CNCelite

8058/8060
8065

Probing (·M· model).

Ref. 2106

FAGOR

FAGOR AUTOMATION

TRANSLATION OF THE ORIGINAL MANUAL

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

MACHINE SAFETY

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

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

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

HARDWARE EXPANSIONS

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

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

COMPUTER VIRUSES

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

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

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

DUAL-USE PRODUCTS

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

FAGOR AUTOMATION

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

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

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

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

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

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

INDEX

About this manual. 5
About the product. 7
Declaration of CE conformity and warranty conditions. 13
Safety conditions. 15
Returning conditions. 19
CNC maintenance. 21
New features. 23

CHAPTER 1 PREVIOUS NOTIONS ABOUT THE PROBE.

1.1 Activate the probe. 26
1.2 Behavior of the feedrate in probing movements. 28
1.3 First and last subroutines of the probing cycles. 29
1.3.1 Subroutines supplied by Fagor. 30
1.4 Safe probing cycles. 32

CHAPTER 2 PROBING.

2.1 G100/G103. Probing. 33
2.2 G101/G102. Include/exclude the measuring error in the theoretical coordinate. 36
2.3 G104. Probe movement up to the programmed position. 39
2.4 Properties of measurement related variables. 40

CHAPTER 3 CANNED CYCLES. ISO CODED PROGRAMMING.

3.1 #PROBE 1. Tool calibration (dimensions and wear). 43
3.1.1 Programming the cycle. 45
3.1.2 Basic operation. 49
3.2 #PROBE 2. Probe calibration 50
3.2.1 Programming the cycle. 51
3.2.2 Basic operation. 52
3.3 #PROBE 3. Surfacing measuring. 53
3.3.1 Programming the cycle. 54
3.3.2 Basic operation. 56
3.4 #PROBE 4. Outside corner measuring. 57
3.4.1 Programming the cycle. 58
3.4.2 Basic operation. 59
3.5 #PROBE 5. Inside corner measuring. 60
3.5.1 Programming the cycle. 61
3.5.2 Basic operation. 62
3.6 #PROBE 6. Measuring the angle with the abscissa axis. 63
3.6.1 Programming the cycle. 64
3.6.2 Basic operation. 65
3.7 #PROBE 7. Outside corner and angle measuring. 66
3.7.1 Programming the cycle. 67
3.7.2 Basic operation. 68
3.8 #PROBE 8. Hole measuring. 69
3.8.1 Programming the cycle. 70
3.8.2 Basic operation. 71
3.9 #PROBE 9. Circular boss measuring. 72
3.9.1 Programming the cycle. 73
3.9.2 Basic operation. 74
3.10 #PROBE 10. Rectangular part centering. 76
3.10.1 Programming the cycle. 77
3.10.2 Basic operation. 79
3.11 #PROBE 11. Circular part centering. 81
3.11.1 Programming the cycle. 82
3.11.2 Basic operation. 84
3.12 #PROBE 12. Tabletop probe calibration. 86
3.12.1 Programming the cycle. 87
3.12.2 Basic operation. 88
3.13 Check the data of the canned cycles (variables). 91

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2106

CHAPTER 4 CANNED CYCLES. CYCLE EDITOR.

4.1 How to define the data of the editor. 95
4.2 Tool calibration (dimensions and wear) 96
4.2.1 Programming the cycle. 98
4.2.2 Basic operation. 101
4.3 Measurement probe calibration. 102
4.3.1 Programming the cycle. 104
4.3.2 Basic operation. 106
4.4 Tabletop probe calibration. 107
4.4.1 Programming the cycle. 108
4.4.2 Basic operation. 110
4.5 Surfacing measuring. 113
4.5.1 Programming the cycle. 114
4.5.2 Basic operation. 117
4.6 Outside corner measuring. 118
4.6.1 Programming the cycle. 119
4.6.2 Basic operation. 121
4.7 Inside corner measuring. 122
4.7.1 Programming the cycle. 123
4.7.2 Basic operation. 125
4.8 Measuring the angle with the abscissa axis. 126
4.8.1 Programming the cycle. 127
4.8.2 Basic operation. 128
4.9 Outside corner and angle measuring. 129
4.9.1 Programming the cycle. 130
4.9.2 Basic operation. 132
4.10 Hole measuring. 134
4.10.1 Programming the cycle. 135
4.10.2 Basic operation. 137
4.11 Circular boss measuring. 138
4.11.1 Programming the cycle. 139
4.11.2 Basic operation. 141
4.12 Rectangular or circular part/pocket centering. 143
4.12.1 Programming the cycle. 144
4.12.2 Basic operation. 148
4.13 Simulating a cycle from the editor. 150

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2106

ABOUT THIS MANUAL.

Title.	Probing (·M· model).
Models.	CNCelite 8058 8060 8065
Type of documentation.	End user manual. This manual describes how to operate the probe and the probing canned cycles, ISO and conversational, for the M model. **Remarks.** Always use the manual reference associated with the software version or a later manual reference. You can download the latest manual reference from the download section on our website. **Limitations.** The availability of some of the features described in this manual are dependent on the acquired software options. Moreover, the machine manufacturer (OEM) customizes the CNC performance of each machine using the machine parameters and the PLC. Because of this, the manual may describe features that are not available for the CNC or the machine. Consult the machine manufacturer for the available features.
Electronic document.	man_elite_58_60_65_m_prb.pdf. Manual available from the download section of our website.
Language.	English [EN]. Refer to our website, download area, the languages available for each manual.
Date of publication.	June, 2021
Manual reference	Ref. 2106
Associated version.	v2.00
Responsibility exemption.	The information described in this manual may be subject to changes due to technical modifications. Fagor Automation reserves the right to change the contents of this manual without prior notice.
Trademarks.	This manual may contain third-party trademarks or trade names, however, they do not have them associated ® or ™ symbols. All the trade marks appearing in the manual belong to the corresponding owners. The use of these marks by third parties for their own purpose could violate the rights of the owners.
Website / Email.	http://www.fagorautomation.com Email: info@fagorautomation.es

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2106

FAGOR AUTOMATION

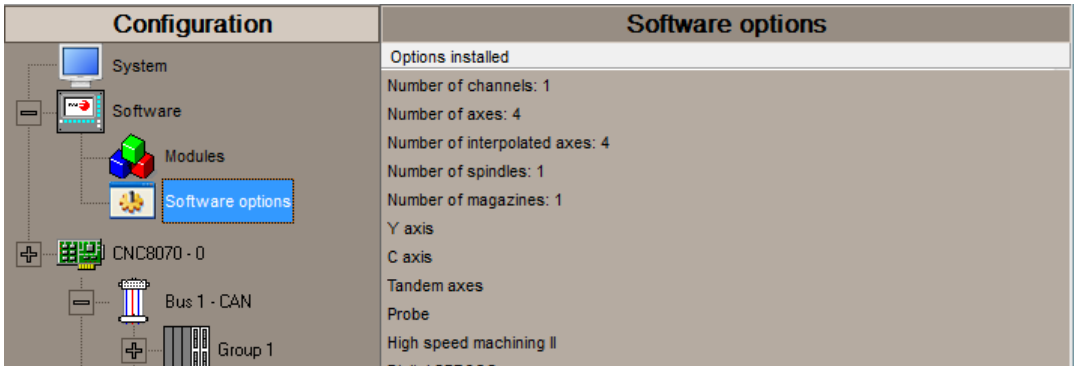
CNCelite
8058 8060
8065

REF. 2106

About the product.

SOFTWARE OPTIONS.

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



Software option	Description.
SOFT ADDIT AXES	Option to add axes to the default configuration.
SOFT ADDIT SPINDLES	Option to add spindles to the default configuration.
SOFT ADDIT TOOL MAGAZ	Option to add magazines to the default configuration.
SOFT ADDIT CHANNELS	Option to add channels to the default configuration.
SOFT 4 AXES INTERPOLATION LIMIT	Limited to 4 interpolated axes.
SOFT DIGITAL SERCOS	Option for a Sercos digital bus.
SOFT THIRD PARTY DRIVES	Option to use EtherCAT third party drives.
SOFT THIRD PARTY I/Os	Option to use third party I/O modules.
SOFT OPEN SYSTEM	Option for open systems. The CNC is a closed system that offers all the features needed to machine parts. Nevertheless, at times there are some customers who use third-party applications to take measurements, perform statistics or other tasks apart from machining a part. This feature must be active when installing this type of application, even if they are Office files. Once the application has been installed, it is recommended to close the CNC in order to prevent the operators from installing other kinds of applications that could slow the system down and affect the machining operations.

Software option	Description.
SOFT i4.0 CONNECTIVITY PACK	Options for Industry 4.0 connectivity. This option provides various data exchange standards (for example, OPC UA), which allows the CNC (and therefore the machine tool) to be integrated into a data acquisition network or into a MES or SCADA system.
SOFT EDIT/SIMUL	Option to enable edisimu mode (edition and simulation) on the CNC, which can edit, modify and simulate part programs.
SOFT DUAL-PURPOSE (M-T)	Option to enable the dual-purpose machine, which allows milling and turning cycles. On Y-axis lathes, this option allows for pockets, bosses and even irregular pockets with islands to be made during milling cycles. On a C-axis mill, this option allows turning cycles to be used.
SOFT TOOL RADIUS COMP	Option to enable radius compensation. This compensation programs the contour to be machined based on part dimensions without taking into account the dimensions of the tool that will be used later on. This avoids having to calculate and define the tool paths based on the tool radius.
SOFT PROFILE EDITOR	Option to enable the profile editor in edisimu mode and in the cycle editor. This editor can graphically, and in a guided way, define rectangular, circular profiles or any profile made up of straight and circular sections an it can also import dxf files. After defining the profile, the CNC generates the required blocks and add them to the program.
SOFT HD GRAPHICS In a multi-channel system, this feature requires the MP-PLUS (83700201) processor.	High definition solid 3D graphics for the execution and simulation of part-programs and canned cycles of the editor. During machining, the HD graphics display, in real time, the tool removing the material from the part, allowing the condition of the part to be seen at all times. These graphics are required for the collision control (FCAS).
SOFT IIP CONVERSATIONAL	The IIP (Interactive Icon-based Pages) mode, or conversational mode, works with the CNC in a graphical and guided way based on predefined cycles. There is no need to work with part programs, have any previous programming knowledge or be familiar with Fagor CNCs. Working in conversational mode is easier than in ISO mode, as it ensures proper data entry and minimizes the number of operations to be defined.
SOFT RTCP This feature requires the MP-PLUS (83700201) processor.	Option to enable dynamic RTCP (Rotating Tool Center Point) required to machine with 4, 5 and 6 axis kinematics; for example, angular and orthogonal spindles, tilting tables, etc. The RTCP orientation of the tool may be changed without modifying the position occupied by the tool tip on the part.
SOFT C AXIS	Option to enable C-axis kinematics and associated canned cycles. The machine parameters of each axis or spindle indicate whether it can operate as a C axis or not. For this reason, it is not necessary to add specific axes to the configuration.
SOFT Y AXIS	Option to enable lathe Y-axis kinematics and associated canned cycles.
SOFT TANDEM AXES	Option to enable tandem axle control. A tandem axis consists of two motors mechanically coupled to each other forming a single transmission system (axis or spindle). A tandem axis helps provide the necessary torque to move an axis when a single motor is not capable of supplying enough torque to do it. When activating this feature, it should be kept in mind that for each tandem axis of the machine, another axis must be added to the entire configuration. For example, on a large 3-axis lathe (X Z and tailstock), if the tailstock is a tandem axis, the final purchase order for the machine must indicate 4 axes.

Software option	Description.
SOFT SYNCHRONISM	Option to enable the synchronization of paired axes and spindles, in speed or position, and through a given ratio.
SOFT KINEMATIC CALIBRATION	Option to enable tool calibration. For the first time, this kinematics calibration allows for the kinematics offsets to be calculated using various approximate data and, also, from time to time to correct any possible deviations caused by day-to-day machining operations.
SOFT 60 HSSA I MACHINING SYSTEM	Option to enable the HSSA-I (High Speed Surface Accuracy) algorithm for high speed machining (HSC). This new HSSA algorithm allows for high speed machining optimization, where higher cutting speeds, smoother contours, a better surface finishing and greater precision are achieved.
SOFT HSSA II MACHINING SYSTEM	Option to enable the HSSA-II (High Speed Surface Accuracy) algorithm for high speed machining (HSC). This new HSSA algorithm allows for high speed machining optimization, where higher cutting speeds, smoother contours, a better surface finishing and greater precision are achieved. The HSSA-II algorithm has the following advantages compared to the HSSA-I algorithm. • Advanced algorithm for point preprocessing in real time. • Extended curvature algorithm with dynamic limitations. Improved acceleration and jerk control. • Greater number of pre-processed points. • Filters to smooth out the dynamic machine behavior.
SOFT TANGENTIAL CONTROL	Option to enable tangential control. "Tangential Control" maintains a rotary axis always in the same orientation with respect to the programmed tool path. The machining path is defined on the axes of the active plane and the CNC maintains the orientation of the rotary axis along the entire tool path.
SOFT PROBE	Option to enable functions G100, G103 and G104 (for probe movements) and probe canned cycles (which help to measure part surfaces and to calibrate tools). For the laser model, it only activates the non-cycle function G100. The CNC may have two probes; usually a tabletop probe to calibrate tools and a measuring probe to measure the part.
SOFT FVC STANDARD **SOFT FVC UP TO 10m3** **SOFT FVC MORE TO 10m3**	Options to enable volumetric compensation. The precision of the parts is limited by the machine manufacturing tolerances, wear, the effect of temperature, etc., especially on 5-axis machines. Volumetric compensation corrects these geometric errors to a larger extent, thus improving the precision of the positioning. The volume to be compensated is defined by a point cloud and for each point the error to be corrected is measured. When mapping the total work volume of the machine, the CNC knows the exact position of the tool at all times. There are 3 options, which depend on the size of the machine. • FVC STANDARD: Compensation for 15625 points (maximum 1000 points per axis). Quick calibration (time), but less precise than the other two, but sufficient for the desired tolerances. • FVC UP TO 10m3: Volume compensation up to 10 m³. More accurate than FVC STANDARD, but requires a more accurate calibration using a Tracer or Tracker laser). • FVC MORE TO 10m3: Volume compensation greater than 10 m³. More accurate than FVC STANDARD, but requires a more accurate calibration using a Tracer or Tracker laser.

