, a value of 30° is assumed

W: Angular position of the thread starting point.

Its meaning depends on K.

K not programmed: indicates the spindle angular position of the thread's starting point

K programmed: indicates the angular spindle position corresponding to the thread measuring point (thread repair)

V: Number of thread entries (starts)

M: Increment (positive M) or decrement (negative M) of the thread pitch per spindle turn

H: Thread entry (start) type

H=0: Radial entry

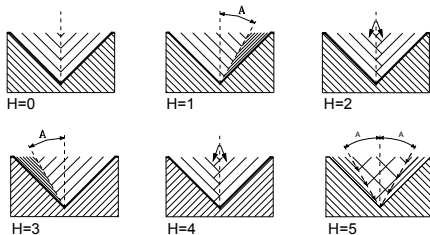
H=1: Entry by starting flank (to the right)

H=2: Zig-zag radial entry by the first center. (beginning from the initial side or right)

H=3: Entry from final flank (to the left)

H=4: Entry in radial zigzag from the final center (beginning from the final side or left)

H=5: Entry in zigzag from the flanks (alternating the beginning on both flanks)



U: Partial thread repair or machining a thread by entering directly into it.

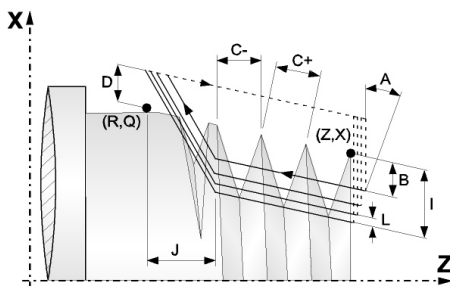
U=0: Full threading or repair

U=1: Partial threading or repair

N: Normal or blind thread

N=0: Normal thread

N=1: Blind thread



## Function M D V Meaning

**G87**

\* Face threading canned cycle.

**G87 X Z Q R [K] I B [E] [D] L C [J] [A] [W]  
[V] [M] [H] [U] N**

X: X coordinate of the thread's starting point

Z: Z coordinate of the thread's starting point

Q: X coordinate of the thread's end point

R: Z coordinate of the thread's end point

K: Z coordinate of the point where the thread is measured; It is used with parameter W for thread repair

I: Thread depth

I>0: machined in the negative direction of the Z axis

I<0: machined in the positive direction of the Z axis

B: Depth of the passes

B<0: Constant pitch increment

B>0: Decreasing pitch increment

E: Minimum value that the penetration pass can reach when B>0

D: Safety distance along Z

L: Finishing stock

C: Thread pitch

C>0: Pitch according to the taper.

C<0: Pitch according to the associated axis

J: Exit from the thread. X distance from the final point (R, Q) to the point where the withdrawal from the thread begins (in radius)

A: Tool penetration angle according to the Z axis.

A=0: Axial penetration

A<0: Penetration in zigzag

If not programmed, a value of 30° is assumed

W: Angular position of the thread starting point.

Its meaning depends on K.

K not programmed: indicates the spindle angular position of the thread's starting point

K programmed: indicates the angular spindle position corresponding to the thread measuring point (thread repair)

V: Number of thread entries (starts)

M: Increment (positive M) or decrement (negative M) of the thread pitch per spindle turn

H: Thread entry (start) type

H=0: Radial entry

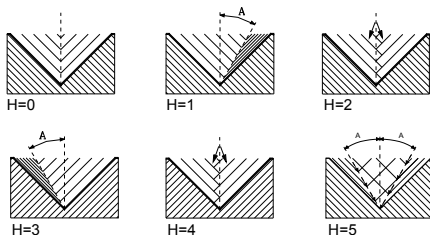
H=1: Entry by starting flank (to the right)

H=2: Zig-zag radial entry by the first center. (beginning from the initial side or right)

H=3: Entry from final flank (to the left)

H=4: Entry in radial zigzag from the final center (beginning from the final side or left)

H=5: Entry in zigzag from the flanks (alternating the beginning on both flanks)



U: Partial thread repair or machining a thread by entering directly into it.

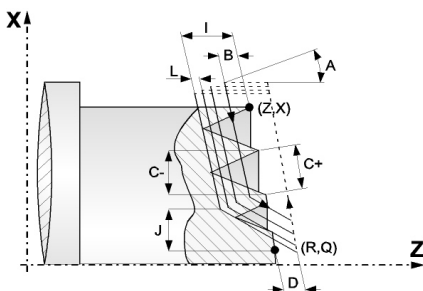
U=0: Full threading or repair



U=1: Partial threading or repair

N: Normal or blind thread

N=0: Normal thread

N=1: Blind thread



**Function**    **M**   **D**   **V**   **Meaning**
**G88**          \*   Grooving canned cycle along the X axis.

**G88 X Z Q R [C] D [K]**

X: X coordinate of the groove's starting point

Z: Z coordinate of the groove's starting point

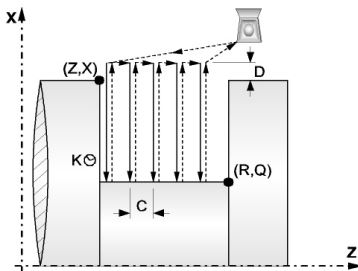
Q: X coordinate of the groove's end point

R: Z coordinate of the groove's end point

C: Grooving pass

D: X axis safety distance (in radius)

K: Dwell, in hundredths of a second, after each penetration until the withdrawal begins



Function M D V Meaning

**G89** \* Z axis grooving canned cycle.

**G89 X Z Q R [C] [D] [K]**

X: X coordinate of the groove's starting point

Z: Z coordinate of the groove's starting point

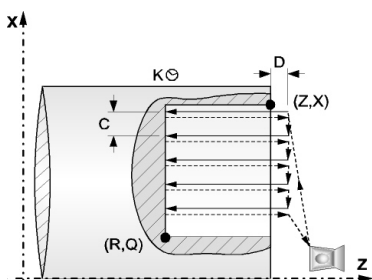
Q: X coordinate of the groove's end point

R: Z coordinate of the groove's end point

C: Grooving pass (in radius)

D: Safety distance along Z

K: Dwell, in hundredths of a second, after each penetration until the withdrawal begins





Function	M	D	V	Meaning
----------	---	---	---	---------

<b>G160</b>			*	Drilling / tapping canned cycle on the face of the part.
-------------	--	--	---	--

**Drilling:****G160 X Z I B Q A J [D] [K] [H] [C] S [R] N****Tapping:****G160 X Z I B0 Q A J [D] S [R] N**

X: X coordinate of the cycle starting point

Z: Z coordinate of the cycle starting point

I: Machining depth

B: Type of machining

B=0: Tapping

B&gt;0: Drilling (where B indicates the drilling peck)

Q: Angular position of the spindle for the first machining operation

A: Angular increment between machining operations

J: Total number of machining operations

D: Safety distance along Z

K: Dwell at the bottom of the hole, in hundredths of a second

H: Rapid withdrawal (G0) distance after each drilling step

C: Approach distance to the previous drilling step in rapid (G00)

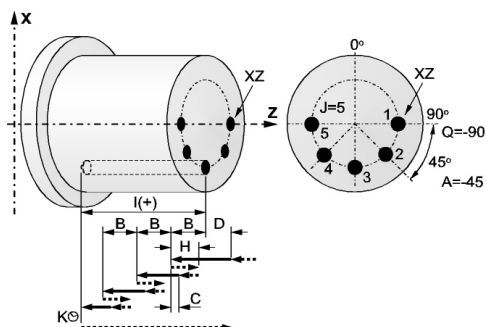
S: Live tool turning speed and direction. The sign defines the turning direction.

R: For the drilling cycle (B&gt;0), factor that reduces the drilling peck B. In the threading cycle (B=0), indicates the type of threading:

R=0: Tapping with clutch

R=1: Rigid tapping

N: Number of the spindle for the live tool



Function	M	D	V	Meaning
<b>G161</b>			*	Drilling / tapping canned cycle on the side of the part.

**Drilling:**

**G161 X Z I B Q A J [D] [K] [H] [C] S [R] N**

**Tapping:**

**G161 X Z I B0 Q A J D S [R] N**

X: X coordinate of the cycle starting point

Z: Z coordinate of the cycle starting point

I: Machining depth

B: Type of machining

B=0: Tapping

B>0: Drilling (where B indicates the drilling peck)

Q: Angular position of the spindle for the first machining operation

A: Angular increment between machining operations

J: Total number of machining operations

D: Safety distance along X

K: Dwell at the bottom of the hole, in hundredths of a second

H: Rapid withdrawal (G0) distance after each drilling step

C: Approach distance to the previous drilling step in rapid (G00)

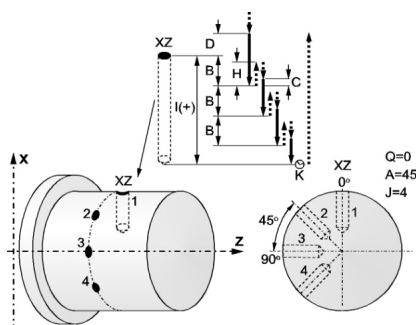
S: Live tool turning speed and direction. The sign defines the turning direction.

R: For the drilling cycle (B>0), factor that reduces the drilling peck B. In the threading cycle (B=0), indicates the type of threading:

R=0: Tapping with clutch

R=1: Rigid tapping

N: Number of the spindle for the live tool



Function	M	D	V	Meaning
----------	---	---	---	---------

<b>G162</b>			*	Slot milling canned cycle along the side of the part.
-------------	--	--	---	---

**G162 X Z L I Q A J [D] F S N**

X: X coordinate of the cycle starting point

Z: Z coordinate of the cycle starting point

L: Length of the slot referred to the starting point

I: Depth of the slot referred to the starting point

Q: Angular position of the spindle for the first machining operation

A: Angular increment between machining operations

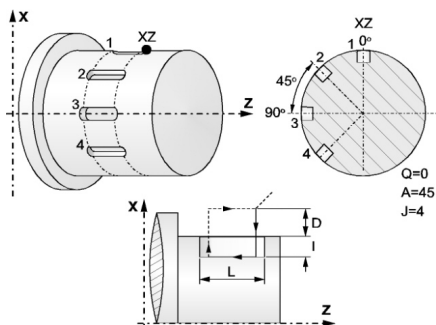
J: Total number of machining operations

D: Safety distance along X

F: Machining feedrate

S: Live tool turning speed and direction. The sign defines the turning direction.

N: Number of the spindle for the live tool



Function	M	D	V	Meaning
<b>G163</b>			*	Slot milling canned cycle along the face of the part.

**G163 X Z L I Q A J [D] F S N**

X: X coordinate of the cycle starting point

Z: Z coordinate of the cycle starting point

L: Length of the slot referred to the starting point

I: Depth of the slot referred to the starting point

Q: Angular position of the spindle for the first machining operation

A: Angular increment between machining operations

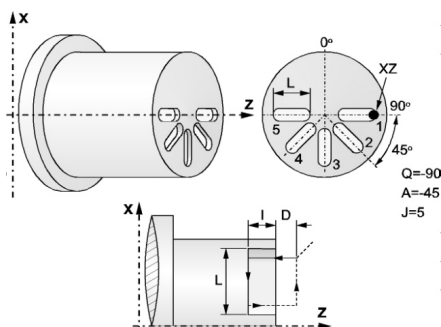
J: Total number of machining operations

D: Safety distance along Z

F: Machining feedrate

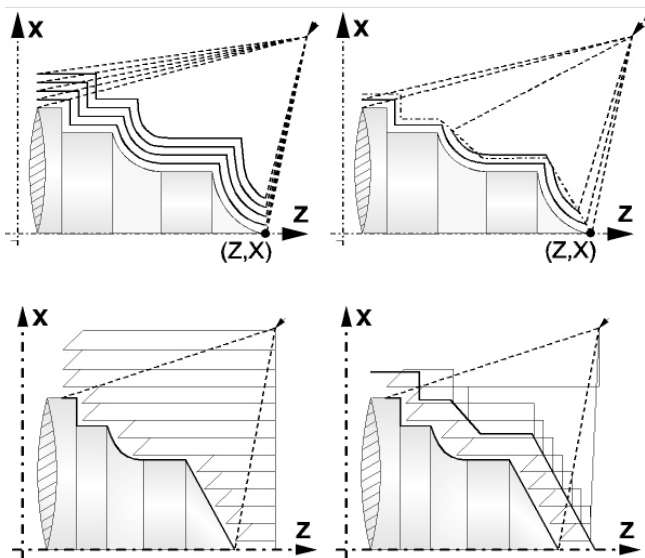
S: Live tool turning speed and direction. The sign defines the turning direction.

N: Number of the spindle for the live tool



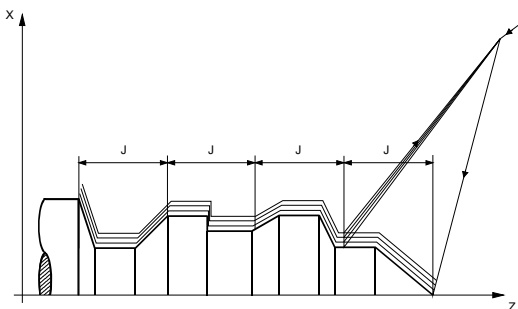
## G66 G68 G69 machining optimization

If only the desired profile is defined, the CNC assumes that the original stock is cylindrical and machines it as shown on the left.



When the part profile is known, it is recommended to define both profiles, that of the rough part and the desired final profile. The machining is faster since only the stock between both profiles is removed.

By using parameter J it is possible to define the maximum movement for each section along the machining axis (machining is divided into longitudinal sections equal to J millimeters).





The CNC has two work modes:

- Basic mode (M / T):

It provides all the functionalities of the machine (maintenance, CNC programs, PLC programs, parameters, diagnosis, communication lines, etc.).

- (MC / TC) conversational mode:

Intuitive operation and programming of the machine. Allows for cycles to be programmed without using the ISO language.

Programming in conversational mode is based on completing a series of cycles according to the machining to be performed.

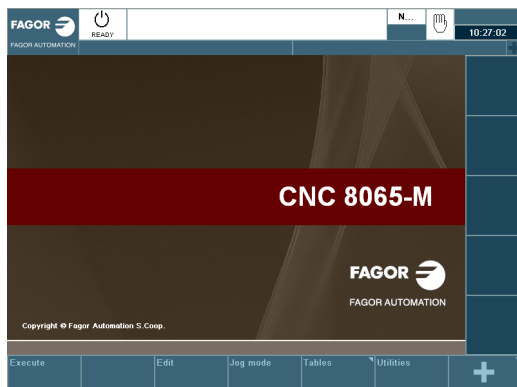
These cycles can be stored in a program or executed without being stored.

## Conversational mode

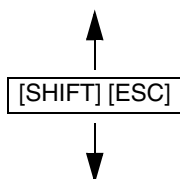
### Accessing the conversational mode.

Once the CNC has been started up, press the [SHIFT] [ESC] keys to switch to conversational mode.

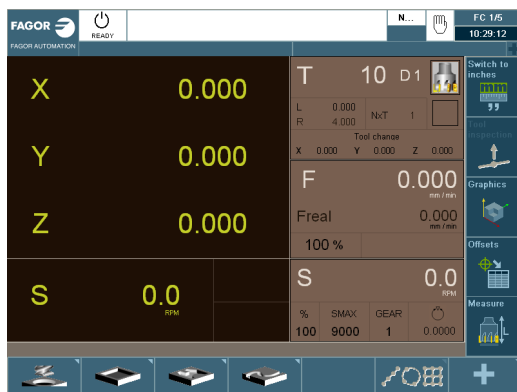
Press [SHIFT][ESC] again to return to basic mode.



Basic (M / T) mode



(MC / TC) conversational mode





## Conversational mode screens.

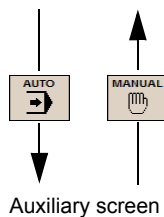
The conversational mode has two screens where the information is displayed.

- Standard screen.
- Auxiliary screen.

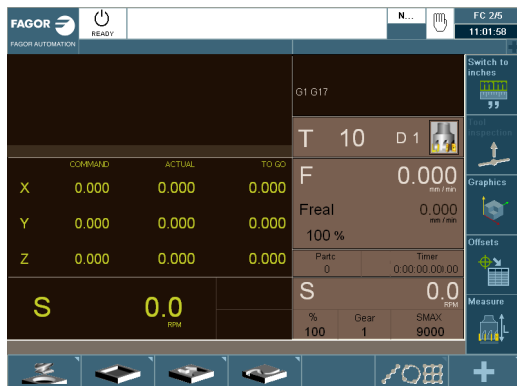
To switch from one screen to another, press the two-color key.



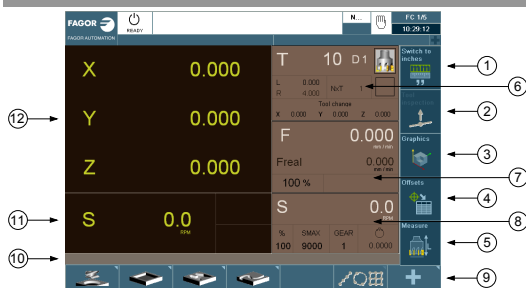
Standard screen



Auxiliary screen



## Description of the conversation mode standard screen.



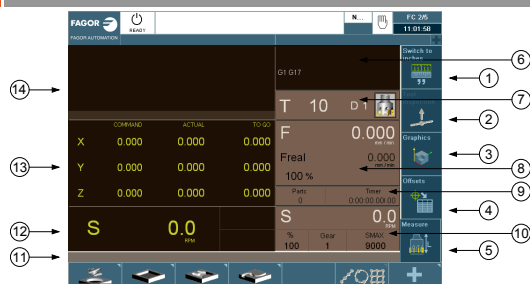
1. Softkey for selecting units mm/inches.
2. Softkey to go into tool inspection.
3. Softkey to access the graphics in execution mode.
4. Softkey for selecting offsets.
5. Softkey for tool calibration.
6. Window that displays information on the selected tool.
7. Window that displays information on the feedrate of the axis that is selected.
8. Window that displays information on the spindle.
9. Softkeys for cycle editing.
10. Message bar.
11. Real spindle rpm.
12. Position (coordinates) of the axes. The symbol  $\phi$  indicates that the axis is working in diameters.

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.



Pressing the [DEL] key removes the program found on the header (Program selected for execution). This prevents program execution from commencing if the [START] key is pressed.

## Description of the conversation mode auxiliary screen.

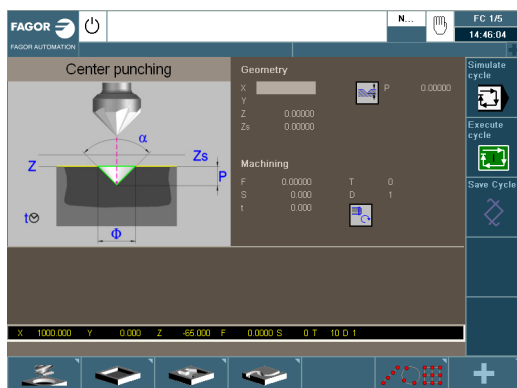


1. Softkey for selecting units mm/inches.
2. Softkey to go into tool inspection.
3. Softkey to access the graphics in execution mode.
4. Softkey for selecting offsets.
5. Softkey for tool calibration.
6. Window that shows the status of the functions G, F, T, D, M. It also shows the identifiers that a high-level command has been programmed, such as FIX, KIN, CS...
7. Window that displays information on the selected tool.
8. Window that displays information on the feedrate of the axis that is selected.
9. Window that shows the value of the variables Partc, CyTime, Timer.
10. Window showing spindle related information:
11. Message bar.
12. Window with spindle related information:
  - Theoretical speed.
  - Speed in RPM.
  - Speed in m/min.
13. Window with axis related information:
  - COMMAND.
  - ACTUAL.
  - TO GO.
14. Window that shows the lines of the program being executed.