Software option	Description.
SOFT CONV USER CYCLES	Option to enable user conversational cycles. The user and the OEM can add their own canned cycles (user cycles) using the FGUIM application that comes installed on the CNC. The application offers a guided way to define a new component and its softkey menu without having to be familiar with script languages. User cycles work in a similar way as Fagor canned cycles.
SOFT PROGTL3	Option to enable the ProGTL3 programming language (ISO language extension), allowing profiles to be programmed using a geometric language and without the need to use an external CAD system. This language can program lines and circles where the end point is defined as the intersection of 2 other sections, pockets, ruled surfaces, etc.
SOFT PPTRANS	Option to enable the program translator, which can convert programs written in other languages to Fagor ISO code.
SOFT DMC	Option to enable the DMC (Dynamic Machining Control). DMC adapts the feedrate during machining to maintain the cutting power as close as possible to ideal machining conditions.
SOFT FMC	Option to enable the FMC (Fagor Machining Calculator). The FMC application consists of a database of materials to be machined and machining operations, with an interface to choose suitable cutting conditions for these operations.
SOFT FFC	Option to enable the FFC (Fagor Feed Control). During the execution of a canned cycle of the editor, the FFC function makes it possible to replace the feedrate and speed programmed in the cycle with the active values of the execution, which are acted upon by the feed override and speed override.
SOFT 60/65/70 OPERATING TERMS	Option to enable a temporary user license for the CNC, which is valid until the date set by the OEM. While the license is valid, the CNC will be fully operational (according to the purchased software options).
SOFT FCAS	Option to enable the FCAS (Fagor Collision Avoidance System). The FCAS option, within the system limitations, monitors the automatic, MDI/MDA, manual and tool inspection movements in real time, so as to avoid collisions between the tool and the machine. The FCAS option requires that the HD graphics to be active and that there is a defined a model configuration of the machine adjusted to reality (.xca file), which includes all its moving parts.
SOFT GENERATE ISO CODE	ISO generation converts canned cycles, calls to subroutines, loops, etc. into their equivalent ISO code (G, F, S, etc functions), so the user can modify it and adapt it to his needs (eliminate unwanted movements, etc.). The CNC generates the new ISO code while simulating the program, either from the DISIMU mode or from the conversational mode.
SOFT PWM CONTROL	Option to enable PWM (Pulse - Width Modulation) control on laser machines. This feature is essential for cutting very thick sheets, where the CNC must create a series of PWM pulses to control laser power when drilling the initial point. This function is only available for Sercos bus control systems and must also use one of the two fast digital outputs available from the central unit.
SOFT GAP CONTROL	Option to enable gap control, which makes it possible to set a fixed distance between the laser nozzle and the sheet surface with the use of a sensor. The CNC compensates the difference between the distance measured by the sensor and the programmed distance with additional movements on the axis programmed for the gap.

Software option	Description.
SOFT MANUAL NESTING	Option to enable nesting in the automatic option. Nesting consists of creating a pattern on the sheet material using previously defined figures (in dxf, dwg or parametric files), so as to use most of the sheet as possible. Once the pattern has been defined, the CNC creates a program. During manual nesting, the operator distributes the parts on top of the sheet material.
SOFT AUTO NESTING	Option to enable nesting in the automatic option. Nesting consists of creating a pattern on the sheet material using previously defined figures (in dxf, dwg or parametric files), so as to use most of the sheet as possible. Once the pattern has been defined, the CNC creates a program. During automatic nesting, the application distributes the figures on the sheet material and optimizes the spaces.
SOFT DRILL CYCL OL	Option to enable ISO drilling cycles (G80, G81, G82, G83).

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2106

DECLARATION OF CE CONFORMITY AND WARRANTY CONDITIONS.

DECLARATION OF CONFORMITY

The declaration of conformity is available from the downloads section of the Fagor Automation corporate website.

https://www.fagorautomation.com/en/downloads/

Type of file: Declaration of conformity.

WARRANTY TERMS

The sales and warranty conditions are available from the downloads section of the Fagor Automation corporate website.

https://www.fagorautomation.com/en/downloads/

Type of file: General sales - warranty conditions.

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2106

SAFETY CONDITIONS.

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

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

PRECAUTIONS BEFORE CLEANING THE UNIT

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

PRECAUTIONS DURING REPAIRS

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

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

PRECAUTIONS AGAINST PERSONAL HARM

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

PRECAUTIONS AGAINST DAMAGE TO THE PRODUCT

Work environment.

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

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

Install this unit in the proper place.

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

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

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

Enclosures.

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

Avoid disturbances coming from the machine.

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

Use the proper power supply.

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

Connecting the power supply to ground.

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

Analog inputs and outputs connection.

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

Ambient conditions.

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

Central unit enclosure.

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

Power switch.

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

SAFETY SYMBOLS

Symbols that may appear in the manual.

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

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

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

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

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

Symbols that the product may carry.

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

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

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2106

RETURNING CONDITIONS.

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

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

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2106

CNC MAINTENANCE.

CLEANING

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

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

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

PRECAUTIONS BEFORE CLEANING THE UNIT

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

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

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2106

NEW FEATURES.

Manual reference:	Ref. 2106
Date of publication:	June, 2021
Associated software:	v2.00

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

List of features.	Manual.	
New CNC model. 8058.	[CHN]	[CYC-M]
	[CYC-T]	[ERR]
	[EXA-M]	[EXA-T]
	[INST]	[OPT]
	[OPT-MC]	[OPT-TC]
	[PPC]	[PPTRANS]
	[PRB-M]	[PRB-T]
	[PRG]	[RIOS]
	[RIOS-A]	[RIOS-E]
	[VAR]	

[CHN]..... Execution channels.
[CYC-M]..... Machining canned cycles (·M· model).
[CYC-T]..... Machining canned cycles (·T· model).
[ERR]..... Error solving manual.
[EXA-M] ..... Examples manual (·M· model).
[EXA-T]..... Examples manual (·T· model).
[INST]..... Installation manual.
[OPT] ..... Operating manual.
[OPT-MC]..... Operating manual (MC option).
[OPT-TC]..... Operating manual (TC option).
[PPC] ..... Panel PC.
[PPTRANS]..... Part-program translator.
[PRB-M] ..... Probing (·M· model).
[PRB-T]..... Probing (·T· model).
[PRG]..... Programming manual.
[RIOS]..... Remote modules (RIO5, RIOW, RIOR).
[RIOS-A] ..... "ABSIND" module.
[RIOS-E] ..... EtherCAT remote modules (RIOW-E Inline)
[VAR]..... CNC variables.

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2106

PREVIOUS NOTIONS ABOUT THE PROBE.

1

Number of probes in the system and active probe.

The CNC may have configured two probes, it will usually be a tabletop probe to calibrate tools and a touch probe to measure the part.

Before any probing moves, select the probe to be used. See *"1.1 Activate the probe."* on page 26.

Probe operation.

Both probes operate by levels, not by flanks.

Probing.

With function G100, it is possible to program movements that will end when the CNC receives the probe signal (when the probe makes contact). When done probing, the CNC updates the real coordinates.

With function G103, it is possible to program movements that will end when the CNC stops receiving the probe signal (when the probe no longer makes contact). When done probing, the CNC updates the real coordinates.

The G104 function prevents a G100 or G103 probe movement from finishing with the probe signal The CNC updates the coordinates with the probe signal, but without interrupting the movement which continues until the probe reaches the programmed position.

Programming the canned cycles.

The probing canned cycles may be edited in ISO code or with using the cycle editor. These cycles may be defined anywhere in the program, that is, in the main program as well as in a subroutine. ISO coded cycles can also be executed in MDI.

Probe parameter setting.

The machine manufacturer must have properly set the following machine parameters.

- General machine parameters.

  PROBE PROBEDATA PROBETYPE1 PROBETYPE2
  PRBDI1 PRBDI2 PRBPULSE1 PRBPULSE2

- General machine parameters per channel.

  PROBEDATA PRB1MAX PRB1MIN PRB2MAX
  PRB2MIN PRB3MAX PRB3MIN

- Axis machine parameters.

  PROBEAXIS PROBERANGE PROBEFEED PROBEDELAY
  PROBEDELAY2

1.1 Activate the probe.

The CNC can have configured two probes. Before any probing move, the CNC must know which is the active probe, or, which is the same, which of the two probes it must attend to. It is selected via part-program or MDI using the instruction #SELECT PROBE.

If a probing move is executed without activating the probe, it will not send any signal to the CNC when it makes contact. This can cause the probe to break because the probing move will not be stopped.

Programming.

When programming this instruction, you must define which probe is active and whether it's active high or low.

Programming format.

The programming format is the following; the list of arguments appears between curly brackets and the optional ones between angle brackets.

```
#SELECT PROBE [<{probe}><, {pulse}>]
```

{probe} Optional. Number of probe to activate.

If not programmed, the CNC uses the active probe.

{pulse} Optional. Logic level to activate probe. The CNC uses the high level with "POS" and the low level with "NEG".

If not programmed, the CNC uses the default probe activation level.

Although both parameters are optional, at least one of them must be programmed.

```
#SELECT PROBE [1]
#SELECT PROBE [NEG]
#SELECT PROBE [2, POS]
#SELECT PROBE [1, NEG]
```

Probe number. Which is probe 1 and which probe 2?

The names of the probes are set in the order they have been defined in the machine parameters. The CNC assumes as first probe the one connected to the input indicated in machine parameter PRBDI1 and as second probe the one connected to the input indicated in machine parameter PRBDI2.

Logic level to activate probe; high (5 V / 24 V) or low (0 V).

Changing the default activation level reverses the operation of functions G100 and G103. When changing the probe logic activation level, G100 makes a movement until the probe stops making contact and G103 makes a movement until the probe makes contact. Since probing canned cycles use functions G100 and G103, changing the logic activation level also changes the operation of the canned cycles accordingly.

The logic activation level indicates whether the probe operations are active high (24V or 5 V) or active low (0V) of the signal provided by the probe. Programming the logic activation level is optional because each probe has been assigned one by default.

The logic activation level of each probe by default is set in the machine parameters (parameters PRBPULSE1 for probe ·1· and PRBPULSE2 for probe ·2·) and it depends on the connection between the probe and the CNC.

Properties of the instruction and influence of reset, turning the CNC off and of the M30 function.

The instruction #SELECT PROBE is modal. The probe and the selected logic activation level stays active after an M02 or M30 and after an error or a reset. On power-up and after validating the machine parameters, the CNC activates probe ·1· and initializes the logic activation level of both probes with the values set in the machine parameters.

Knowing which is the active probe.

The CNC offers the following variable to know which is the active probe. The variable can only be read via part-program, MDI, PLC and interface.

Variable.	Meaning.
(V.)[ch].G.ACTIVPROBE	This variable indicates which one is the active probe in channel n.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2106

1.2 Behavior of the feedrate in probing movements.

The probing moves are carried out at the active feedrate, the one defined for machining. If the probing feedrate is changed, the new feedrate will be the active one for the machining moves.

The feedrate may be selected by programmed using the "F" code which remains active until another value is programmed. In the canned cycles, the feedrate may be programmed inside the parameters of the cycle.

The units depend on the active work mode; G93, G94 or G95.

G93 Machining time in seconds.

G94 Feedrate in millimeters/minute (inches/minute).

G95 Feedrate in millimeters/revolution (inches/revolution).

The active feedrate may be varied between 0% and 200% using the selector switch on the CNC's operator panel or it may be selected by program or by PLC.

Maximum probing feedrate.

The maximum probing feedrate in each axis will be limited by machine parameter PROBEFEED and this value will not be exceeded even when programming a higher value.

1.3 First and last subroutines of the probing cycles.

The subroutines provided by Fagor offer basic handling of the probes. These subroutines must be configured by the OEM.

Fagor provides the subroutine Sub_Probe_Tool_Begin.fst associated with probe input 1 and the subroutine Sub_Probe_Piece_Begin.fst associated with probe input 2.

At the beginning and end of the probing cycles, both ISO and those of the editor, the CNC executes the following subroutines. The subroutines for the tool calibration cycles and for the measuring cycles are different.

Subroutine.	Properties.
Sub_Probe_Tool_Begin.fst	The CNC executes the subroutine at the beginning of all tool calibration cycles.
Sub_Probe_Tool_End.fst	The CNC executes the subroutine at the end of all tool calibration cycles.
Sub_Probe_Piece_Begin.fst	The CNC executes the subroutine at the beginning of all part measuring cycles.
Sub_Probe_Piece_End.fst	The CNC executes the subroutine at the end of all part measuring cycles.

These subroutines are saved in the folder ..\MTB\Sub. Since they are OEM subroutines, being the CNC in USER mode, the CNC loads them into RAM memory when starting up the application. Being the CNC in SETUP mode, the CNC loads these subroutines into RAM memory the first time it executes them inside the program.

Subroutines and M functions defined in the cycle.

Up to 4 M functions may be defined in each probing cycle to be executed before the cycle and another 4 M functions to be executed afterwards. All these functions may have a subroutine associated with them.

When starting the execution, the cycle executes first the subroutines Sub_Probe_Tool_Begin.fst or Sub_Probe_Piece_Begin.fst (accordingly) and then the M-before functions with their associated subroutines.

At the end of the execution, the cycle executes firsts the M-after functions with their corresponding associated subroutines and then the subroutines Sub_Probe_Tool_End.fst or Sub_Probe_Piece_End.fst (accordingly).