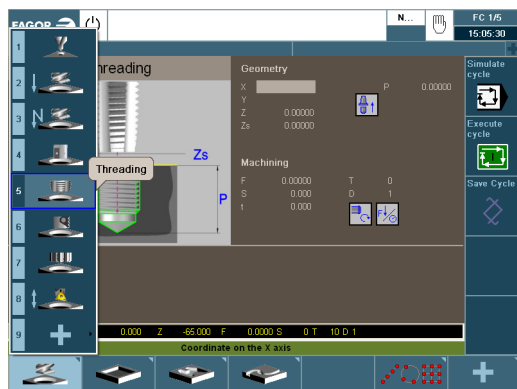
## Cycle editing

### 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.

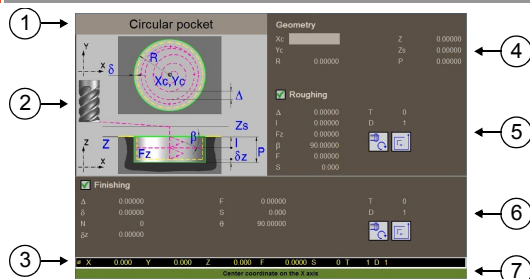


Once the cycle to be edited has been selected, enter the data in the windows corresponding to each one of the parameters. To save the data and move on to the next parameter, press [ENTER].

The cycle icons can be modified using the two-color key.



## Description of the cycle screen.

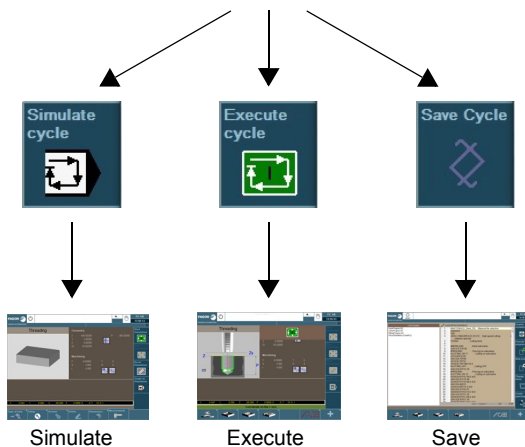
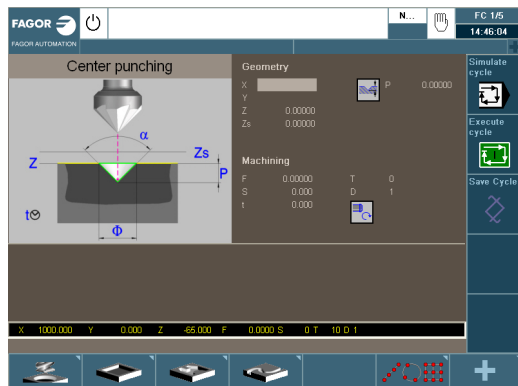


1. Name of the cycle.
2. Image showing the cycle parameters.
3. Current parameters for the CNC (position, tool, corrector).
4. Geometric parameters.
5. Roughing parameters.
6. Finishing parameters.
7. Description of the selected parameter to be edited.

## Simulation, execution and saving of a cycle

### Simulation, execution and saving of a cycle.

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



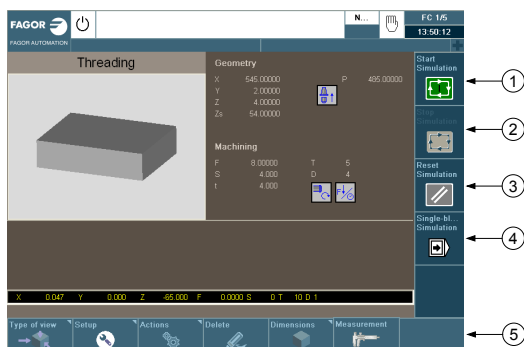
## Simulating a cycle.

Simulate  
cycle



After editing a cycle, it can be simulated by pressing the vertical simulation softkey.

The simulation screen displays the following softkeys:



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.

*Note: by pressing [CTRL] + [G] zoom into graphic area to see the entire cycle area.*

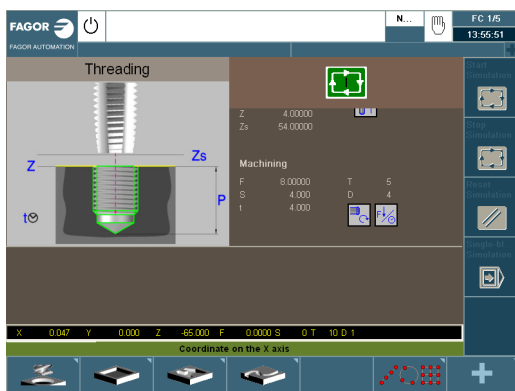
## Cycle execution.

Execute  
cycle



After editing a cycle, it can be executed by pressing the vertical softkey [Execute cycle].

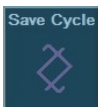
Following, an icon with the start symbol will appear. This symbol lets the user know that the cycle can be executed.



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

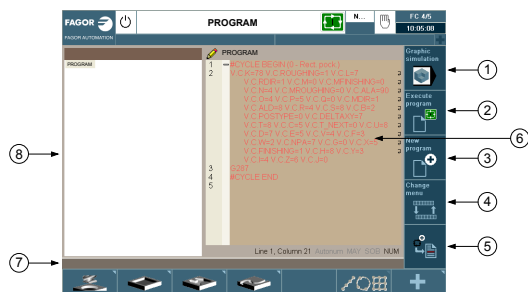


## Saving a cycle.



After editing a cycle, it can be saved by pressing the vertical softkey [Save cycle].

The saving screen displays the following softkeys:



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.  
After pressing this softkey, the following softkeys appear:
  - Open program
  - Operat. with blocks
  - Undo/Redo
  - Insert rect. pocket
5. Replace or insert the edited cycle. To insert a cycle, move the cursor and the cycle will be inserted below it.
6. Window that shows the cycles and ISO-coded blocks that make up the selected part. To edit a cycle, press [RECALL] with the cursor over the cycle.
7. Message bar.
8. Window showing the list of saved programs. 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.



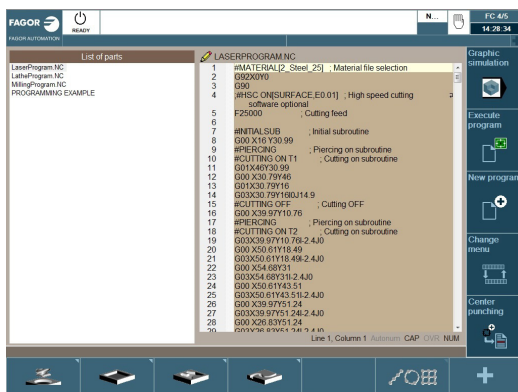
To toggle the focus between the list of programs and the part editor, press [CTRL]+[F2] or the change window key.

## Simulation and execution of a part program

### Simulation and execution of a part program.

Follow these steps when simulating or executing a part program:

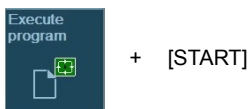
- Press [EDIT] to access the list of part-programs stored.
- Select on the left column the program to be simulated or executed.
- Press the vertical softkey "Graphic simulation" or "Execute program".



Simulate:



Execute:

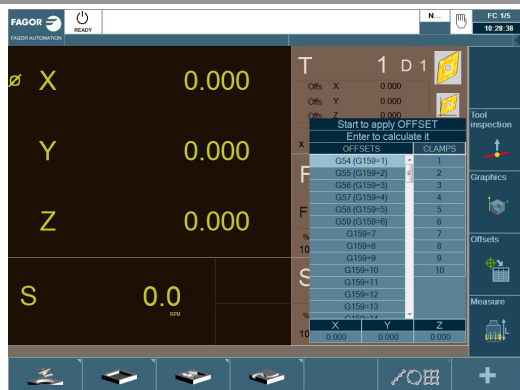


To toggle the focus between the list of programs and the part editor, press [CTRL]+[F2] or the change window key.

*Note: with the cursor in the program section (right side), by pressing F11 + Execute, with a START it will launch the execution of the part-program from the block where the cursor is located.*

## Offsets table

Offset table (zero and fixture offset).



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 save it. The new zero offset is applied to all the axes of the channel.

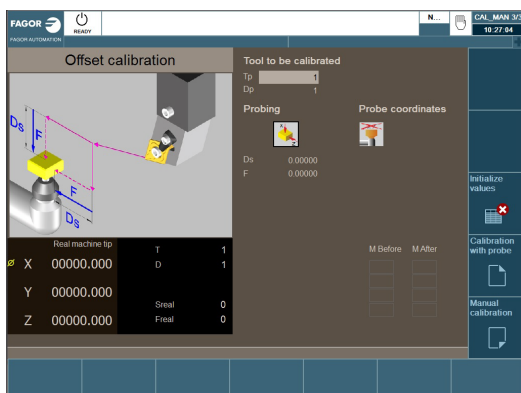
## Tool calibration

### Tool calibration.

#### Measure



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.



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.

#### Calibration with probe



#### Manual 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.

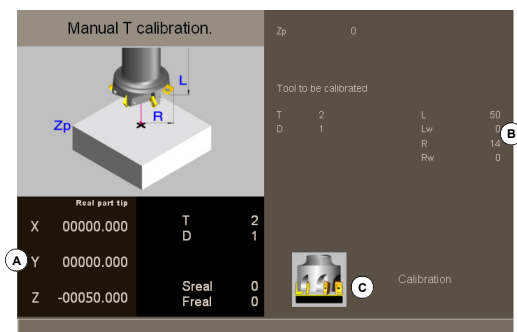
## Manual calibration (without a probe).

### Manual calibration



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. Area to define the calibration data.
- C. Drawing indicating that tool calibration is allowed. If the drawing is not displayed, some of the data is missing. This diagram, which blinks, indicates that the tool is active and, therefore, it can be calibrated.

Calibration  
with probe

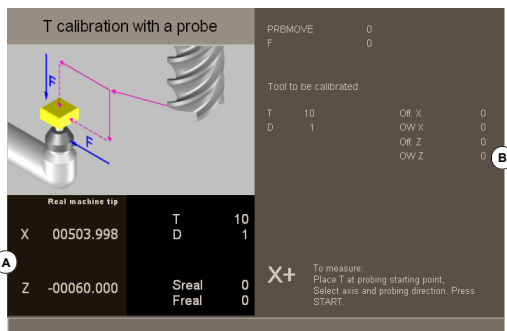


## Semi-automatic calibration (with a probe).

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 or radius of the milling tools.

On a lathe model, it may be used to calibrate the offsets of any tool.













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

B. Area to define the calibration data.

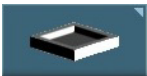






The tool must be in the spindle. After the calibration, the wear is reset to zero.

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




# CONVERSATIONAL CYCLES (M)

	Z axis machining.	
	1	 Center punching
	2	 Drilling 1
	3	 Drilling 2
	4	 Bore milling
	5	 Tapping
	6	 Thread milling
	7	 Reaming
	8	 Boring 1
	9	 Boring with spindle orientation

*Note: repetitions can be made for Z machining cycles. See page 121.*

	Pockets / Bosses.	
	1	 Simple rectangular pocket
	2	 Rectangular pocket
	3	 Circular pocket
	4	 Pre-empted pocket
	5	 Rectangular boss
	6	 Circular boss













*Note: repetitions can be made for pockets / bosses cycles. See page 121.*










	2D/3D profile pockets.	
	1	 2D pocket
	2	 3D pocket

*Note: repetitions can be made for 2D profile pocket cycles. See page 121.*

*Note: repetitions cannot be made for 3D profile pocket cycles.*



	Roughing.	
	1 	Profile by points
	2 	Profile
	3 	Random
	4 	Grooving
	Repetitions.	
	1 	Points in line
	2 	Rectangle
	3 	Grid
	4 	Rect. boss
	5 	Profile points
	6 	Without multiple machining

	Machining in Z (using rotary axis).	
	1	 Multiple center-punching
	2	 Multiple reaming
	3	 Multiple boring
	4	 Multiple bore milling
	5	 Multiple thread milling
	6	 Multiple drilling
	7	 Multiple threading
	8	 Multiple slot milling

## Profiles and pockets (rotary axis).



Profiles and pockets in the ZA/ZY plane



Profile in the ZA/ZY plane



ZA/ZY rectangular pocket



ZA/ZY circular pocket



ZA/ZY pre-empted pocket



ZA/ZY 2D profile pocket



Profiles and pockets on the XA/XY plane



Profile on the XA/XY plane



XA/XY rectangular pocket



XA/XY circular pocket





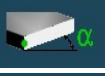








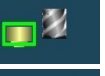

XA/XY pre-empted pocket



XA/XY 2D profile pocket

*Note: the selected rotary axis (in this case, axis A) affects the work plane.*

*Note: repetitions can be made for profile and pocket cycles (rotary axis). See page 121.*

	Measuring and centering cycles.	
4		 Measurement
	1	 Outside corner and angle measurement
	2	 Boss measurement
	3	 Hole measurement
	4	 Measuring an inside corner
	5	 Measuring an outside corner
	6	 Surface measurement
	7	 Angle measurement
		Part centering
		Probe calibration
	1	 Tabletop probe calibration
	2	 Calibrating the measuring probe

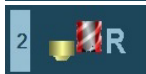
## Calibration cycles.



### Tool calibration



L-R wear measurement



R wear measurement



L wear measurement



L-R calibration



R calibration

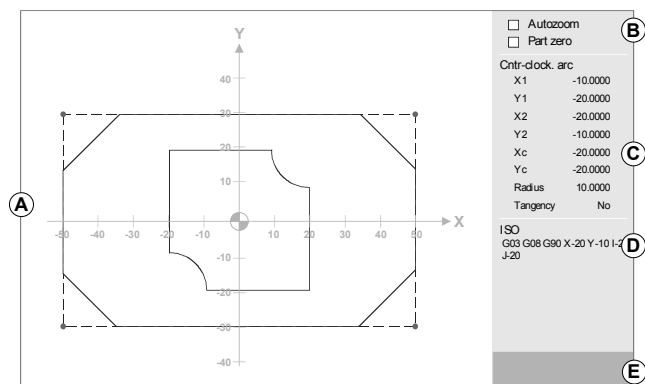


L calibration

## Profile editor

### Interface description

The profile editor is used to edit quickly and easily simple rectangular, circular profiles and any type of profile consisting of straight and curved sections. As the profile data is entered, the editor shows a graphic representation of the profile.


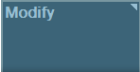
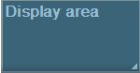
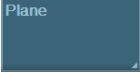

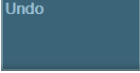
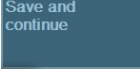


- A. Graphic area. Graphic representation of the profile being drawn, axes coordinated with autoscale and name of the axes that make up the plane. The name of the axis indicates the positive direction of the axis.
- B. Status of the autozoom and part zero options, regarding the display of the profile at the editor.
- C. Data entry area.
- D. Translation (conversion) of the selected profile or part of it into ISO code.
- E. Area used to enter the values of the corners or the ISO coded text to be added to the element.

## Softkey menu

The options that may be selected from the softkey menu make it possible to edit profiles, modify edited profiles, select the zoom, the work plane, undo the last change and end the editing session. While editing or modifying the profile, the softkey menu offers the option to undo the last operation.











Likewise, it offers the option to save the profile at any time.

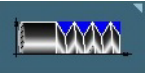
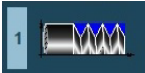
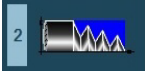

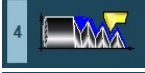
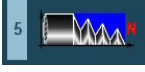
	Softkey menu.
	<b>Edit.</b> Edit a new profile, enlarge an existing profile or import a profile saved in DXF format.
	<b>Modify.</b> Modify, insert or delete elements of a profile.
	<b>Displayed area.</b> Modify the zoom of the graphics area.
	<b>Plane.</b> Define the work plane.
	<b>End.</b> End the profile editing session and insert the edited profiles into the program.
	<b>Undo.</b> Undo the last operation.
	<b>Save and continue.</b> Save the profile and continue editing. Using this key does not require that the profile be completed.


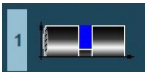



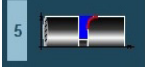



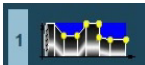















# CONVERSATIONAL CYCLES (T)


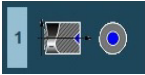
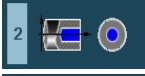
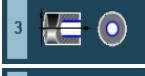


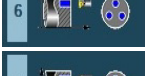


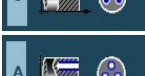


Turning.		
	1	 Simple turning
	2	 Turning with vertex rounding
	3	 Simple facing
	4	 Facing with vertex rounding
	5	 Corner chamfering
	6	 Chamfering between points
	7	 Corner chamfering 2
	8	 Corner rounding
	9	 Rounding between points




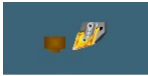

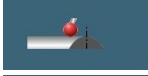
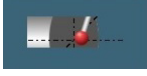
Threads.		
	1 	Longitudinal threading
	2 	Taper threading
	3 	Face threading
	4 	Thread repair
	5 	Threading with n starts (entries)

Grooving.		
	1 	Simple longitudinal grooving
	2 	Simple face grooving
	3 	Inclined longitudinal grooving
	4 	Inclined face grooving
	5 	Cut off

Profiles.	
	
	Point-to-point turning
	Profile turning
	Profiles and pockets in the ZC/YZ plane
	Profile in the ZC/YZ plane
	ZC/YZ rectangular pocket
	ZC/YZ circular pocket:
	ZC/YZ pre-empted pocket
	ZC/YZ 2D profile pocket
	Profiles and pockets in the XC/XY plane
	Profile in the XC/XY plane
	XC/XY rectangular pocket
	XC/XY circular pocket
	XC/XY pre-empted pocket
	XC/XY 2D profile pocket

*NOTE: repetitions can be made for all of the above profile cycles, except point-to-point turning and profile turning. See page 121.*

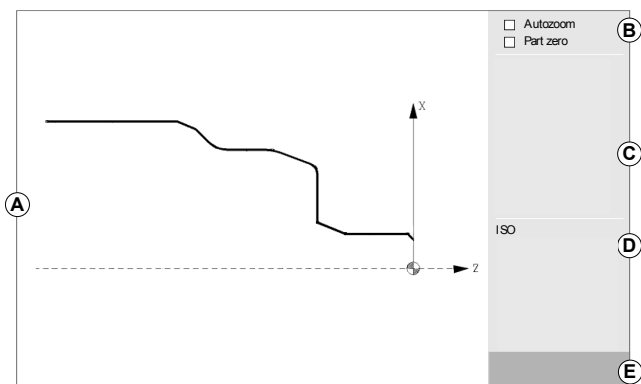
	Z axis machining.	
		Center punching
		Drilling 1
		Tapping
		Multiple center-punching
		Multiple reaming
		Multiple boring
		Multiple bore milling
		Multiple thread milling
		Multiple drilling
		Multiple threading
		Multiple slot milling

Positioning.		
		Positioning
		Positioning and M functions
Measuring and calibration cycles.		
		T calibration
		Probe calibration
		Part longitudinal calibration with a probe
		Part front centering with a probe

## Profile editor

### Interface description

The profile editor is used to edit quickly and easily simple rectangular, circular profiles and any type of profile consisting of straight and curved sections. As the profile data is entered, the editor shows a graphic representation of the profile.



- A. Graphic area. Graphic representation of the profile being drawn, axes coordinated with autoscale and name of the axes that make up the plane. The name of the axis indicates the positive direction of the axis.
- B. Status of the autozoom and part zero options, regarding the display of the profile at the editor.
- C. Data entry area.
- D. Translation (conversion) of the selected profile or part of it into ISO code.
- E. Area used to enter the values of the corners or the ISO coded text to be added to the element.