1.3.1 Subroutines supplied by Fagor.

Subroutine Sub_Probe_Tool_Begin.fst supplied by Fagor (may be modified by the user).

```
#ESBLK
; Activate PROBE1 Hardware by PLC output.

; Check PROBE is READY with PLC Input from Probe Hardware.
#MSG["WAIT FOR ENABLING PROBE1"]
#WAIT FOR[V.PLC.PROBE1ENA==1]
#MSG[""]

; Select PROBE1 input for using in Probing.
#SELECT PROBE[1]
#FLUSH
$IF [V.G.PRBST1==1]
  #WARNING["PROBE1 SIGNAL TOUCHING"]
  M0
$ENDIF

; Check probe in safe mode feature is actived.
$IF [V.PLC.PROBE1MONIT==0]
  #MSG["PROBE NOT IN SAFE MODE"]
$ENDIF
#RETDSBLK

#COMMENT BEGIN
PLC signals to add in the logic:

PROBE1ENA; PROBE1 is enabled confirmation signal to CNC.
If not used in PLC, it is actived by default.

PROBE1MONIT; It activates PROBE1 in SAFE mode.
If probe is actived in no G100/3 motion, CNC will stop motion and shows an error.
If not used in PLC, it is actived by default.

#COMMENT END
```

Subroutine Sub_Probe_Tool_End.fst supplied by Fagor (may be modified by the user).

```
#ESBLK
;Deactivate PROBE1 Hardware by PLC output

#RETDSBLK
```

Subroutine Sub_Probe_Piece_Begin.fst supplied by Fagor (may be modified by the user).

```
#ESBLK
; Activate PROBE 2 Hardware by PLC output.

; Check PROBE is READY with PLC Input from Probe Hardware.
#MSG["WAIT FOR ENABLING PROBE2"]
#WAIT FOR[V.PLC.PROBE2ENA==1]
#MSG[""]

; Select PROBE 2 input for using in Probing.
#SELECT PROBE[2]
#FLUSH
$IF [V.G.PRBST2==1]
  #WARNING["PROBE2 SIGNAL TOUCHING"]
  M0
$ENDIF

; Check probe in safe mode feature is actived.
$IF [V.PLC.PROBE2MONIT==0]
  #MSG["PROBE NOT IN SAFE MODE"]
$ENDIF
#RETDSBLK

#COMMENT BEGIN
PLC signals to add in the logic:

PROBE2ENA; PROBE 2 is enabled confirmation signal to CNC.
If not used in PLC, it is actived by default.

PROBE2MONIT; It activates PROBE2 in SAFE mode.
If probe is actived in no G100/3 motion, CNC will stop motion and shows an error.
If not used in PLC, it is actived by default.

#COMMENT END
```

Subroutine Sub_Probe_Piece_End.fst supplied by Fagor (may be modified by the user).

```
#ESBLK
;Deactivate PROBE2 Hardware by PLC output

#RETDSBLK
```

1.4 Safe probing cycles.

The probe is protected against collisions in positioning and withdrawal movements, inside the probing cycles and in any movement where G100 has not been programmed. The CNC supports monitoring with RTCP and inclined planes.

Collision management is not compatible with non-stop probing processes (G104). Movements that use this type of probing must disable the safe mode of the probe (PROBE1MONIT=0 or PROBE2MONIT=0).

Loop level monitoring, controlling all the collision scenarios in any of the two probes. The CNC can monitor the probes connected to the local inputs and to the remote CAN inputs. The CNC monitors the two probe inputs at the same time.

For moves in automatic mode, if the CNC detects a collision in a positioning or withdrawal movement inside a probing cycle, or even outside of it, it stops the movement, displays the corresponding error message, opens the emergency relay and activates the _ALARM (level "0") signal, When a collision occurs while the axes are not moving, the CNC does not display an error. The CNC will only allow jogging the probe away.

For manual (jog) movements, if the CNC detects a collision, it stops the probing movement and displays the corresponding error. The CNC will only allow moving the probe away.

PLC marks.

The following PLC marks allow enabling or disabling collision monitoring for a probe in safe mode.

PROBE1ENA
PROBE2ENA

These are active marks by default. These mark indicate that the probe has is enabled. When executing a G100 or G103 command, the CNC will issue an error message if the mark of the active probe (the one selected with #SELECT PROBE) is not enabled. These marks do not limit the monitoring of the safe mode.

These marks should be tested in the subroutines Sub_Probe_Tool_Begin.fst and Sub_Probe_Piece_Begin.fst so the subroutine waits until the mark is active.

PROBE1MONIT
PROBE2MONIT

These marks are associated with the safe mode of the probe. If the mark is active, the probe is in safe mode monitoring collisions.

These marks should be tested in the subroutines Sub_Probe_Tool_Begin.fst and Sub_Probe_Piece_Begin.fst to warn, if they are deactivated, that the probe is in non-safe mode.

PROBING.

2

2.1 G100/G103. Probing.

With function G100, it is possible to program movements that will end when the CNC receives the probe signal (when the probe makes contact) or when the probe reaches the programmed position. When done probing, the CNC assumes as the theoretical position the current position of the axes involved in the movement, their real (actual) position at that instant.

With function G103, it is possible to program movements that will end when the CNC stops receiving the probe signal (when the probe stops making contact) or when the probe reaches the programmed position. When done probing, the CNC assumes as the theoretical position the current position of the axes involved in the movement, their real (actual) position at that instant.

Functions G100 and G103 do not execute the tool change to select the probe, the probe must be selected in a previous block of the program. Likewise, when using more than one probe, the probe to be used must be selected before probing.

Probing programming.

The probing movement is defined using function G100 or G103 followed by the coordinates of the probe's target point. Programming the feedrate is optional; if not programmed, these movements are carried out at the active feedrate.

Programming format.

The programming format is: Optional parameters are indicated between angle brackets.

```
G100 X..C <F>
G103 X..C <F>
```

X..C	Coordinates of the probing point.
F	Optional. Feedrate. If not programmed, the CNC uses the active feedrate.

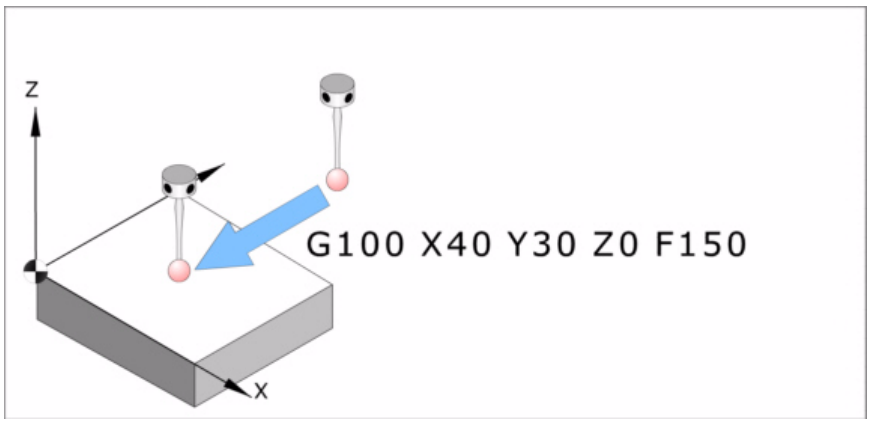
```
G100 X45.23 Y20.25 Z23.45
G100 Z50 F100
G103 X2.6 Z3 F20
G103 Z1 F20
```

Probing feedrate.

The CNC uses the same feedrate for probing and for machining. The feedrate "F" set for the probe will be the feedrate active at the CNC when done probing.

The maximum probing feedrate in each axis will be limited by machine parameter PROBEFEED and this value will not be exceeded even when programming a higher value or exceeded with the switch on the operator panel.

The active feedrate may be varied between 0% and 200% using the selector switch on the CNC's operator panel or it may be selected by program or by PLC.



Properties of the function and Influence of the reset, turning the CNC off and of the M30 function.

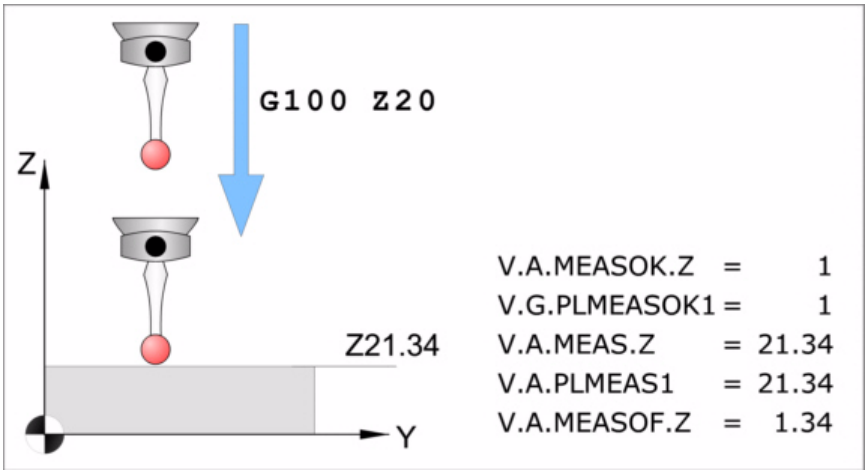
Functions G100 and G103 are not modal. After executing one of these functions, the CNC restores the function G0, G1, G2 ó G3, G33 or G63 that was active.

Updating variables after probing.

When done probing, the CNC updates the following variables. After a probing, the CNC updates all the variables of all the axes of the channel even if they were not involved in the probing movements. For the axes not involved in the probing movements, the variables that save the measured value take the value of the real position of the axis and the variables that indicate the measured error are reset to zero.

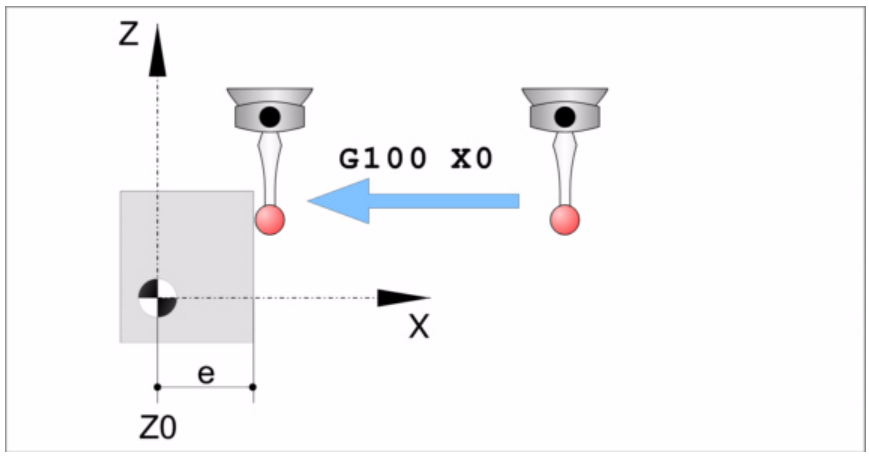
Mnemoni.	Variable.
V.G.MEASOK	The probe has made contact (G100) or stopped making contact (G103). • The variable takes the value of ·1· if the probe has made contact (G100) or has stopped making contact (G103). • The variables takes the value of ·0· if the probe reaches the programmed coordinate.
V.A.MEASOK.xn	Probing done on any axis of the channel. • The variables of the axes involved in the probing operation take the value of ·1· when the probing movement ends. • The variables of the rest of the axes take the value of ·0·. The variable keeps its value after a reset.
V.G.PLMEASOK1 V.G.PLMEASOK2 V.G.PLMEASOK3	Probing on the plane axes completed. • The variables of the axes involved in the probing operation take the value of ·1· when the probing operation ends. • The variables of the rest of the axes take the value of ·0·.
V.A.MEAS.xn	Measured value. Machine coordinates of the tool base. • The variables of the axes involved in the probing operation take the measured value. • The variables of the rest of the axes take the real position value of the axis.
V.A.ATIPMEAS.xn	Measured value. Part coordinates of the tool tip. • The variables of the axes involved in the probing operation take the measured value. • The variables of the rest of the axes take the real position value of the axis.

Mnemoni.	Variable.
V.G.PLMEAS1 V.G.PLMEAS2 V.G.PLMEAS3	Value measured on the axes of the plane. Part coordinates of the tool tip. • The variables of the axes involved in the probing operation take the measured value. • The variables of the rest of the axes take the real position value of the axis.
V.A.MEASOF.xn	Measuring error. • The variables of the axes involved in the probing operation take the measuring error (difference between the programmed coordinate and the one measured). • The variables of the rest of the axes take the value of ·0·.



2.2 G101/G102. Include/exclude the measuring error in the theoretical coordinate.

The measuring error is the difference between the programmed coordinate and the coordinate reached by the probe.



After probing, the CNC assumes the current axis position as the theoretical position. Functions G101 and G102 determine whether to consider or ignore the measuring error when updating the theoretical coordinate.

G101 Include the measuring error in the theoretical coordinate.

G102 Exclude the measuring error in the theoretical coordinate.

Influence of the reset, turning the CNC off and of the M30.

Functions G101 and G102 are modal and incompatible with each other. On power-up, after an M02 or M30 and after an EMERGENCY or a RESET, the CNC maintains the values programmed with G101.

G101 Include the measuring error in the theoretical coordinate.

When executing this function, the CNC includes the error resulting from the measurement to set the theoretical axis positions; in other words, the CNC will assume as theoretical axis position the programmed coordinate (position reached by the probe + the measuring error).

Function G101 must be executed after taking a measurement. The CNC lets program any axis of the channel in a G101 block even if it has not been involved in the previous measurement (G100/G103/G104).

The CNC lets make a measurement (G100/G103/G104) on any axis of the channel even when function G101 is active. The measurement on an axis does not change the G101 of other axes and, therefore, it does not change its variable (V.)A.MEASIN.xn.

Programming format.

To include the measuring error, program function G101 and then the axes in which to include the measuring error. For each axis, you must define how many times the measuring error is added to the coordinate. Usually, the measuring error needs to be included only once.

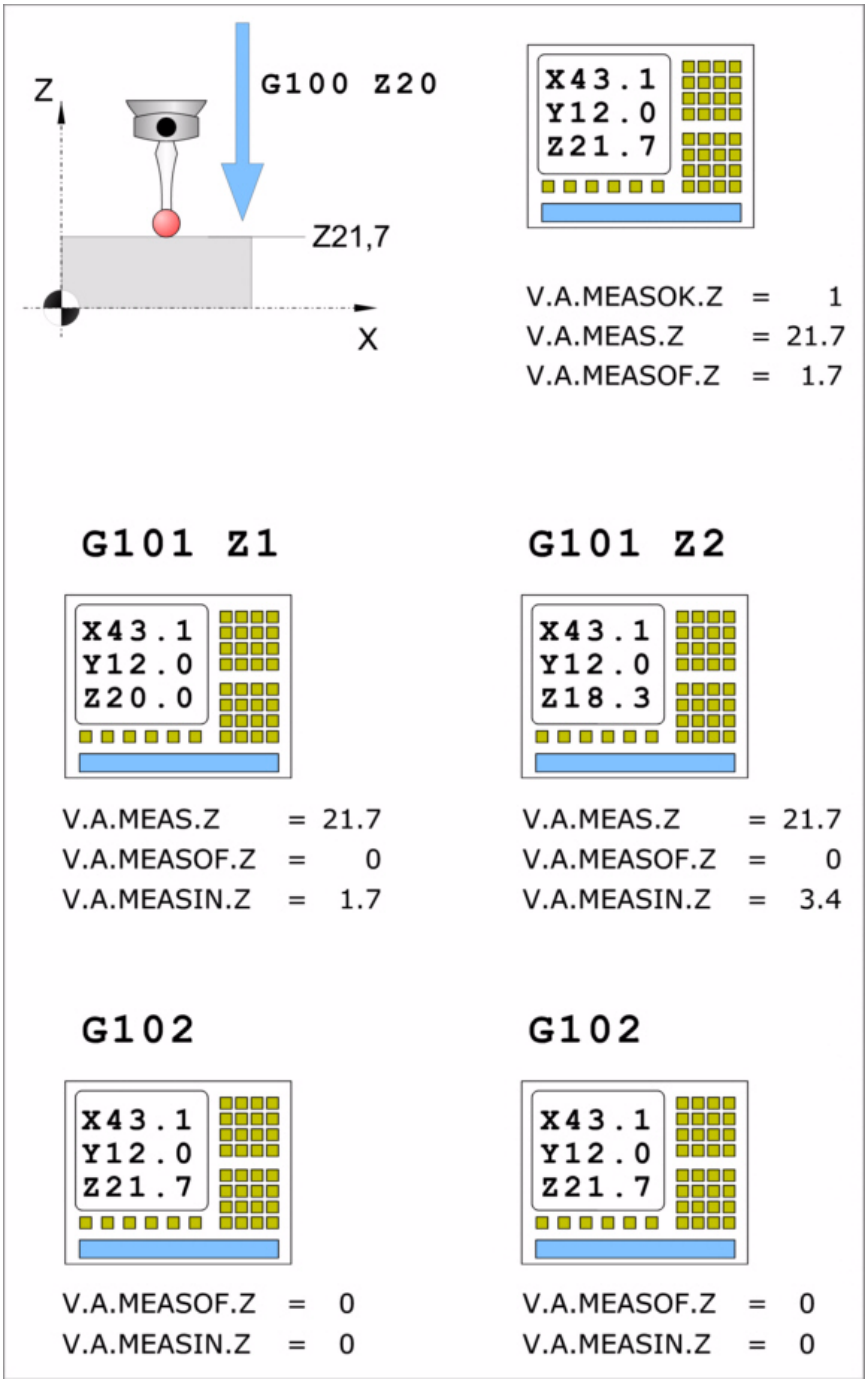
G101 X..C

X..C Axes whose theoretical coordinate includes the measuring error .

```
G101 X1 Y1 Z1
G101 X2
```

Updating the variables after executing function G101.

Variable	Value
(V.)[n].A.MEASOF.Xn	It is initialized to 0 (zero).
(V.)[n].A.MEASIN.Xn	Measuring error added to the Xn axis.



G102 Exclude the measuring error in the theoretical coordinate.

After executing this function, the CNC will ignore the error resulting from the measurement to set the theoretical position of the axes; i.e. the CNC considers the coordinate reached as theoretical coordinate.

The CNC lets program any axis of the channel in a G102 block even if it does not have a measuring offset included (G101).

Programming format.

To ignore the measuring error, program function G102 and then the axes in which to ignore it. If no axis is programmed, the CNC ignores the measuring error in all the axes.

The programming format is: Optional parameters are indicated between angle brackets.

G102 <X..C>

X..C	Optional. Axes whose theoretical coordinate does not include the measuring error .

```
G102 X Y
G102
```

Once function G102 is executed, function G101 cannot be executed again until a new measurement is taken.

Updating the variables after executing function G102.

Variable	Value
(V.)[n].A.MEASIN.Xn	It is initialized to 0 (zero).

2.3 G104. Probe movement up to the programmed position.

When programming function G104 together with G100 or G103, the CNC makes the selected probing movement, updates the coordinates when it receives the probe signal, but keeps moving the axes until they reach their programmed position.

Function G101 may be used to make the CNC assume the measuring error resulting from a G104 movement,

Probing programming.

The G104 must be programmed together with a G100 or G103 probe movement; otherwise, it will be ignored.

Programming format.

The programming format is: Optional parameters are indicated between angle brackets.

G100 G104 X..C <F>
G103 G104 X..C <F>

X..C	Coordinates of the probing point.
F	Optional. Feedrate. If not programmed, the CNC uses the active feedrate.

```
G100 G104 Z23.45
G103 G104 Z1 F20
```

Properties of the function and Influence of the reset, turning the CNC off and of the M30 function.

Function G104 is not modal; it only acts in the block where it is programmed.

2.4 Properties of measurement related variables.

For further information about the access and the use of variables, refer to the programming manual.

The following variables are read-only (R) synchronous and are evaluated while in execution. The mnemonics of the variables have generic names.

- Replace the "Xn" character by the name, logic number or index in the channel of the axis.
- Replace the "n" character with the channel number, maintaining the brackets. The first channel is identified with the number 1, "0" is not a valid number.

Mnemonic	PRG	PLC	INT	
(V.)[n].A.MEASOK.Xn	R	R	R	Probing done on the Xn axis. "0" = No "1"= Yes
(V.)[n].G.PLMEASOK1	R	—	—	Probing done on the first axis of the plane (abscissa). "0" = No "1"= Yes
(V.)[n].G.PLMEASOK2	R	—	—	Probing done on the second axis of the plane (ordinate). "0" = No "1"= Yes
(V.)[n].G.PLMEASOK3	R	—	—	Probing done on the axis perpendicular to the plane. "0" = No "1"= Yes
(V.)[n].A.MEAS.Xn	R	R	R	Value measured on the Xn axis. Machine coordinates of the tool base.
(V.)[n].A.ATIPMEAS.Xn	R	—	—	Value measured on the Xn axis. Part coordinates of the tool tip.
(V.)[n].G.PLMEAS1	R	—	—	Value measured on the first axis of the plane (abscissa). Part coordinates of the tool tip.
(V.)[n].G.PLMEAS2	R	—	—	Value measured on the second axis of the plane (ordinate). Part coordinates of the tool tip.
(V.)[n].G.PLMEAS3	R	—	—	Value measured on the axis perpendicular to the plane. Part coordinates of the tool tip.
(V.)[n].A.MEASOF.Xn	R	R	R	Measuring error. Difference between the programmed coordinate and the value measured on the Xn axis.
(V.)[n].A.MEASIN.Xn	R	R	R	Measuring error added to the Xn axis.

CANNED CYCLES. ISO CODED PROGRAMMING.

3

Canned cycles may be edited in ISO code (described in this chapter) or with using the cycle editor. See chapter *"4 Canned cycles. Cycle editor."*.

The cycles may be defined anywhere in the program, that is, in the main program as well as in a subroutine. ISO coded cycles can also be executed via MDI mode.

Programming ISO coded cycles.

ISO coded cycles are defined with the #PROBE instruction followed by the number of the cycle to be executed and the call parameters. The calling parameters may be programmed in anyorder.

#PROBE 1	Tool calibration (dimensions and wear).
#PROBE 2	Probe calibration
#PROBE 3	Surfacing measuring.
#PROBE 4	Outside corner measuring.
#PROBE 5	Inside corner measuring.
#PROBE 6	Measuring the angle with the abscissa axis.
#PROBE 7	Outside corner and angle measuring.
#PROBE 8	Hole measuring.
#PROBE 9	Circular boss measuring.
#PROBE 10	Rectangular part centering.
#PROBE 11	Circular part centering.
#PROBE 12	Tabletop probe calibration

Probing canned cycles are not modal; therefore, they must be programmed every time any of them is to be executed. The execution of these cycles does not change the program history.

Cycle data programming.

The cycle number and the rest of parameters may be defined with a number, an arithmetic parameter or expression whose result is a number.

```
#PROBE 4 X10 Y25 Z20 B5 F10
```

```
P1=4 P2=10
#PROBE P1 XP2 Y25 Z[P2*2] B5 FP2
```

When using global parameters, bear in mind that some cycles modify the value of these parameters at the end of the execution. Refer to each cycle to see which parameters it modifies.

Limitations for executing the cycles.

These cycles cannot be executed if tool radius compensation is active.

Canned cycles and the work planes.

Tool and probe calibrations cycles are carried out in the G17, G18 and G19 planes. The rest of the cycles can also be executed in any plane defined with function G20.

Depending on the active plane, the theoretical coordinates of the cycle (corner to be probed, hole center, etc.) may be defined as follows.

- Using the name of the axes that form the plane.

  When the plane is formed by any of the A B C axes, this will not be possible because parameters "A", "B" and "C" may have a special meaning inside the cycle.

- Using parameters "X" "Y" "Z".

  If the plane is formed by the X-Y-Z axes, the "X" parameter is interpreted as the theoretical coordinate in the X axis, the "Y" as the theoretical coordinate in Y and parameter "Z" as the theoretical coordinate in Z.

  If the plane is formed by other axes, the "X" parameter is interpreted as the theoretical coordinate along the abscissa axis, the "Y" as the theoretical coordinate along the ordinate axis and parameter "Z" as the theoretical coordinate along the perpendicular axis.

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

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

On a mill model CNC (milling software installed).

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

`#LATHECY ON` - To activate the turning canned cycles.

`#LATHECY OFF` - To deactivate the turning canned cycles.

On a lathe model CNC (lathe software installed).

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

`#MILLCY ON` - To activate the milling canned cycles.

`#MILLCY OFF` - To deactivate the milling canned cycles.

3.1 #PROBE 1. Tool calibration (dimensions and wear).

This cycle may be used to calibrate the dimensions of a tool or a touch probe. Once the cycle has concluded, it updates the dimensions in the tool table and the tool wears. The following operations are possible with this cycle.

- Calibrate the length of a tool.
- Measure the length wear of a tool.
- Calibrate the radius of a tool.
- Measure the radius wear of a tool.
- Calibrate the radius and length of a tool.
- Measure the radius wear and length wear of a tool.

The calibration is done using a tabletop probe.

The cycle calibrates the active tool. The tool must be selected at the CNC before executing the cycle.

Requirements prior to the calibration.

If it is the first time that the tool or the probe is being calibrated, you must enter its approximate dimensions in the tool table. If it is a probe, the "R" value will correspond to the radius of the probe ball.

Tabletop probe.

Executing this cycle requires a table-top probe, installed in a fixed position of the machine and with its sides parallel to the axes of the plane. The probe position must be given in absolute coordinates referred to machine reference zero using the machine parameters PRB1MIN, PRB1MAX, PRB2MIN, PRB2MAX, PRB3MIN, PRB3MAX.

Data returned by the cycle after a calibration.

Once the calibration has concluded, the cycle updates the dimensions in the tool table and initializes the tool wears to 0 (zero). The cycle only updates the value of the calibrated dimensions; if only the radius has been calibrated, it does not change the length and vice versa.

Once the cycle is over, the CNC will return the detected error in the following arithmetic parameters. A detected error is the difference between the measured value and the one assigned earlier in the table.

P298 Error detected on the tool radius.

P299 Error detected on the tool length.

If when calling the cycle, the calibration of each cutting edge is requested, (parameter ·N·), the lengths are assigned to arithmetic parameters P271 and on; the radii are assigned to arithmetic parameters P251 and on.

Data returned by the cycle after a wear measurement.

Once the wear measurement has ended, the cycle updates in the tool table the measured wears; it does not modify the length value or the radius value. The cycle only updates the value of the measured wear; if only the radius wear has been measured, it does not change the length wear and vice versa.

Once the cycle has ended and if a maximum wear has been set (parameters ·L· and ·M·), the CNC compares the measured wear with the maximum allowed.

- If the measured wear exceeds the maximum allowed, the CNC sets the "expired tool" indicator. When defining the cycle, it is possible to define whether the CNC interrupts the cycle or whether it changes the tool with another one of the same family.

- If the measured wear exceeds the maximum allowed, the cycle will returns the measured wear in the following arithmetic parameters.

  P298 Radius wear.

  P299 Length wear.

If when calling the cycle, the wear measurement of each cutting edge was requested (parameter ·N·), the length wears are assigned to arithmetic parameters P271 and on; the radius wears are assigned to arithmetic parameters P251 and on.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2106

3.1.1 Programming the cycle.

The programming format for this cycle is. Optional parameters are indicated between angle brackets.

#PROBE 1 B <I> <J> F <K> <S> <N> <D> <E> <L> <M> <C> <X U Y V Z W>

B	Safety distance.
I	Optional. Type of calibration and/or measurement of tool wear (length and or radius).
J	Optional. Type of operation (wear calibration or measurement).
F	Probing feedrate.
K	Optional. Sides of the probe to be used.
S	Optional. Speed and turning direction of the tool.
N	Optional. Number of edges to measure.
D	Optional. Distance from the probing point to the tool shaft.
E	Optional. Distance from the probing point to the tool base.
L	Optional. Maximum length wear allowed.
M	Optional. Maximum radius wear allowed.
C	Optional. Behavior when exceeding the maximum wear.
X··W	Optional. Tabletop probe position.

The programming format depends on the operation to carry out, calibrate the length, calibrate the radius, measure length wear or measure radius wear. Depending on the operation to be carried out, it will not be necessary to define all the parameters.

Brief format for calibrating or measuring tool length wear.

The programming format for this cycle is. Optional parameters are indicated between angle brackets.

- To calibrate the tool length.

  #PROBE 1 B <I0> <J0> F <X U Y V Z W>
  #PROBE 1 B <I1> <J0> F <S> <N> <D> <E> <X U Y V Z W>

- To measure tool length wear.

  #PROBE 1 B <I0> <J1> F <L> <C> <X U Y V Z W>
  #PROBE 1 B <I1> <J1> F <S> <N> <D> <E> <L> <C> <X U Y V Z W>

Brief format for calibrating or measuring tool radius wear.

The programming format for this cycle is. Optional parameters are indicated between angle brackets.

- To calibrate tool radius.

  #PROBE 1 B <I2> <J0> F <K> <S> <N> <X U Y V Z W>

- To measure the tool radius wear.

  #PROBE 1 B <I2> <J1> F <K> <S> <N> <M> <C> <X U Y V Z W>

Calibrate and/or measure tool radius wear and/or tool length wear.

The programming format for this cycle is. Optional parameters are indicated between angle brackets.

- To calibrate the tool radius and length.

  #PROBE 1 B <I3> <J0> F <K> <S> <N> <D> <E> <X U Y V Z W>

- Measure or calibrate the tool radius wear and length wear.

  #PROBE 1 B <I3> <J1> F <K> <S> <N> <D> <E> <L> <M> <C> <X U Y V Z W>

Parameter description.