## Softkey menu

The options that may be selected from the softkey menu make it possible to edit profiles, modify edited profiles, select the zoom, the work plane, undo the last change and end the editing session. While editing or modifying the profile, the softkey menu offers the option to undo the last operation.

Likewise, it offers the option to save the profile at any time.

	Softkey menu.
Edit	<b>Edit.</b> Edit a new profile, enlarge an existing profile or import a profile saved in DXF format.
Modify	<b>Modify.</b> Modify, insert or delete elements of a profile.
Display area	<b>Displayed area.</b> Modify the zoom of the graphics area.
Plane	<b>Plane.</b> Define the work plane.
Finish	<b>End.</b> End the profile editing session and insert the edited profiles into the program.
Undo	<b>Undo.</b> Undo the last operation.
Save and continue	<b>Save and continue.</b> Save the profile and continue editing. Using this key does not require that the profile be completed.

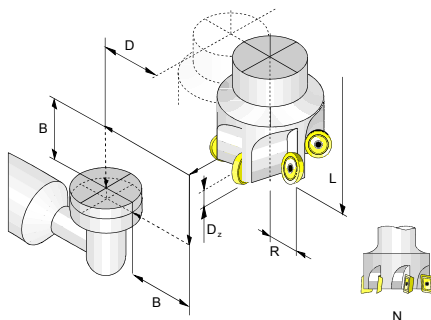




# MEASURING AND CALIBRATION CYCLES (M)

*Note: The parameters indicated between angular brackets are optional.*

Cycle	Meaning
<b>#PROBE 1</b>	Tool calibration (dimensions and wear).
	<b>#PROBE 1 B [I] [J] F [K] [S] [N] [D] [E] [L] [M] [C] [X U Y V Z W]</b>
	B: Safety distance
	I: Dimension of the tool to be calibrated (if I0, length on the axis; if I1, length at one end; if I2, radius; if I3, radius and length)
	J: Type of operation (if J0, calibration; if J1, wear measurement)
	F: Probing feedrate
	K: Side of the probe being used (if K0, X+ side; if K1, X- side; if K2, Y+ side; if K3, Y- side)
	S: Speed and turning direction of the tool
	N: Number of edges to be measured (if N=0 or when not programmed, one turning measurement; if N=+n, exact measurement of the first edge and the rest are equidistant, if N=-n, exact measurement of the n edges)
	D: Distance from the probing point to the tool shaft
	E: Distance from the probing point to the tool base
	L: Maximum length wear allowed
	M: Maximum radius wear allowed
	C: Behavior when exceeding the maximum wear
	X..W: Tabletop probe position



**Cycle**

**Meaning**

**#PROBE 2**

Probe calibration.

**#PROBE 2 X Y Z B J E H F**

X..Y: Real coordinates of the arc center

When these coordinates are programmed, they are considered the exact coordinates of the center. If they are not programmed, the CNC will deem that they are not accurate and needs the function M19 for a correct calibration.

Z: Position where probing takes place

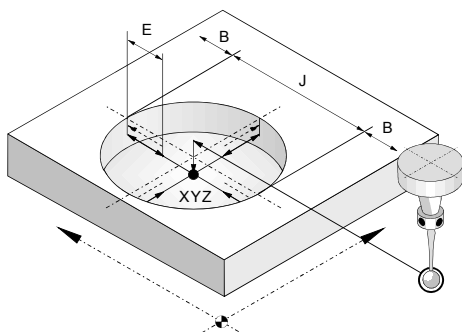
B: Safety distance

J: Theoretical diameter of the hole

E: Withdrawal distance after initial probing

H: Feedrate for the first probing movement

F: Feedrate for the second probing movement



**Cycle****Meaning****#PROBE 3****Surfacing measuring.****#PROBE 3 X Y Z B [K] F [C] [L] [T] [D]**

X Y Z: Theoretical coordinates of the probing point

B: Safety distance

K: Probing axis (if K0, abscissa axis; if K1, ordinate axis; if K2, axis perpendicular to the plane)

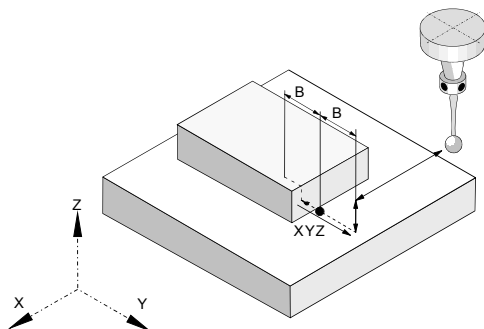
F: Probing feedrate

C: Cycle end point (if C0, at the calling point; if C1 at the measured point)

L: Tolerance for the measuring error

T: Tool whose wear is to be corrected

D: Offset whose wear is to be corrected



**Cycle**

**Meaning**

**#PROBE 4**

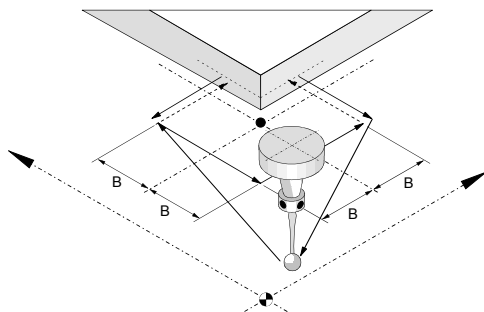
Outside corner measuring.

**#PROBE 4 X Y Z B F**

X..Z: Theoretical coordinates corner being measured

B: Safety distance

F: Probing feedrate



**#PROBE 5**

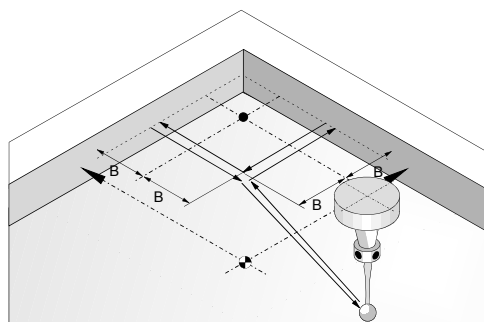
Inside corner measuring.

**#PROBE 5 X Y Z B F**

X..Z: Theoretical coordinates corner being measured

B: Safety distance

F: Probing feedrate



## Cycle

## Meaning

### #PROBE 6

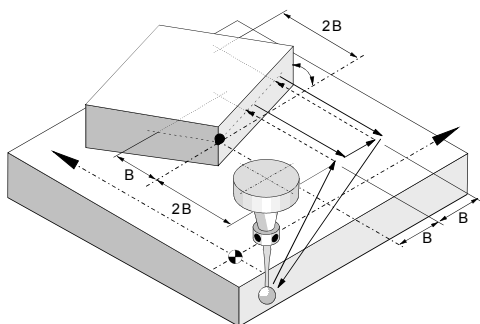
Angle measurement on the abscissa axis.

#### #PROBE 6 X Y Z B F

X..Z: Theoretical coordinates corner being measured

B: Safety distance

F: Probing feedrate



### #PROBE 7

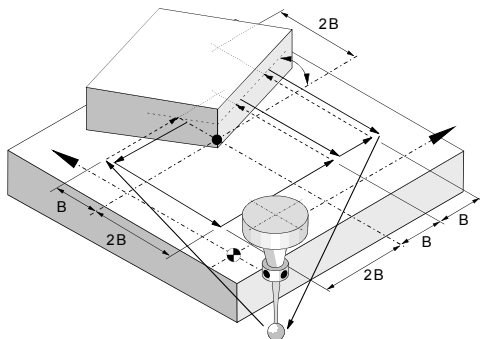
Outside corner and angle measurement.

#### #PROBE 7 X Y Z B F

X..Z: Theoretical coordinates corner being measured

B: Safety distance

F: Probing feedrate



**Cycle**

**Meaning**

**#PROBE 8**

Hole measuring.

**#PROBE 8 X Y Z B J E [C] H F**

X..Z: Theoretical coordinates of the arc center

B: Safety distance

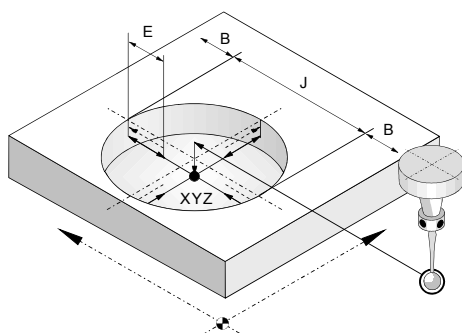
J: Theoretical diameter of the hole

E: Withdrawal distance after initial probing

C: Cycle end point (if C0, at the calling point; if C1 in the actual - real - center of the hole)

H: Feedrate for the first probing movement

F: Feedrate for the second probing movement



**Cycle****Meaning****#PROBE 9**

Circular boss measuring.

**#PROBE 9 X Y Z B J E [C] H F**

X..Z: Coordinates of the boss center

B: Safety distance

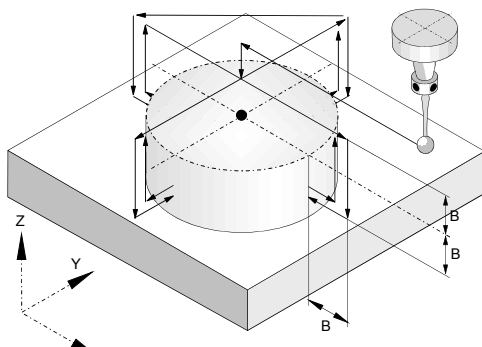
J: Theoretical boss diameter

E: Withdrawal distance after initial probing

C: Cycle end point (if C0, at the calling point;  
if C1 in the actual - real - center of the hole)

H: Feedrate for the first probing movement

F: Feedrate for the second probing movement



**Cycle**

**Meaning**

**#PROBE 10**

Rectangular part centering.