·B· Safety distance.

This parameter only admits positive values greater than 0 (zero).

Distance with respect to the probing point that the tool approaches in G00 before touching the probe. The tool must be placed at a distance greater than this value when the cycle is called.

·I· Type of tool calibration or wear (length and/or radius) measurement.

Optional parameter, by default 0.

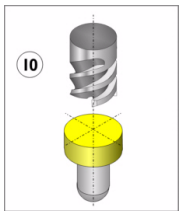
This parameter indicates the dimension of the tool to be calibrated or measured. Tool length calibration or measurement may be done on the tool shaft or on its tip.

I=0 Length, along its shaft.

I=1 Length, at its tip.

I=2 The radius.

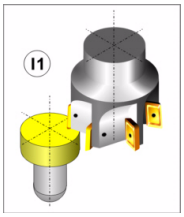
I=3 The tool radius and length.



Calibration or measuring on the tool shaft.

This method is useful for drilling tools, ball end-mills, or tools whose diameter is smaller than the probe's probing surface.

This calibration is done with the spindle stopped.



Calibration or measuring at the tool tip.

This method is useful for calibrating tools with several cutting edges, or tools whose diameter is larger than the probe's probing surface.

This calibration may be carried out either with the spindle stopped or turning in the opposite direction to the cutting direction.

·J· Type of operation (calibration or wear measurement).

Optional parameter, by default 0.

This parameter indicates the type of operation to carry out, calibrate the tool or measure the wear.

J=0 Tool calibration.

J=1 Measure tool wear.

·F· Probing feedrate.

This parameter sets the probing feedrate. The rest of the movements will be carried out in G00.

·K· Probe side to be used.

Optional parameter, by default 0. This parameter is only valid when parameter ·I· has been defined as I2 or I3.

This parameter indicates which side of the probe will be used to calibrate or measure the radius wear.

K=0 Calibration on the X+ side.

K=1 Calibration on the X- side.

K=2 Calibration on the Y+ side.

K=3 Calibration on the Y- side.

·S· Tool turning speed and direction.

Optional parameter, by default 0 (calibration with the spindle stopped).

This parameter sets the speed and turning direction of the tool; the sign sets the turning direction (positive if M3 and negative if M4). If this parameter is set, select a climb cutting direction.

If parameter ·I· has been set as I0, the spindle must be stopped. For the rest of values of parameter ·I·, programming a speed is optional.

·N· Number of edges to measure.

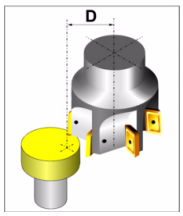
Optional parameter, by default 0 (only one measurement). This parameter is only valid when parameter ·S· has been defined with a value greater than 0 (zero).

This parameter indicates the number of edges to measure; the sign indicates the type of measurement to make.

- If this parameter is not set or is set to zero, The CNC only takes one measurement.
- If this parameter is set with a positive value, the cycle measures all the edges like the first one, looking for the actual position of its tip, instead of assuming that the edges are equidistant (360/N).

·D· Distance from the probing point to the tool shaft.

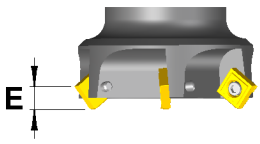
Optional parameter; by default the tool radius. This parameter is only valid when parameter ·I· has been defined as I1 or I3.



This parameter indicates the distance from the tool shaft to the tool probing point.

·E· Distance from the probing point to the tool base.

Optional parameter, by default 0. This parameter is only valid when parameter ·I· has been defined as I2 or I3.



This parameter indicates the distance from the tool base to the tool probing point.

·L· Maximum length wear allowed.

Optional parameter, by default 0.

If this parameter is not set or is set to zero, the CNC does not reject the tool due to length wear and only updates the tool table with the measured value.

If the measured wear is lower than the value defined in this parameter, the CNC updates the tool table with the measured value. If the measured wear exceeds the defined value, the CNC rejects the tool and acts according to the setting of parameter ·C·.

·M· Maximum radius wear allowed.

Optional parameter, by default 0.

If this parameter is not set or is set to zero, the CNC does not reject the tool due to radius wear and only updates the tool table with the measured value.

If the measured wear is lower than the value defined in this parameter, the CNC updates the tool table with the measured value. If the measured wear exceeds the defined value, the CNC rejects the tool and acts according to the setting of parameter ·C·.

·C· Behavior when exceeding the maximum of wear.

Optional parameter, by default 0. This parameter is only valid when parameter ·L· or ·M· has been defined with a value greater than 0 (zero).

This parameter indicates what the CNC does when the measured wear exceeds the value set in parameters ·L· or ·M·. In any case, the CNC sets the "expired tool" indicator in the tool table.

C=0 The CNC shows a message of rejected tool and stops the cycle.

C=1 The cycle replaces the worn out tool with another one of the same family.

·X U Y V Z W· Tabletop probe position.

They are optional parameters that usually need not be defined. In certain machines, due to lack of repeatability in the mechanical positioning of the probe, the probe must be calibrated again before each calibration. Instead of re-defining the machine parameters every time the probe is calibrated, those coordinates may be indicated in these parameters.

Parameters X Y Z refer to the minimum coordinates of the probe on the first axis, second axis and on the axis perpendicular to the plane respectively. Parameters U V W refer to the maximum coordinates of the probe on the first axis, second axis and on the axis perpendicular to the plane respectively.

This data does not modify the machine parameters. The CNC takes this data into account only during this calibration. If any of this data is left out, the CNC takes the value assigned to the corresponding machine parameter.

3.1.2 Basic operation.

Calibrate or measure tool length wear.

1 Approach movement.

Rapid probe movement (G00) from the cycle calling point to the safety point. The safety point will be located at a distance ·B· over the probe and at a distance ·D· from the center of the tool to the point to be probed.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.

2 Movement to locate an edge (only if ·N· has been defined with a value other than 0).

The CNC starts the spindle at an ·S· speed and moves the tool at the feedrate ·F· along the axis perpendicular to the active plane until receiving the probe signal. Once it knows the position of an edge, the tool withdraws to the safety point and stops the spindle.

3 Probing movement.

Probing movement along the axis perpendicular to the active plane at the feedrate ·F· until the probe signal is received. Withdrawal to the safety point.

If ·N· has been defined to a value other than 0, this movement is repeated ·N· times.

4 Withdrawal movement.

Movement in the work plane up to the cycle calling point.

Calibrate or measure tool radius wear.

1 Approach movement.

Rapid probe movement (G00) from the cycle calling point to the safety point. The safety point will be located at a ·B· distance from the probe.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.

2 Movement to locate an edge (only if ·N· has been defined with a value other than 0).

The CNC starts the spindle at an ·S· speed and moves the tool at the feedrate ·F· along the selected axis until receiving the probe signal. The CNC stops the spindle and carries out some probing moves to locate the edge more precisely. Once it knows the position of an edge, the tool withdraws to the safety point.

3 Probing movement.

Probing movement at the indicated feedrate ·F· until the probe signal is received. Withdrawal to the safety point.

If ·N· has been defined to a value other than 0, this movement is repeated ·N· times.

4 Withdrawal movement.

This movement is carried out in two stages. It first moves in the work plane and then along the perpendicular axis and then in the work plane.

Calibrate or measure tool radius or length wear.

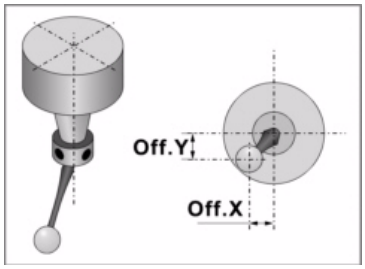
It carries out the movements of the previous sections sequentially; it first calibrates and/or measures the radius and then the length. The tool returns to the cycle calling point only after measuring the tool length.

When the number of edges ·N· is other than zero, the position of the edges is known after measuring or calibrating the radius, thus not being necessary to do it when measuring or calibrating the length.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2106

3.2 #PROBE 2. Probe calibration

This cycle may be used to calibrate the probe located in the spindle. This probe will be the one used in the tool calibration canned cycles.



The cycle measures the deviation of the probe ball with respect to the tool-holder axis and the electronic radius of the probe. The electronic radius is the deflection of the probe since it makes contact until the probing signal is activated.

For the calibration, a hole previously machined and with known center and dimensions is used.

Requirements prior to the calibration.

The probe tool must be calibrated in length and its dimensions defined in the tool table. The fields of the tool table for each probe will have the following meaning:

R	Probe ball radius. This value must be entered manually in the table.
L	Probe length. This value will be assigned by the tool length calibration cycle.
Off.X	Deviation of the probe ball shaft with respect to the tool holder along the abscissa axis. This value will be assigned by this cycle.
Off.Y	Deviation of the probe ball shaft with respect to the tool holder along the ordinate axis. This value will be assigned by this cycle.
RW	Electronic radius. Deflection of the probe since it makes contact until the probing signal is activated. The electronic radius is set with a negative value.

The following steps must be followed for calibrating the probe:

1. Once the probe characteristics have been checked, manually enter the ball radius value (R) in the tool table.
2. After selecting the corresponding tool number and tool offset, execute the tool length calibrating cycle. This cycle updates the length value and resets the "Off.Z" value to zero.
3. Execute the probe calibration canned cycle that updates the "Off.X" and "Off.Y" values (deviation of the axis of the probe ball with respect to the axis of the tool holder) and RW (electronic radius).

Data returned by the cycle after the measurement.

Once the cycle has concluded, it updates the tool table with the value tool offset that is currently active. Offsets "Off.X" and "Off.Y" for the abscissa and ordinate axes respectively indicate the deviation of the probe ball. "RW" indicates the deflection of the probe since it makes contact until the probing signal is activated.

Likewise, the CNC returns the best value to be assigned to axis and spindle machine parameter PROBEDELAY in the following arithmetic parameters.

P299	Best value of parameter PROBEDELAY for all the axes and the spindle.

3.2.1 Programming the cycle.

The programming format for this cycle is.

#PROBE 2 X Y Z B J E H F

X Y	Real coordinates of the arc center.
Z	Coordinates where probing takes place.
B	Safety distance.
J	Theoretical hole diameter.
E	Withdrawal distance after initial probing.
H	Feedrate for the initial probing movement.
F	Probing feedrate.

·X Y· Real coordinates of the hole center.

Optional parameter; if not programmed, the cycle takes the probe position as theoretical center of the hole.

The canned cycles may be programmed in any work plane. See *"Canned cycles and the work planes."* on page 42.

·Z· Coordinates along the longitudinal axis where probing takes place.

Optional parameter; if not programmed, the cycle takes the probe position as the coordinate where probing will take place.

·B· Safety distance.

This parameter only admits positive values greater than 0 (zero).

·J· Real diameter of the hole.

This parameter only admits positive values greater than 0 (zero).

This parameter sets the real diameter of the hole. This cycle may be used to measure holes with diameters no larger than "J+B".

·E· Withdrawal distance after initial probing.

This parameter only admits positive values greater than 0 (zero).

This parameter sets the distance the probe withdrawals after the first probing movement. Once it withdraws this distance, the CNC makes a second probing movement.

·H· Feedrate for the first probing movement.

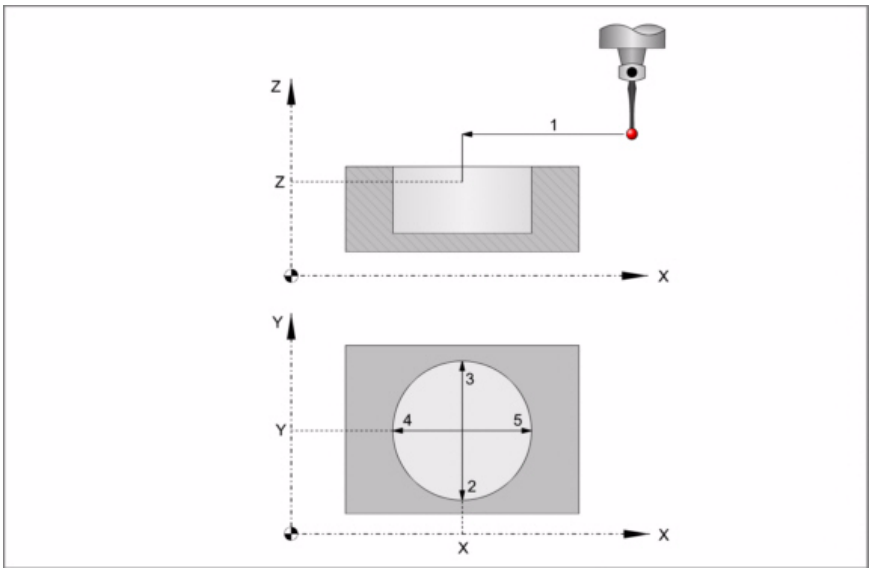
This parameter sets the feedrate for the first probing movement. Then, the CNC will repeat the probing movement at feedrate ·F·.

·F· Probing feedrate.

This parameter sets the feedrate for the second probing movement. The rest of the movements are carried out in G00 except the first probing movement which is carried out at feedrate ·H·.

FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2106

3.2.2 Basic operation.



1 If the approach coordinates X Y Z have been programmed, approach movement. Probe's rapid movement (G00) from the cycle calling point to the center of the hole.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.

2 Probing of the first point

The probe touches in four different points. In each point, it makes a first probing movement, a withdrawal movement and a second probing move. After the second probing move, the probe returns to the center of the hole.

First probing movement. Probe movement along the ordinate axis at feedrate ·H· until the probe signal is received. The maximum probing distance is "B+J/2". If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.

Withdrawal movement. Probe's rapid movement (G00) the distance indicated by ·E·.

Second probe. Probe movement along the ordinate axis at feedrate ·F· until the probe signal is received.

Withdrawal movement to the center of the hole. Rapid probe movement (G00) from the probing point to the center of the hole.

3 Probing of the second point.

It is done along the ordinate axis same as the previous one.

4 Probing of the third point.

It is done along the abscissa axis same as the previous one.

5 Probing of the fourth point.

It is done along the abscissa axis same as the previous one.

6 180º probe rotation. The probe touches again in the previous four points.

It is up to the user to make sure that the probe can rotate without danger.

- *The hole must be large enough to let the probe rotate taking into account its real offsets.*
- *If the probe has a cable, check that the cable does not hinder the rotation of the probe.*

7 Rotation of the probe to its previous position. If the approach coordinates X Y Z have been programmed, withdrawal movement. Rapid probe movement (G00) from the center of the hole to the cycle calling point.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane and then in the work plane.

3.3 #PROBE 3. Surfacing measuring.

This cycle measures the part along the selected axis. With this cycle, it is also possible to correct the value of the wear of the tool used to machine the surface. The wear correction only takes place when the measuring error exceeds a programmed value.

Tool wear compensation.

To enable wear compensation, the calling instruction must define all the parameters ·T· (tool) and ·D· (offset). The wear correction only takes place when the measuring error exceeds the tolerance programmed in parameter ·L·.

Data returned by the cycle after the measurement.

Once the cycle has ended, the CNC returns the real values obtained in the measurement in the following arithmetic parameters:

P298 Actual (real) surface coordinate.

P299 Detected error. Difference between the actual surface coordinate and the programmed theoretical coordinate.

If wear correction is enabled in the calling instruction, the CNC updates those values in the programmed tool. This correction is applied only if the measuring error is equal to or greater than the programmed tolerance.

Depending on the axis used for measuring (parameter ·K·), the cycle will correct either the tool length value or the tool radius value.

- If the measurement is made on the axis perpendicular to the work plane, it will correct the length wear.
- If the measurement is made with one of the axis forming the plane, it will correct the tool radius wear.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2106

3.3.1 Programming the cycle.

The programming format for this cycle is. Optional parameters are indicated between angle brackets.

#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	Optional. Probing axis.
F	Probing feedrate.
C	Optional. Cycle ending point (calling point or measured point).
L	Optional. Tolerance for the measuring error.
T	Optional. Tool to be corrected.
D	Optional. Tool offset to be corrected.

·X Y Z· Theoretical coordinates of the probing point.

The canned cycles may be programmed in any work plane. See *"Canned cycles and the work planes."* on page 42.

·B· Safety distance.

This parameter only admits positive values greater than 0 (zero).

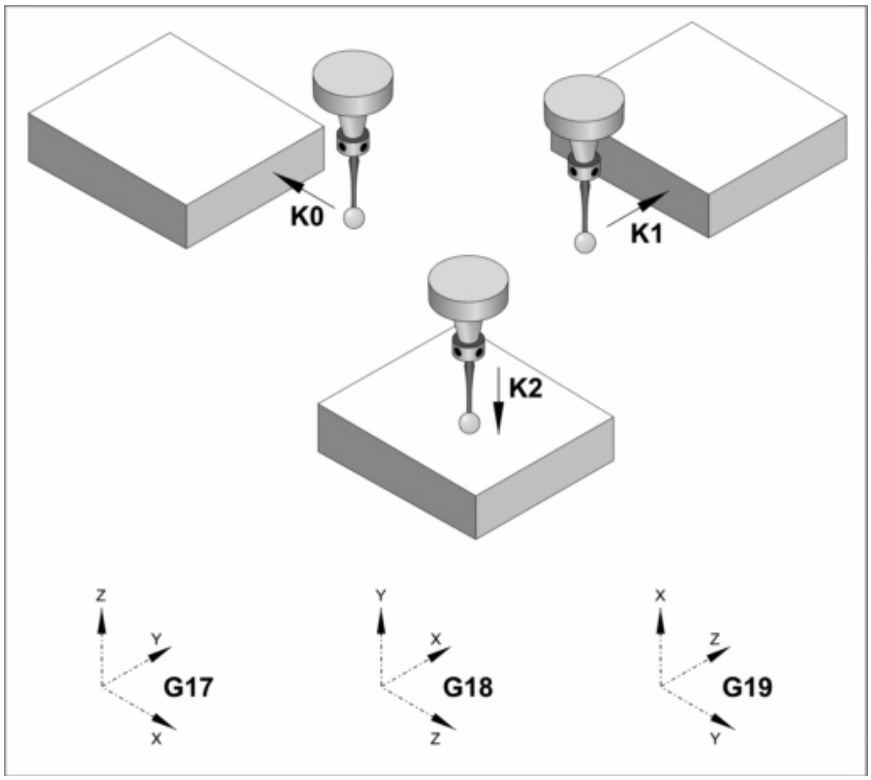
Distance with respect to the point to measure and along the probing axis, to which the probe approaches in G00 before making the probing movement. When calling the cycle, the probe must be located, with respect to the point to be measured, at a greater distance than this value

·K· Probing axis.

Optional parameter, by default 0.

This parameter indicates the axis to be used to measure the surface.

K=0	Probing along the abscissa axis [G17(X) G18(Z) G19 (Y)].
K=1	Probing along the ordinate axis [G17(Y) G18(X) G19(Z)].
K=2	Probing along the axis perpendicular to the plane [G17(Z) G18(Y) G19(Z)].



·F· Probing feedrate.

This parameter sets the probing feedrate. The rest of the movements will be carried out in G00.

·C· Cycle ending point (calling point or measured point).

Optional parameter, by default 0.

This parameter indicates where the probing cycle ends.

C=0 The probe returns to the point from where the cycle was called.

C=1 The cycle ends with the probe positioned over the measured point. The longitudinal axis returns to the point where the cycle was called.

·L· Tolerance for the measuring error.

Optional parameter, by default 0. This parameter only admits positive values.

If the measuring error (difference between the theoretical and the real values) is within this tolerance, the CNC does not change the tool data. If the measuring error is equal to or greater than this tolerance, the CNC corrects the data of the tool defined in parameters ·T· and ·D·.

·T· Tool to be corrected.

Optional parameter, by default 0. If T=0 (or not programmed), tool wear is not corrected. To correct tool wear, program parameters ·T· and ·D· with a value other than zero.

Tool whose wear is to be corrected, which will be the tool used to machine the surface.

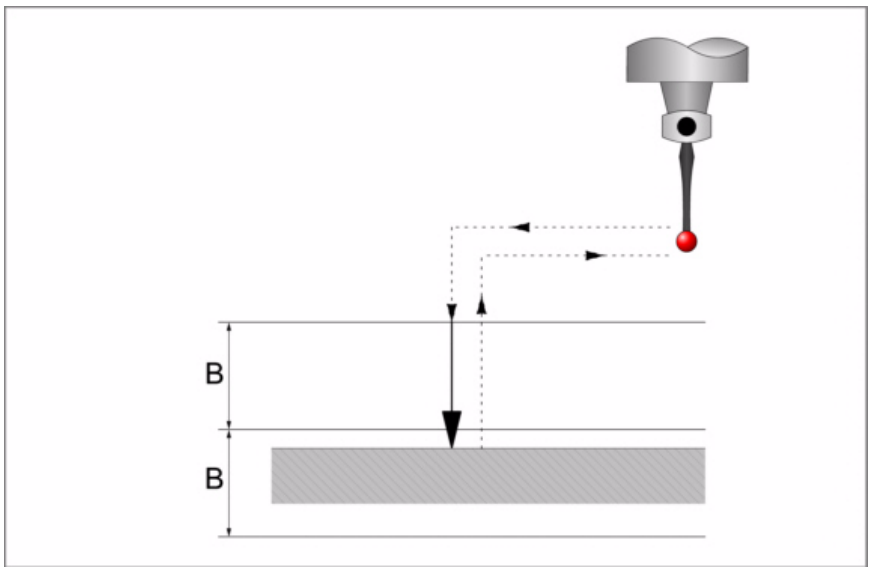
·D· Tool offset to be corrected.

Optional parameter, by default 0. If D=0 (or not programmed), tool wear is not corrected. To correct tool wear, program parameters ·T· and ·D· with a value other than zero.

Tool offset whose wear is to be corrected, which will be the tool offset used to machine the surface.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2106

3.3.2 Basic operation.



1 Approach movement.

Rapid probe movement (G00) from the cycle calling point to the approach point located at a distance ·B· from the probing point and along the probing axis (K).

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.

2 Probing movement.

Probing movement. Movement of the probe along the selected axis (K) at the indicated feedrate (F), until the probe signal is received. The maximum probing distance is ·2B·. If once this distance has been reached, the CNC has not yet received the probe signal, it will issue the relevant error code and stop the movement of the axes.

Once probing is over, the CNC will assume the actual position of the axes when the probe signal is received as their theoretical position.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the approach point.

3 Withdrawal movement.

Rapid probe movement (G00) from the probing point to the cycle calling point.

It first moves along the perpendicular axis up to the coordinate of the cycle calling point along that axis. If programmed(C0), it makes a movement in the main plane to the cycle calling point.

3.4 #PROBE 4. Outside corner measuring.

This cycle calculates the position of an outside corner.

Data returned by the cycle after the measurement.

Once the cycle has ended, the CNC returns the real values obtained in the measurement in the following arithmetic parameters:

P296 Real coordinate of the corner along the abscissa axis.

P297 Real coordinate of the corner along the ordinate axis.

P298 Error detected along the abscissa axis. Difference between the actual corner coordinate ant the programmed theoretical coordinate.

P299 Error detected along the ordinate axis. Difference between the actual corner coordinate ant the programmed theoretical coordinate.

3.4.1 Programming the cycle.

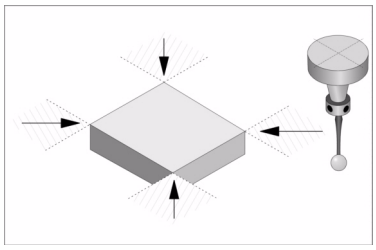
The programming format for this cycle is. Optional parameters are indicated between angle brackets.

#PROBE 4 X Y Z B F

X Y Z	Theoretical coordinates corner being measured.
B	Safety distance.
F	Probing feedrate.

·X Y Z· Theoretical coordinates of the corner being measured.

The canned cycles may be programmed in any work plane. See *"Canned cycles and the work planes."* on page 42.



Depending on the part corner to be measured, the probe must be placed in the corresponding shaded area (see figure) before calling the cycle.

·B· Safety distance.

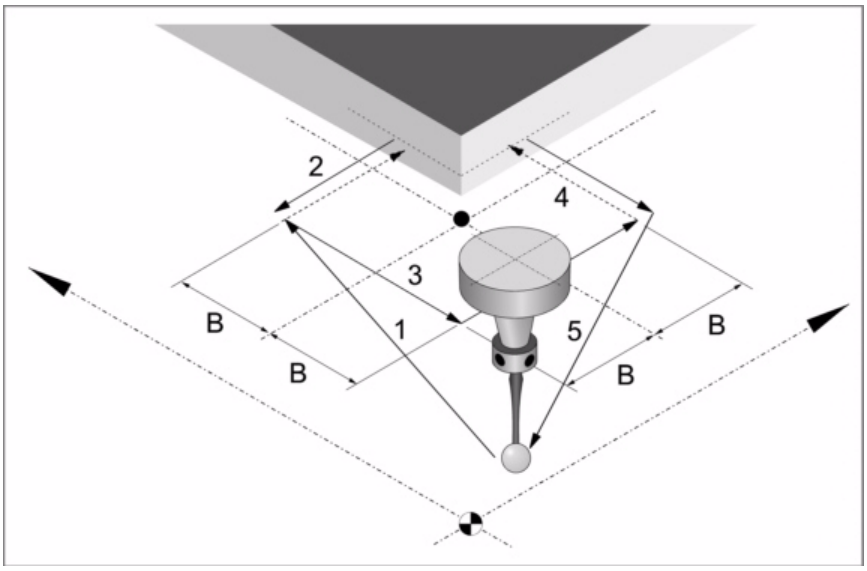
This parameter only admits positive values greater than 0 (zero).

Distance with respect to the point to measure, to which the probe approaches in G00 before making the probing movement. When calling the cycle, the probe must be located, with respect to the point to be measured, at a greater distance than this value

·F· Probing feedrate.

This parameter sets the probing feedrate. The rest of the movements will be carried out in G00.

3.4.2 Basic operation.



1 Approach movement to the first side to be probed.

Rapid probe movement (G00) from the cycle calling point to the first approach point located at a ·B· distance from the side to be probed.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.

2 Probing on the first side.

The probing on each side consists of a probing move and a withdrawal move.

Probing movement. Probe movement along the abscissa axis at feedrate ·F· until the probe signal is received. The maximum probing distance is ·2B·. If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the first approach point.

3 Approach movement to the second side to be probed.

Rapid probe move (G00) from the first approach point to the second.

This approach movement is made in two stages. It first moves in the work plane and then along the ordinate axis and then along the abscissa axis.

4 Probing on the second side.

Probing movement. Probe movement along the ordinate axis at feedrate ·F· until the probe signal is received. The maximum probing distance is ·2B·. If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the second approach point.

5 Withdrawal movement.

Rapid probe movement (G00) from the second approach point to the cycle calling point.

This withdrawal movement is made in two stages. It first moves in the work plane and then along the perpendicular axis and then in the work plane.

3.5 #PROBE 5. Inside corner measuring.

This cycle calculates the position of an inside corner.

Data returned by the cycle after the measurement.

Once the cycle has ended, the CNC returns the real values obtained in the measurement in the following arithmetic parameters:

P296 Real coordinate of the corner along the abscissa axis.

P297 Real coordinate of the corner along the ordinate axis.

P298 Error detected along the abscissa axis. Difference between the actual corner coordinate ant the programmed theoretical coordinate.

P299 Error detected along the ordinate axis. Difference between the actual corner coordinate ant the programmed theoretical coordinate.

3.5.1 Programming the cycle.

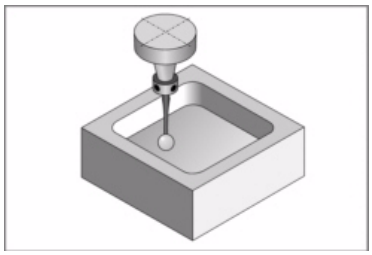
The programming format for this cycle is. Optional parameters are indicated between angle brackets.

#PROBE 5 X Y Z B F

X Y Z	Theoretical coordinates corner being measured.
B	Safety distance.
F	Probing feedrate.

·X Y Z· Theoretical coordinates of the corner being measured.

The canned cycles may be programmed in any work plane. See *"Canned cycles and the work planes."* on page 42.



The probe must be placed within the pocket before calling the cycle.

·B· Safety distance.