**#PROBE 10 [XYZ] I J [K] [L] [B] DE [H]  
[F] [Q] [C]**

X..Z: Probe position when calling the cycle

I: Part length along the abscissa axis

J: Part length along the ordinate axis

K: Axis and direction of the first probing movement (if K0, in the positive X direction; if K1, in the negative X direction; if K2, in the positive Y direction; if K1, in the negative Y direction)

L: Surface coordinate measurement (if L0, do not measure the coordinate; if L1, measure the coordinate)

B: Safety distance

D: Safety distance along Z

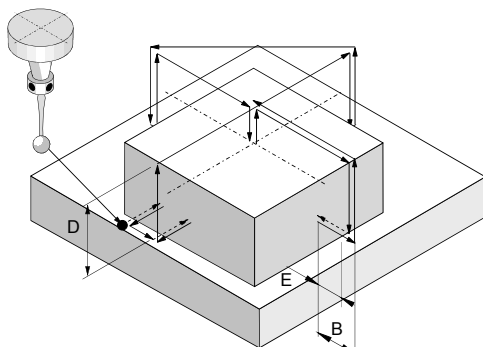
E: Withdrawal distance after initial probing

H: Feedrate for the first probing movement

F: Feedrate for the second probing movement

Q: Feedrate for approach movements

C: Centering type





**Cycle****Meaning****#PROBE 11**

Circular part centering.

**#PROBE 11 [X Y Z] J [K] [L] [B] D E [H]  
[F] [Q] [C]**

X..Z: Probe position when calling the cycle

J: Part diameter

K: Axis and direction of the first probing movement (if K0, in the positive X direction; if K1, in the negative X direction; if K2, in the positive Y direction; if K1, in the negative Y direction)

L: Surface coordinate measurement (if L0, do not measure the coordinate; if L1, measure the coordinate)

B: Safety distance

D: Safety distance along Z

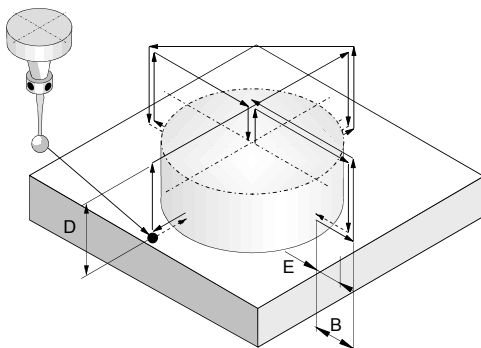
E: Withdrawal distance after initial probing

H: Feedrate for the first probing movement

F: Feedrate for the second probing movement

Q: Feedrate for approach movements

C: Centering type



**Cycle**

**Meaning**

**#PROBE 12**

Tabletop probe calibration.

**#PROBE 12 B E [H] [F] [I] [X U Y V Z W]**

B: Safety distance

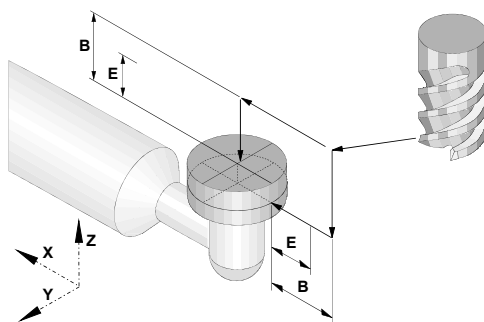
E: Withdrawal distance after initial probing

H: Feedrate for the first probing movement

F: Feedrate for the second probing movement

I: Type of calibration (if I0, single calibration; if I1 double calibration)

X..W: Tabletop probe position



# MEASURING AND CALIBRATION CYCLES (T)

*Note: The parameters indicated between angular brackets are optional.*

Cycle	Meaning
-------	---------

<b>#PROBE 1</b>	Tool calibration.
-----------------	-------------------

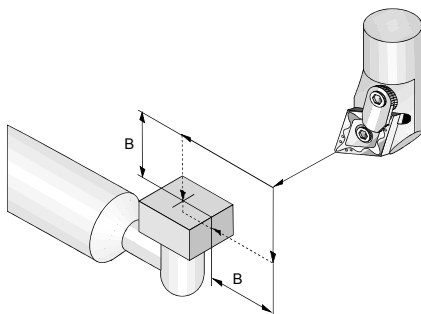
**#PROBE 1 B F [K] [X U Y V Z W]**

B: Safety distance (in radius)

F: Probing feedrate

K: Sides of the probe being used (if K0, X Z sides; if K1, X Z Y+ sides; if K2, X Z Y- sides)

X..W: Tabletop probe position



<b>#PROBE 2</b>	Tabletop probe calibration.
-----------------	-----------------------------

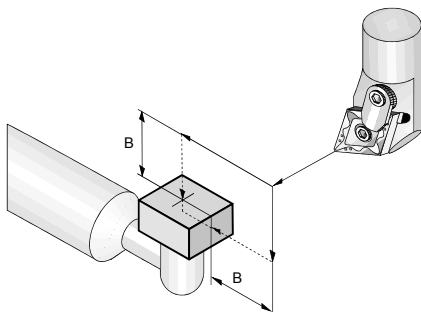
**#PROBE 2 B F [K] [X U Y V Z W]**

B: Safety distance (in radius)

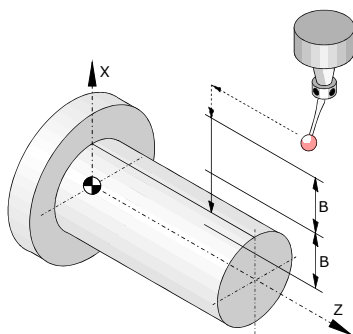
F: Probing feedrate

K: Sides of the probe being used (if K0, X Z sides; if K1, X Z Y+ sides; if K2, X Z Y- sides)

X..W: Tabletop probe position



Cycle	Meaning
<b>#PROBE 3</b>	Part measurement along the ordinate axis.
<b>#PROBE 3 X Z B F [L] [T D]</b>	
X: Theoretical coordinate of the probing point along the ordinate axis	
Z: Theoretical coordinate of the probing point along the abscissa axis	
B: Safety distance (in radius)	
F: Probing feedrate	
L: Tolerance for the measuring error	
T: Tool whose wear is to be corrected	
D: Offset whose wear is to be corrected	



## Cycle

## Meaning

### #PROBE 4

Part measurement along the abscissa axis.

### #PROBE 4 X Z B F [L] [T D]

X: Theoretical coordinate of the probing point along the ordinate axis

Z: Theoretical coordinate of the probing point along the abscissa axis

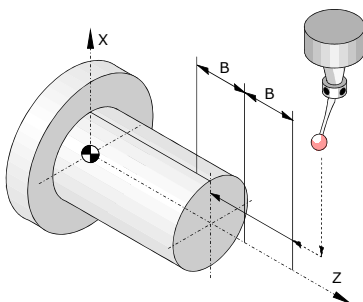
B: Safety distance

F: Probing feedrate

L: Tolerance for the measuring error

T: Tool whose wear is to be corrected

D: Offset whose wear is to be corrected





## Flow controlling instructions

Instruction	Meaning
<b>\$GOTO</b>	Block skip.  <b>\$GOTO N...:</b> jump to a block number (it may be a number, parameter or arithmetic expression whose result is a number). The number must be programmed after the “:” E.g.: \$GOTO N60 ... N60:  <b>\$GOTO [...]:</b> jump to a label (it may be a sequence of up to 14 characters consisting of uppercase and lowercase letters and numbers, without blank spaces quotation marks). E.g.: \$GOTO [L=1] ... [L=1]

Instruction	Meaning
<b>\$IF</b> <b>\$ENDIF</b> <b>\$ELSEIF</b> <b>\$ELSE</b> <b>\$GOTO</b>	<p>Conditional execution IF (conditional jump).</p> <p><b>\$IF &lt;condition&gt;</b>  <b>\$ENDIF</b>  <b>\$ELSEIF</b>  <b>\$ELSE</b>  <b>\$GOTO</b></p> <p>Note 1:  \$IF P1==1  ...  ...  \$ENDIF  ...  If the condition is true, it executes the blocks between \$IF and \$ENDIF.  If the condition is false, the execution jumps to the block after \$ENDIF.</p> <p>Note 2:  \$IF P1==1  ...  ...  \$ELSE  ...  \$ENDIF  ...  If the condition is true, it executes the blocks between \$IF and \$ELSE, after the execution it continues from the following block to \$ENDIF.  If the condition is false, it executes the blocks between \$ELSE and \$ENDIF, after the execution it continues from the next block to \$ENDIF.</p> <p>Note 3:  \$IF P1=1 \$GOTO N60  ...  ...  N60:  If the condition is true, it jumps to N60, otherwise it continues from the following block to \$IF.</p>



Instruction	Meaning
<b>\$SWITCH</b> <b>\$CASE</b> <b>\$ENDSWITCH</b> <b>\$BREAK</b> <b>\$DEFAULT</b>	Conditional execution SWITCH.  <b>\$SWITCH &lt;expression1&gt;</b> <b>\$SWITCH &lt;expression2&gt;</b> <b>\$ENDSWITCH</b> <b>\$BREAK</b> <b>\$DEFAULT</b>  <p>Note: this instruction calculates the result of &lt;expression1&gt; and executes the blocks contained between the \$CASE instruction, whose &lt;expression2&gt; has the same value as the calculated result and the corresponding \$BREAK instruction.</p> <p>The \$SWITCH instruction always ends with a \$ENDSWITCH.</p> <p>The \$CASE instruction always ends with a \$BREAK. As many \$CASE instructions as necessary may be programmed.</p> <p>As an option, a \$DEFAULT instruction may be inserted in such a way that if the result of &lt;expression1&gt; does not coincide with the value of any &lt;expression2&gt;, it executes the blocks contained between \$DEFAULT and \$ENDSWITCH.</p> <p>E.g.:</p> <pre> \$SWITCH [P1=P2/P3] \$CASE 10 ... (executes this section if P1=10) \$BREAK \$CASE 20 ... (executes this section if P1=20) \$BREAK \$CASE 40 ... (executes this section if P1=40) \$BREAK \$DEFAULT ... (executes this section for other P1 values) \$ENDSWITCH           </pre>

Instruction	Meaning
<b>\$FOR</b> <b>\$ENDFOR</b> <b>\$BREAK</b> <b>\$CONTINUE</b>	Block repetition FOR.  <b>\$FOR &lt;N&gt;=&lt;expr1&gt;,&lt;expr2&gt;,&lt;expr3&gt;</b> <b>\$ENDFOR</b> <b>\$BREAK</b> <b>\$CONTINUE</b>

Note: when executing this instruction, <n> takes the value of <expr1> and it changes its value up to the value of <expr2>, in steps indicated by <expr3>. At each step, it executes the blocks contained between \$FOR and \$ENDFOR.