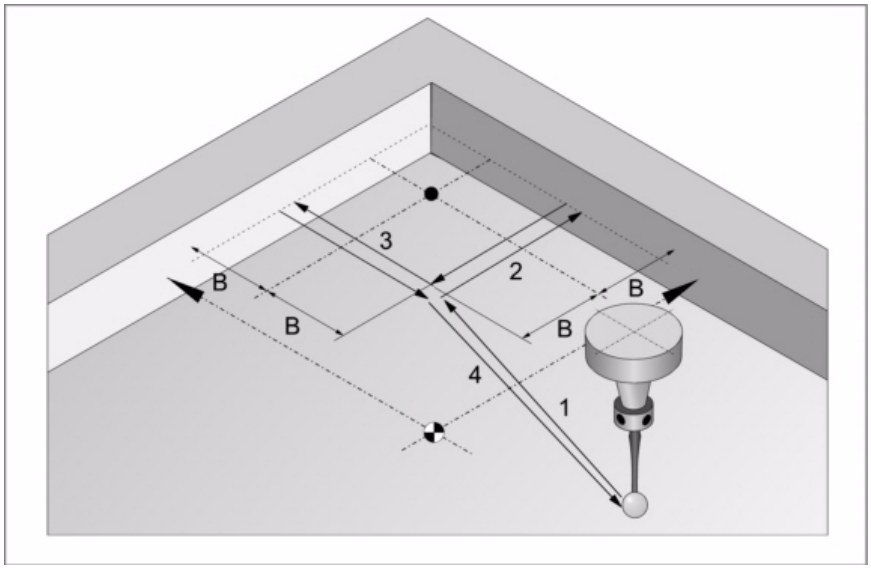
This parameter only admits positive values greater than 0 (zero).

Distance with respect to the point to measure, to which the probe approaches in G00 before making the probing movement. When calling the cycle, the probe must be located, with respect to the point to be measured, at a greater distance than this value

·F· Probing feedrate.

This parameter sets the probing feedrate. The rest of the movements will be carried out in G00.

3.5.2 Basic operation.



1 Approach movement.

Rapid probe movement (G00) from the cycle calling point to the first approach point located at a ·B· distance from the two sides to be probed.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.

2 Probing on the first side.

The probing on each side consists of a probing move and a withdrawal move.

Probing movement. Probe movement along the abscissa axis at feedrate ·F· until the probe signal is received. The maximum probing distance is ·2B·. If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the approach point.

3 Probing on the second side.

Probing movement. Probe movement along the ordinate axis at feedrate ·F· until the probe signal is received. The maximum probing distance is ·2B·. If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the approach point.

4 Withdrawal movement.

Rapid probe movement (G00) from the approach point to the cycle calling point.

This withdrawal movement is made in two stages. It first moves in the work plane and then along the perpendicular axis and then in the work plane.

3.6 #PROBE 6. Measuring the angle with the abscissa axis.

This cycle calculates the angle of the part with respect to the abscissa axis. This cycle may be used to measure angles between ±45º.

- If the angle to be measured is equal to or greater than 45º, the CNC will display the relevant error.
- If the angle to be measured is equal to or smaller than –45º the probe will collide with the part.

Data returned by the cycle after the measurement.

Once the cycle has ended, the CNC returns the real values obtained in the measurement in the following arithmetic parameters:

P295 Inclination angle between the part and the abscissa axis.

3.6.1 Programming the cycle.

The programming format for this cycle is. Optional parameters are indicated between angle brackets.

#PROBE 6 X Y Z B F

X Y Z	Theoretical coordinates corner being measured.
B	Safety distance.
F	Probing feedrate.

·X Y Z· Theoretical coordinates of the corner being measured.

The canned cycles may be programmed in any work plane. See *"Canned cycles and the work planes."* on page 42.

·B· Safety distance.

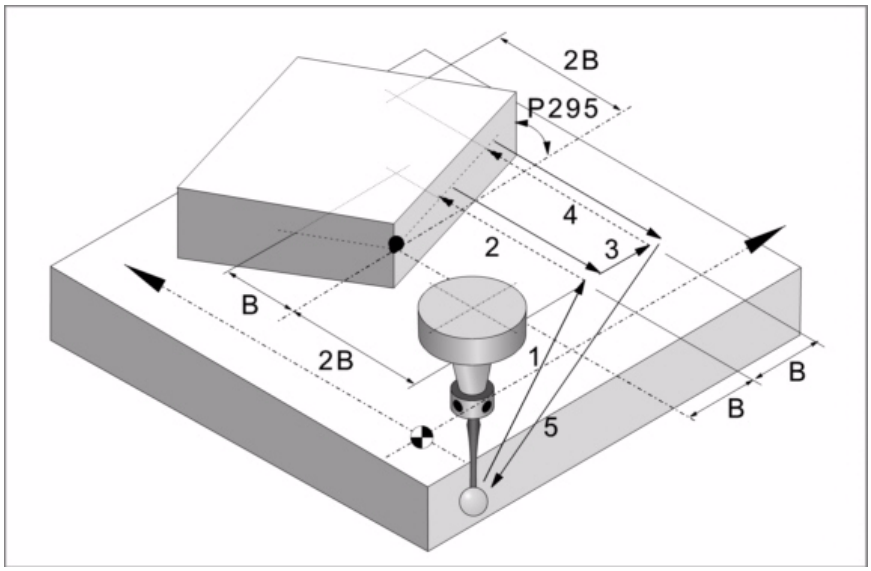
This parameter only admits positive values greater than 0 (zero).

Distance with respect to the point to measure, to which the probe approaches in G00 before making the probing movement. When calling the cycle, the probe must be located at a distance twice this value with respect to the point to be measured,

·F· Probing feedrate.

This parameter sets the probing feedrate. The rest of the movements will be carried out in G00.

3.6.2 Basic operation.



1 First approach movement.

Rapid probe movement (G00) from the cycle calling point to the first approach point located at a distance ·2B· from the side to be probed.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.

2 First probing move.

Probing movement. Probe movement along the ordinate axis at feedrate ·F· until the probe signal is received. The maximum probing distance is ·3B·. If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the first approach point.

3 Second approach movement.

Rapid probe move (G00) from the first approach point to the second. The second approach point is located at a distance ·B· from the first one.

4 Second probe.

Probing movement. Probe movement along the ordinate axis at feedrate ·F· until the probe signal is received. The maximum probing distance is ·4B·. If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the second approach point.

5 Withdrawal movement.

Rapid probe movement (G00) from the approach point to the cycle calling point.

This withdrawal movement is made in two stages. It first moves in the work plane and then along the perpendicular axis and then in the work plane.

3.7 #PROBE 7. Outside corner and angle measuring.

This cycle calculates the position of an outside corner and the angle of the part with respect to the abscissa axis. This cycle may be used to measure angles between ±45º.

- If the angle to be measured is equal to or greater than 45º, the CNC will display the relevant error.
- If the angle to be measured is equal to or smaller than –45º the probe will collide with the part.

Data returned by the cycle after the measurement.

Once the cycle has ended, the CNC returns the real values obtained in the measurement in the following arithmetic parameters:

P295 Inclination angle between the part and the abscissa axis.

P296 Real coordinate of the corner along the abscissa axis.

P297 Real coordinate of the corner along the ordinate axis.

P298 Error detected along the abscissa axis. Difference between the actual corner coordinate ant the programmed theoretical coordinate.

P299 Error detected along the ordinate axis. Difference between the actual corner coordinate ant the programmed theoretical coordinate.

3.7.1 Programming the cycle.

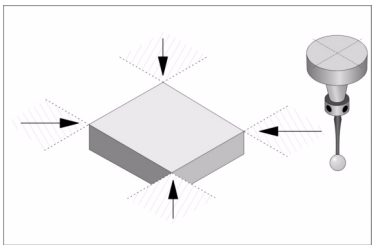
The programming format for this cycle is. Optional parameters are indicated between angle brackets.

#PROBE 7 X Y Z B F

X Y Z	Theoretical coordinates corner being measured.
B	Safety distance.
F	Probing feedrate.

·X Y Z· Theoretical coordinates of the corner being measured.

The canned cycles may be programmed in any work plane. See *"Canned cycles and the work planes."* on page 42.



Depending on the part corner to be measured, the probe must be placed in the corresponding shaded area (see figure) before calling the cycle.

·B· Safety distance.

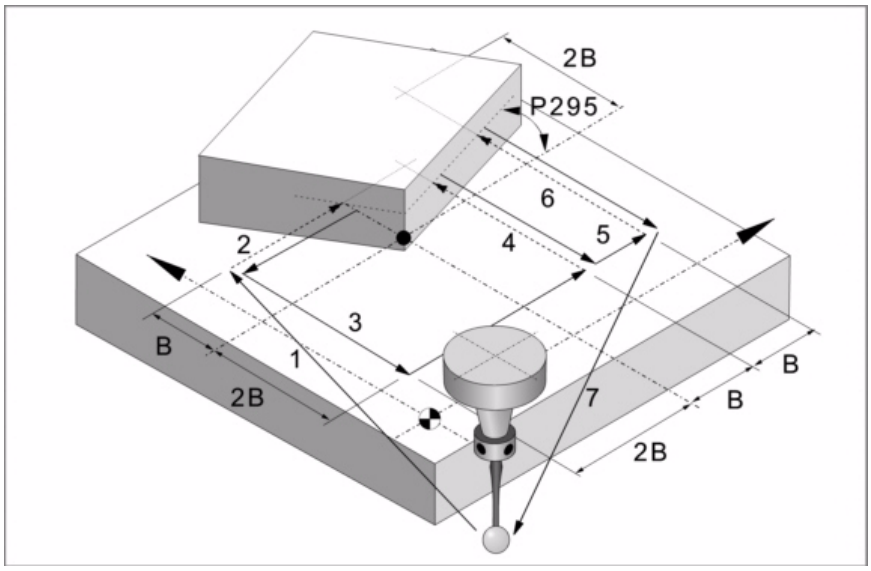
This parameter only admits positive values greater than 0 (zero).

Distance with respect to the point to measure, to which the probe approaches in G00 before making the probing movement. When calling the cycle, the probe must be located at a distance twice this value with respect to the point to be measured,

·F· Probing feedrate.

This parameter sets the probing feedrate. The rest of the movements will be carried out in G00.

3.7.2 Basic operation.



1 First approach movement.

Rapid probe movement (G00) from the cycle calling point to the first approach point located at a ·2B· distance from the side to be probed.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.

2 First probing move.

Probing movement. Probe movement along the ordinate axis at feedrate ·F· until the probe signal is received. The maximum probing distance is ·3B·. If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the first approach point.

3 Second approach movement.

Rapid probe movement (G00) from the first approach point to the second, located at a ·2B· distance from the second side to be probed.

This approach movement is made in two stages. It first moves in the work plane and then along the ordinate axis and then along the abscissa axis.

4 Second probe.

Probing movement. Probe movement along the ordinate axis at feedrate ·F· until the probe signal is received. The maximum probing distance is ·3B·. If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the second approach point.

5 Third approach movement.

Rapid probe move (G00) from the second approach point to the third. It is located at a ·B· distance from the previous one.

6 Third probing move.

Probing movement. Probe movement along the ordinate axis at feedrate ·F· until the probe signal is received. The maximum probing distance is ·4B·. If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the third approach point.

7 Withdrawal movement.

Rapid probe movement (G00) from the approach point to the cycle calling point.

This withdrawal movement is made in two stages. It first moves in the work plane and then along the perpendicular axis and then in the work plane.

3.8 #PROBE 8. Hole measuring.

This cycle calculates the diameter of a hole and the real position of the center.

Data returned by the cycle after the measurement.

Once the cycle has ended, the CNC returns the real values obtained in the measurement in the following arithmetic parameters:

P294	Hole diameter.
P295	Error detected in the hole diameter. Difference between the real diameter and the programmed theoretical diameter.
P296	Real coordinate of the center along the abscissa axis.
P297	Real center coordinate along the ordinate axis.
P298	Error detected along the abscissa axis. Difference between the actual center coordinate and the programmed theoretical coordinate.
P299	Error detected along the ordinate axis. Difference between the actual center coordinate and the programmed theoretical coordinate.

3.8.1 Programming the cycle.

The programming format for this cycle is. Optional parameters are indicated between angle brackets.

#PROBE 8 X Y Z B J E <C> H F

X Y Z	Theoretical coordinates of the arc center.
B	Safety distance.
J	Theoretical hole diameter.
E	Withdrawal distance after initial probing.
C	Optional. Cycle ending point (calling point or measured point).
H	Feedrate for the initial probing movement.
F	Probing feedrate.

·X Y Z· Theoretical coordinates of the hole center.

The canned cycles may be programmed in any work plane. See *"Canned cycles and the work planes."* on page 42.

·B· Safety distance.

This parameter only admits positive values greater than 0 (zero).

·J· Theoretical diameter of the hole.

This parameter only admits positive values greater than 0 (zero).

This parameter sets the theoretical diameter of the hole. This cycle may be used to measure holes with diameters no larger than "J+B".

·E· Withdrawal distance after initial probing.

This parameter only admits positive values greater than 0 (zero).

This parameter sets the distance the probe withdrawals after the first probing movement. Once it withdraws this distance, the CNC makes a second probing movement.

·C· Cycle ending point (calling point or measured point).

Optional parameter, by default 0.

This parameter indicates where the probing cycle ends.

C=0	The probe returns to the point from where the cycle was called.
C=1	The cycle will finish with the probe at the real (calculated) hole center.

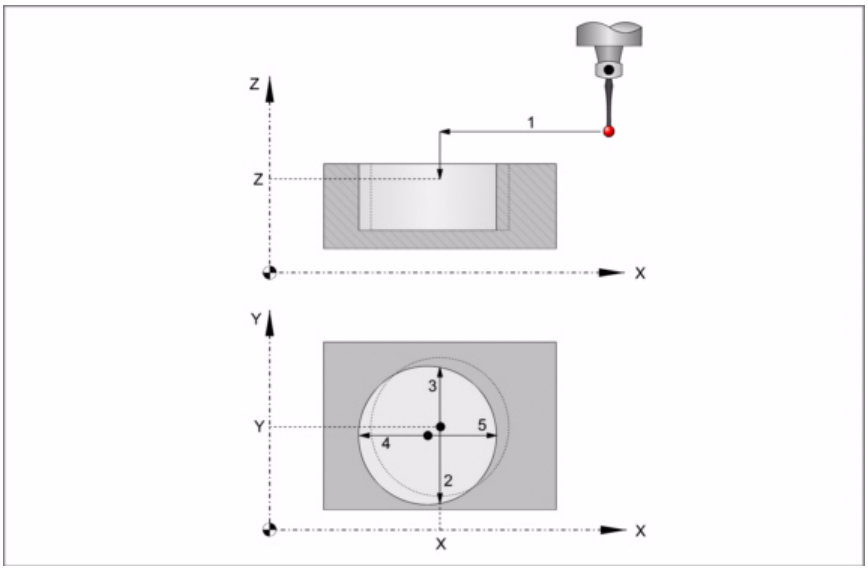
·H· Feedrate for the first probing movement.