The \$BREAK instruction lets ending block repetition even if the stop condition is not met. The execution of the program will continue at the block after \$ENDFOR.

The \$CONTINUE instruction starts the next repetition even when the current one has not finished. The blocks programmed after \$CONTINUE up to \$ENDFOR will be ignored in this repetition.

E.g.:

\$FOR P1=10,20,2

... (executes this section starting with P1=10,

... (increases by 2 for each repetition, until

... (reaches the value of 20, then ends

\$ENDFOR

Instruction	Meaning
<b>\$WHILE</b> <b>\$ENDWHILE</b> <b>\$BREAK</b> <b>\$CONTINUE</b>	Conditional block repetition WHILE.  <b>\$WHILE &lt;condition&gt;</b> <b>\$ENDWHILE</b> <b>\$BREAK</b> <b>\$CONTINUE</b>  Note: while the condition is true, it executes the blocks contained between \$WHILE and \$ENDWHILE. The condition is analyzed at the beginning of each new repetition. The \$BREAK instruction lets ending block repetition even if the stop condition is not met. The execution of the program will continue at the block after \$ENDWHILE. The \$CONTINUE instruction starts the next repetition even when the current one has not finished. The blocks programmed after \$CONTINUE up to \$ENDWHILE will be ignored in this repetition. Es.: P1=0 \$WHILE P1<=10 P1=P1+1 ... (executes this program section until P1 is greater than 10, then it finishes) \$ENDWHILE ...

Instruction	Meaning
<b>\$DO</b> <b>\$ENDDO</b> <b>\$BREAK</b> <b>\$CONTINUE</b>	<p>Conditional block repetition DO.</p> <p><b>\$DO</b>  <b>\$ENDDO &lt;condition&gt;</b>  <b>\$BREAK</b>  <b>\$CONTINUE</b></p> <p>Note: While the condition is true, it repeats the execution of the blocks contained between \$DO and \$ENDDO. The condition is analyzed at the end of each repetition, therefore the group of blocks is executed at least once.</p> <p>The \$BREAK instruction lets ending block repetition even if the stop condition is not met. The execution of the program continues at the block after \$ENDDO.</p> <p>The \$CONTINUE instruction starts the next repetition even when the current one has not finished. The blocks programmed after \$CONTINUE up to \$ENDDO will be ignored in this repetition.</p> <p>E.g.:</p> <p>P1=0          \$DO          P1=P1+1          ... (executes this program section until P1 is greater than 10, then it finishes)          \$ENDDO P1&lt;=10          ...</p>

Instruction	Meaning
<b>L</b>	<p>Call to a global subroutine.</p> <p><b>L Name</b></p> <p>To call a subroutine that is found in the folder from which it is called.</p> <p><b>L C:\Cnc8070\Users\PATH\Nome</b></p> <p>To call a subroutine that is not found in the folder from which it is called.</p> <p>Note: the name of a called subroutine may also be a number.</p> <p>It must end with the instructions M17, M29 or #RET.</p>
<b>LL</b>	<p>Call to a local subroutine.</p> <p><b>LL Name</b></p> <p>Note: the name of a called subroutine may also be a number.</p> <p>The subroutine must start with the instruction.</p> <p><b>%L Name</b></p> <p>It must end with the instructions M17, M29 or #RET.</p> <p>The subroutine must be written inside the header of the main program; in this case, the main program must start with the character %. (The main program is identified with the character % and followed by other characters, provided that the first is not L).</p>
<b>#CALL</b>	Call to a local and global subroutine.
<b>#PCALL</b>	Call to a local and global subroutine initializing parameters.
<b>#MCALL</b>	Call to a modal local and global subroutine initializing parameters.
	A subroutine stops being modal with the instruction #MDOFF.

## Programming instructions

Instruction	Meaning
<b>#ABORT</b>	Abort the execution of the program and resume it in another block or program.
<b>#ACS</b>	Fixture coordinate system.
<b>#ANGAX OFF</b>	Turn angular transformation off.
<b>#ANGAX ON</b>	Turn angular transformation on.
<b>#ANGAX SUSP</b>	Freeze angular transformation.
<b>#ASPLINE ENDTANG</b>	Akima splines. Type of final tangent.
<b>#ASPLINE MODE</b>	Akima splines. Selection of tangent type.
<b>#ASPLINE STARTTANG</b>	Akima splines. Type of starting tangent.
<b>#AXIS</b>	Axis upon which the additive manual intervention is applied.
<b>#CALL AX</b>	Add a new axis to the configuration of the channel.
<b>#CALL SP</b>	Add a spindle to the configuration of the channel.
<b>#CAM ON</b>	Activate the electronic cam (real coordinates).
<b>#CAM OFF</b>	Cancel the electronic cam.
<b>#CAX</b>	Axis C. Activating the spindle as C axis.
<b>#CD OFF</b>	Cancel collision detection.
<b>#CD ON</b>	Activating collision detection.
<b>#CLEAR</b>	Channels. It clears the synchronism marks of the channel.
<b>#CLOSE</b>	Close a file already created or opened using the instruction #OPEN.
<b>#CONTJOG</b>	Manual intervention. Feedrate in continuous jog.
<b>#COMMENT BEGIN</b>	Beginning of comment.
<b>#COMMENT END</b>	End of comment.
<b>#CS</b>	Machining coordinate system.
<b>#CSROT ON</b>	Activate tool orientation in the part coordinate system.

Instruction	Meaning
#CSROT OFF	Cancel tool orientation in the part coordinate system.
#CYL	"C" axis. Machining of the turning side of the part.
#DEF	Macros. Define a macro.
#DEFROT	How to manage the discontinuities in the orientation of rotary axes.
#DELETE	It initializes the global user variables.
#DFHOLD	Disable the feed-hold signal.
#DGWZ	Define the size of the graphics area.
#DSBLK	Cancel the single-block treatment.
#DSTOP	Disable the cycle stop signal.
#EFHOLD	Disable the feed-hold signal.
#ERROR	Display an error on the screen.
#ESBLK	Activate the single-block treatment.
#ESTOP	Enable the cycle stop signal.
#EXBLK	It executes a block in the indicated channel.
#EXEC	It executes a program in the indicated channel.
#FACE	"C" axis. Machining on the face of the part.
#FLUSH	Interrupt block preparation.
#FOLLOW OFF	Independent axis. End the synchronization movement.
#FOLLOW ON	Independent axis. Begin the synchronization movement (real coordinates).
#FREE AX	Remove an axis from the configuration of the channel.
#FREE SP	Remove a spindle from the configuration of the channel.
#HSC OFF	Cancel the HSC mode.
#HSC ON	Activate the HSC mode. Optimizing the contouring error.
#HSC ON [FAST]	Activate the HSC mode. Optimizing the machining speed.
#INCJOG	Manual intervention. Feedrate in incremental jog.
#INIT MACROTAB	Macros. Initialize the table of macros.
#KIN ID	Select a kinematics.

Instruction	Meaning
#KINORG	Transform the current part zero considering the position of the table kinematics.
#LINK	Activate the electronic coupling (slaving) of axes.
#MASTER	Selecting the master spindle of the channel.
#MCS	Program a movement referred to machine zero.
#MCS OFF	Cancel the machine coordinate system.
#MCS ON	Activate the machine coordinate system.
#MDOFF	Turning the subroutine into non-modal.
#MEET	Channels. Activate the mark in the indicated channel.
#MOVE	Independent axis. Positioning move.
#MPG	Manual intervention. Resolution of the handwheels.
#MSG	Display a message on the screen.
#OPEN	Create a new file or open an existing file.
#PARK	Park an axis or spindle.
#PATH	Define the folder that contains the global subroutines.
#POLY	Polynomial interpolation.
#PROBE 1	(·M· model). Tool calibration (dimensions and wear). (·T· model). Tool calibration.
#PROBE 2	(·M· model). Probe calibration. (·T· model). Tabletop probe calibration.
#PROBE 3	(·M· model). Surfacing measuring. (·T· model). Part measurement along the ordinate axis.
#PROBE 4	(·M· model). Outside corner measuring. (·T· model). Part measurement along the abscissa axis.
#PROBE 5	(·M· model). Inside corner measuring.
#PROBE 6	(·M· model). Angle measurement on the abscissa axis.
#PROBE 7	(·M· model). Outside corner and angle measurement.
#PROBE 8	(·M· model). Hole measuring.



Instruction	Meaning
#PROBE 9	(·M· model). Circular boss measuring.
#PROBE 10	(·M· model). Rectangular part centering.
#PROBE 11	(·M· model). Circular part centering.
#PROBE 12	(·M· model). Tabletop probe calibration.
#RENAME AX	Rename the axes of the channel.
#RENAME SP	Rename the spindles of the channel.
#REPOS	Repositioning axes and spindles from an OEM subroutine.
#RET	End of a global or local subroutine.
#ROTATEMZ	Positioning a turret magazine.
#ROUNDPAR	Type of corner rounding.
#RPT	Repeat a group of blocks.
#RTCP	RTCP transformation.
#SCALE	Scaling factor.
#SELECT ORI	Select onto which rotary axes of the kinematics the tool orientation is calculated for a given direction on the work piece (part).
#SELECT PROBE	Probe selection.
#SERVO ON	Activate the closed loop mode.
#SERVO OFF	Activate the open loop mode.
#SET AX	Set a new axes configuration.
#SET OFFSET	Manual intervention. Moving limits.
#SET SP	Set a new spindle configuration.
#SIGNAL	Channels. Activate the mark in its own channel.
#SLOPE	Acceleration control.
#SPLINE OFF	Akima splines. Cancels spline adaptation.
#SPLINE ON	Akima splines. Activate spline adaptation.
#SWTOUT ON	Activate synchronized switching.
#SWTOUT OFF	Cancel synchronized switching.
#SYNC	Spindle synchronization based on the real (actual) coordinate.
#SYNC POS	Manual intervention. Coordinate synchronization.
#TANGCTRL OFF	Cancel tangential control.
#TANGCTRL ON	Activate tangential control.

Instruction	Meaning
#TANGCTRL SUSP	Freeze tangential control.
#TANGFEED RMIN	Minimum radius for applying constant tangential feedrate.
#TCAM ON	Activate the electronic cam (theoretical coordinates).
#TFOLLOW ON	Independent axis. Begin the synchronization movement (theoretical coordinates).
#TIME	Dwell.
#TLC	Correct the implicit tool length compensation of the program.
#TOOL AX	Select the longitudinal axis of the tool.
#TOOL ORI	Tool perpendicular to the inclined plane.
#TSYNC	Spindle synchronization based on the theoretical coordinate.
#UNLINK	Cancel the electronic coupling (slaving) of axes.
#UNPARK	Unpark an axis or spindle.
#UNSYNC	Cancel the spindle synchronization.
#VIRTAX ON	Activate the virtual tool axis.
#VIRTAX OFF	Cancel the virtual tool axis.
#WAIT	Channels. It waits for a mark to be activated in the indicated channel.
#WAIT FOR	Wait for an event before resuming execution.
#WARNING	Display a warning on the screen.
#WARNINGST OP	Display a warning on the screen and interrupt the program.
#WRITE	Edit a block in a file already created or opened using the instruction #OPEN.

## Operators, mathematical functions, logic functions and variables

### Instruction    Meaning

#### Arithmetic operators.

+	Add
-	Subtract / Change sign
*	Multiply
/	Division
+=	Compounded addition
-=	Compounded subtraction
*=	Compounded multiplication
/=	Compounded division
MOD	Module or remainder of a division
**	Exponent

#### Trigonometric functions.

SIN[...]	Sine
COS[...]	Cosine
TAN[...]	Tangent
ASIN[...]	Arc-sine
ACOS[...]	Arc-cosine
ATAN[...]	Arc-tangent
ARG[...]	Arctangent y/x

#### Mathematical functions.

ABS[...]	Absolute value
SQR[...]	Square function
SQRT[...]	Square root
LOG[...]	Decimal logarithm
LN[...]	Neperian logarithm
EXP[...]	"e" function
DEXP[...]	Decimal exponent

**Instruction    Meaning**

**Other functions.**

INT[...]	Returns the integer
FRACT[...]	Returns decimal portion
ROUND[...]	Rounds up or down to the nearest integer
FUP[...]	Returns the integer plus one. If the number is an integer, it returns it )
EXIST[...]	It checks whether the selected variable or parameter exists or not

**Relational operators.**

==	Equal to
!=	Different from, other than
>	Greater than
<	Smaller than
>=	Greater than or equal to
<=	Smaller than or equal to
&	Binary AND
	Binary OR
^	Exclusive OR (XOR)
INV[...]	Inverse

**Logic operators.**

*	Logic AND
+	Logic OR

**Boolean constants.**

TRUE	True
FALSE	Not true

**List of auxiliary (miscellaneous) M functions.**

M00:	Program stop
M01:	Conditional program stop
M02/M30:	End of program
M03:	Start the spindle clockwise
M04:	Start the spindle counterclockwise
M05:	Spindle stop
M06:	Tool change
M17/M29:	End of a global or local subroutine
M19:	Spindle orientation
M41-M44:	Selects gear

**Instruction    Meaning****CNC variable prefix.**

V.A.name	Axis and/or spindle variables
V.C.name	Canned cycle or subroutine calling parameters
V.E.name	Interface related variables
V.G.name	General variables
V.MPA.name	Variables related to axis and/or spindle machine parameters
V.MPG.name	Variables related to general machine parameters
V.MPK.name	Variables related to kinematic machine parameters
V.MPM.name	Variables related to machine parameters for M functions
V.MPMAN.name	Variables related to machine parameters for JOG mode
V.MTB.name	Variables related to OEM machine parameters
V.P.name	User local variables
V.PLC.name	PLC related variables
V.S.name	User global variables
V.SP.name	Spindle related variables
V.TM.name	Variables related to tools or tool magazines

**Instruction    Meaning**
**Definition of the type of turning tool.**

V.TM.DSUBTYPET[tl][ofd]=11:	definition of diamond shaped cutter for turning
V.TM.DSUBTYPET[tl][ofd]=12:	definition of square cutter for turning, grooving or cut-off.
V.TM.DSUBTYPET[tl][ofd]=13:	definition of round cutter for turning
V.TM.DSUBTYPET[tl][ofd]=5:	definition of thread cutter

**Definition of the lathe type:**

V.TM.TURNCONFIGT[tl][ofd]=0:	parallel lathe Z+ X+
V.TM.TURNCONFIGT[tl][ofd]=1:	parallel lathe Z+ X-
V.TM.TURNCONFIGT[tl][ofd]=2:	parallel lathe Z- X+
V.TM.TURNCONFIGT[tl][ofd]=3:	parallel lathe Z- X-
V.TM.TURNCONFIGT[tl][ofd]=4:	vertical lathe X- Z+
V.TM.TURNCONFIGT[tl][ofd]=5:	vertical lathe X+ Z+
V.TM.TURNCONFIGT[tl][ofd]=6:	vertical lathe X- Z-
V.TM.TURNCONFIGT[tl][ofd]=7:	vertical lathe X+ Z-

**Tool orientation**

(example for parallel lathe Z+ X+)

V.TM.LOCODET[tl][ofd]=0:	virtual point coincides with tool center
V.TM.LOCODET[tl][ofd]=1:	virtual point. towards Z+ X-
V.TM.LOCODET[tl][ofd]=2:	virtual point towards X-
V.TM.LOCODET[tl][ofd]=3:	virtual point towards Z- X-
V.TM.LOCODET[tl][ofd]=4:	virtual point towards Z
V.TM.LOCODET[tl][ofd]=5:	virtual point towards Z- X+
V.TM.LOCODET[tl][ofd]=6:	virtual point towards X+
V.TM.LOCODET[tl][ofd]=7:	virtual point towards Z+ X+
V.TM.LOCODET[tl][ofd]=8:	virtual point towards Z+
V.TM.LOCODET[tl][ofd]=9:	virtual point coincides with tool center
V.TM.TOTIPRT[tl][ofd]=	tool tip radius
V.TM.TOWTIPRT[tl][ofd]=	tool tip radius wear
V.TM.NOSEAT[tl][ofd]=	cutter angle
V.TM.NOSEWT[tl][ofd]=	cutter width
V.TM.CUTAT[tl][ofd]=	tool cutting angle

**Instruction    Meaning****Definition of the milling tool type.**

V.TM.DSUBTYPET[tl][ofd]=1:	Flat endmill
V.TM.DSUBTYPET[tl][ofd]=2:	Ball endmill
V.TM.DSUBTYPET[tl][ofd]=3:	Toric endmill
V.TM.DSUBTYPET[tl][ofd]=4:	Tap
V.TM.DSUBTYPET[tl][ofd]=5:	Thread cutter
V.TM.DSUBTYPET[tl][ofd]=6:	Disk endmill
V.TM.DSUBTYPET[tl][ofd]=7:	Drill bit
V.TM.DSUBTYPET[tl][ofd]=8:	Surface milling endmill
V.TM.DSUBTYPET[tl][ofd]=9:	Reamer
V.TM.DSUBTYPET[tl][ofd]=10:	Boring tool
V.TM.DSUBTYPET[tl][ofd]=14:	Measuring probe (milling machine)
V.TM.TORT[tl][ofd]=	definition of the tool radius
V.TM.TOIT[tl][ofd]=	definition of the tool radius wear
V.TM.TOLT[tl][ofd]=	definition of the tool length
V.TM.TOKT[tl][ofd]=	definition of tool length wear
V.TM.TOANT[tl][ofd]=	cutter entry angle
V.TM.TOTIPRT[tl][ofd]=	toric or spherical tool tip radius
V.TM.TOWTIPRT[tl][ofd]=	toric or spherical tool radius wear
V.TM.TOCUTLT[tl][ofd]=	tool cutting length

**Instruction    Meaning**
**Example of how to use CNC variables.**

P1=V.A.ORG[10].Y	Reads P1 for the zero offset 10 on the Y axis
P1=V.G.EXTORG	Reads P1 for the active zero offset number
P1=V.G.TOOL	Reads P1 for the active tool number
P1=V.SP.PRGS	Reads P1 for the programmed S
P1=V.G.PRGF	Reads P1 for the programmed F
P1=V.A.ATIPPOS.Y	Reads P1 for the current Y axis coordinate
Reads P1 for tool radius T10, corrector 1	
P1=V.TM.TORT[10][1]+ V.TM.TOIT[10][1]	
Reads P1 for the tool length T10, corrector 1	
P1=V.TM.TOLT[10][1]+ V.TM.TOKT[10][1]	
Writes the tool radius T10 equal to 5.01 on the tool table, and cancels the radius wear	
V.TM.TORT[10][1]=5.01	
V.TM.TOIT[10][1]=0	
Writes the tool length T10 equal to 89.3 on the tool table, and cancels the length wear	
V.TM.TOLT[10][1]=89.3	
V.TM.TOKT[10][1]=0	







FAGOR AUTOMATION

**Fagor Automation S.Coop**

Bº San Andrés, 19

20500 Arrasate - Mondragón

Gipuzkoa (SPAIN)

Tel.: +34 943 039 800

Fax: +34 943 791 712

E-mail: [info@fagorautomation.es](mailto:info@fagorautomation.es)

[www.fagorautomation.com](http://www.fagorautomation.com)



Fagor Automation está acreditado por el  
Certificado de Empresa ISO 9001 y el  
marcado CE para todos sus productos.