This parameter sets the feedrate for the first probing movement. Then, the CNC will repeat the probing movement at feedrate ·F·.

·F· Probing feedrate.

This parameter sets the feedrate for the second probing movement. The rest of the movements are carried out in G00 except the first probing movement which is carried out at feedrate ·H·.

3.8.2 Basic operation.



1 Approach movement.

Probe's rapid movement (G00) from the cycle calling point to the center of the hole.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.

2 Probing of the first point

The probe touches in four different points. In each point, it makes a first probing movement, a withdrawal movement and a second probing move. After the second probing move, the probe returns to the center of the hole.

First probing movement. Probe movement along the ordinate axis at feedrate ·H· until the probe signal is received. The maximum probing distance is "B+J/2". If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.

Withdrawal movement. Probe's rapid movement (G00) the distance indicated by ·E·.

Second probe. Probe movement along the ordinate axis at feedrate ·F· until the probe signal is received.

Withdrawal movement to the center of the hole. Rapid probe movement (G00) from the probing point to the center of the hole.

3 Probing of the second point.

It is done along the ordinate axis same as the previous one.

4 Probing of the third point.

It is done along the abscissa axis same as the previous one.

5 Probing of the fourth point.

It is done along the abscissa axis same as the previous one.

The probe moves to the real hole center (the calculated one).

6 Withdrawal movement (if it has been defined).

If it has been defined, the probe moves in rapid (G00) from the center of the hole to the cycle calling point.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane and then in the work plane.

3.9 #PROBE 9. Circular boss measuring.

This cycle calculates the diameter of a boss and the real position of the center.

Data returned by the cycle after the measurement.

Once the cycle has ended, the CNC returns the real values obtained in the measurement in the following arithmetic parameters:

P294 Boss diameter.

P295 Error detected in the boss diameter. Difference between the real diameter and the programmed theoretical diameter.

P296 Real coordinate of the center along the abscissa axis.

P297 Real center coordinate along the ordinate axis.

P298 Error detected along the abscissa axis. Difference between the actual center coordinate and the programmed theoretical coordinate.

P299 Error detected along the ordinate axis. Difference between the actual center coordinate and the programmed theoretical coordinate.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2106

3.9.1 Programming the cycle.

The programming format for this cycle is. Optional parameters are indicated between angle brackets.

#PROBE 9 X Y Z B J E <C> H F

X Y Z	Coordinates of the boss center.
B	Safety distance.
J	Theoretical boss diameter.
E	Withdrawal distance after initial probing.
C	Optional. Cycle ending point (calling point or measured point).
H	Feedrate for the initial probing movement.
F	Probing feedrate.

·X Y Z· Theoretical coordinates of the boss center.

The canned cycles may be programmed in any work plane. See *"Canned cycles and the work planes."* on page 42.

·B· Safety distance.

This parameter only admits positive values greater than 0 (zero).

·J· Theoretical diameter of the boss.

This parameter only admits positive values greater than 0 (zero).

This parameter sets the theoretical diameter of the boss. This cycle may be used to measure bosses with diameters no larger than "J+B".

·E· Withdrawal distance after initial probing.

This parameter only admits positive values greater than 0 (zero).

This parameter sets the distance the probe withdrawals after the first probing movement. Once it withdraws this distance, the CNC makes a second probing movement.

·C· Cycle ending point (calling point or measured point).

Optional parameter, by default 0.

This parameter indicates where the probing cycle ends.

C=0	The probe returns to the point from where the cycle was called.
C=1	The cycle ends by positioning the probe over the real (calculated) center of the boss, at a ·B· distance from the programmed coordinate.

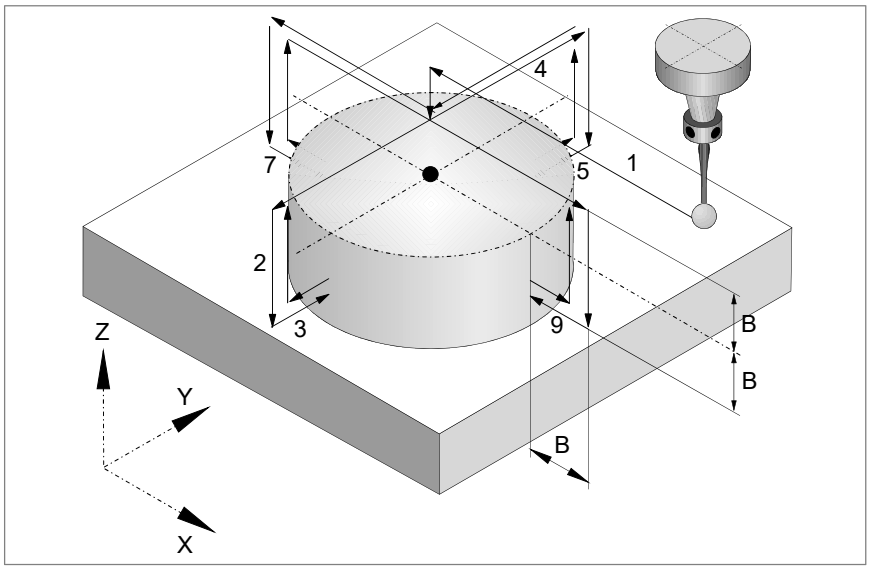
·H· Feedrate for the first probing movement.

This parameter sets the feedrate for the first probing movement. Then, the CNC will repeat the probing movement at feedrate ·F·.

·F· Probing feedrate.

This parameter sets the feedrate for the second probing movement. The rest of the movements are carried out in G00 except the first probing movement which is carried out at feedrate ·H·.

3.9.2 Basic operation.



1 Positioning over the boss center.

Probe's rapid movement (G00) from the cycle calling point to the center of the boss. The probe positions over the boss at a distance ·B· from the surface.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.

2 First approach movement.

Rapid probe movement (G00) from the center of the boss to the first approach point.

This approach movement is made in two stages. It first moves in the work plane and then along the ordinate axis and then along the perpendicular axis a distance ·2B·.

3 Probing of the first point

The probe touches in four different points. In each point, it makes a first probing movement, a withdrawal movement and a second probing move. After the second probing move, the probe withdraws to the approach point.

First probing movement. Probe movement along the ordinate axis at feedrate ·H· until the probe signal is received. The maximum probing distance is "B+J/2". If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.

Withdrawal movement. Probe's rapid movement (G00) the distance indicated by ·E·.

Second probe. Probe movement along the ordinate axis at feedrate ·F· until the probe signal is received.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the approach point.

4 Second approach movement.

Movement of the probe in rapid (G00) from the first approach point to the second.

The perpendicular axis moves at a distance ·B· above the boss, the probe moves to the next approach point above the boss and it goes back down to a distance ·B· from the surface.

5 Probing of the second point.

It is done along the ordinate axis same as the previous one.

6 Third approach movement.

Movement of the probe from the second approach point to the center of the part and then to the third point.

7 Probing of the third point.

It is done along the abscissa axis same as the previous one.

8 Fourth approach movement.

It is done like the previous one.

9 Probing of the fourth point.

It is done along the abscissa axis same as the previous one.

The probe moves in rapid (G00) at a distance ·B· over the boss up to the calculated real boss center.

10 Withdrawal movement (if it has been defined).

If it has been defined, the probe moves in rapid (G00) from the center of the boss to the cycle calling point.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane and then in the work plane.

3.10 #PROBE 10. Rectangular part centering.

This cycle may be used to calculate the center of a rectangular part of known dimensions as well as the inclination of the part with respect to the abscissa axis.

Requirements prior to the calibration.

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.

To execute this cycle, the probe must be properly calibrated.

Data returned by the cycle after the measurement.

Once the cycle has ended, the CNC returns the real values obtained in the measurement in the following arithmetic parameters:

P296 Inclination angle between the part and the abscissa axis.

P297 Part surface coordinate.

P298 Part center coordinate along the abscissa axis.

P299 Part center coordinate along the ordinate axis.

3.10.1 Programming the cycle.

The programming format for this cycle is. Optional parameters are indicated between angle brackets.

#PROBE 10 <X Y Z> I J <K> <L> <B> D E <H> <F> <Q> <C>

X Y Z	Probe position to execute the cycle.
I	Part length along the X (abscissa) axis.
J	Part length along the Y (ordinate) axis.
C	Optional. Centering type.
K	Optional. Axis and direction of the first probing movement.
L	Optional. Surface coordinate measuring.
B	Optional. Safety distance.
D	Safety distance along Z.
E	Withdrawal distance after initial probing.
H	Optional. Feedrate for the initial probing movement.
F	Optional. Probing feedrate.
Q	Optional. Feedrate for approach movements.

·X Y Z· Probe position to execute the cycle.

Optional parameter; by default the probe position when calling the cycle.

The canned cycles may be programmed in any work plane. See *"Canned cycles and the work planes."* on page 42.

·I· Part length along the X (abscissa) axis.

This parameter only admits positive values greater than 0 (zero).

·J· Part length along the Y (ordinate) axis.

This parameter only admits positive values greater than 0 (zero).

·C· Centering type.

Optional parameter, by default 0. This parameter indicates on how many axes the part is to be centered. When the cycle centers the part on both axes, it is possible to select whether to measure or not the angle of the part with respect to the abscissa axis.

C=0	Center the part on both axes taking the angle of the part into account.
C=1	Part centering on a single axis (defined in parameter K).
C=2	Center the part on both axes ignoring the angle of the part.

·K· Axis and direction of the first probing movement.

Optional parameter, by default 0.

This parameter sets the axis on which the fist probing movement will take place.

K=0	The probe moves in the positive direction of the abscissa axis [G17(X) G18(Z) G19 (Y)].
K=1	The probe moves in the negative direction of the abscissa axis [G17(X) G18(Z) G19 (Y)].
K=2	The probe moves in the positive direction of the ordinate axis [G17(Y) G18(X) G19 (Z)].
K=3	The probe moves in the negative direction of the ordinate axis [G17(Y) G18(X) G19 (Z)].

·L· Surface coordinate measuring.

Optional parameter, by default 0. This parameter indicates whether the cycle must also measure the position of the top surface of the part.

L=0	The cycle does not measure the surface coordinate.
L=1	The cycle measures the surface coordinate.

·B· Safety distance.

Optional parameter; by default the distance between the part and the probe position when calling the cycle.

Distance with respect to the point to measure, to which the probe approaches before making the probing movement.

·D· Safety distance along Z.

This parameter only admits positive values greater than 0 (zero).

Distance for the probe to go up for its movements over the part.

·E· Withdrawal distance after initial probing.

This parameter only admits positive values greater than 0 (zero).

This parameter sets the distance the probe withdrawals after the first probing movement. Once it withdraws this distance, the CNC makes a second probing movement.

·H· Feedrate for the first probing movement.

Optional parameter; by default the lowest value of machine parameter PROBEFEED of the axes that make up the trihedron. This parameter only admits positive values greater than 0 (zero).

This parameter sets the feedrate for the first probing movement. Then, the CNC will repeat the probing movement at feedrate ·F·.

·F· Probing feedrate.

Optional parameter; by default 10% of the lowest value of machine parameter PROBEFEED of the axes that make up the trihedron. This parameter only admits positive values greater than 0 (zero).

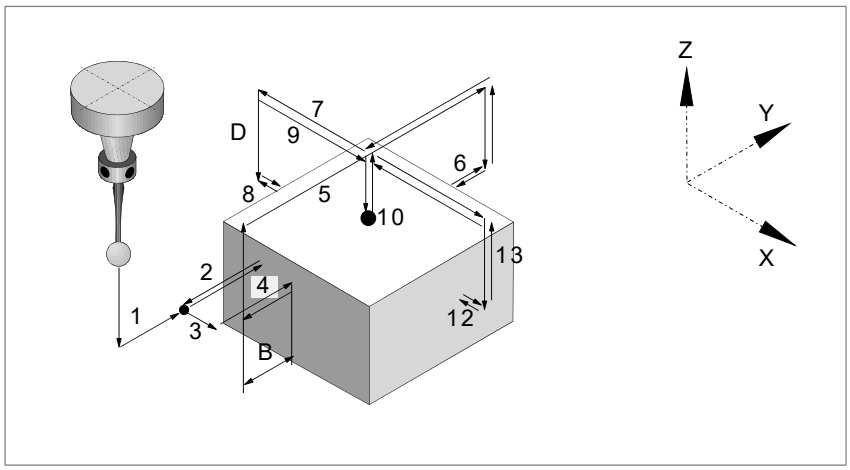
This parameter sets the feedrate for the second probing movement.

·Q· Feedrate for approach movements.

Optional parameter; by default, Feedrate G00.

This parameter sets the approaching feedrate.

3.10.2 Basic operation.



1 Approach movement (if any of parameters ·XYZ· has been defined).

Movement of the probe at feedrate ·Q· from the cycle calling point to the first approach point. defined by parameters ·XYZ·.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.

2 Probing of the first point

The cycle makes a first probing movement, a withdrawal movement and a second probing move. After the second probing move, the probe withdraws to the approach point.

First probing movement. Probe movement along the selected axis at feedrate ·H· until the probe signal is received.

Withdrawal movement. Return of the probe in rapid (G00) the distance indicated in ·E·.

Second probe. Probing movement at feedrate ·F· until the probe signal is received.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the starting point.

3 If C=0 has been programmed, repeat the probing somewhere else on the surface. The cycle uses this second probing to calculate the inclination of the part.

Approach movement. Movement parallel to the probed side to touch a different point of the same side, located at 1/4 the length programmed in ·J·.

Probing movement. Probing movement at feedrate ·F· until the probe signal is received.

Withdrawal movement. The probe returns to the safety position located at a ·B· distance from the part.

4 Approach movement to the second probing point.

Movement of the probe from the first approach point to the second located in front of the first one.

The perpendicular axis moves the ·D· distance up in rapid (G00). Then the probe moves over the part, at feedrate ·Q·, to the next approach point. The probe moves a ·D· distance back down at feedrate ·H·

For the movement, the cycle takes into consideration the length of the part and its inclination angle. If in this last movement, the probe touches the part, the CNC shows an error at the cycle parameter value.

5 Probing of the second point.

It is done like the previous one. The probe returns to the safety position located at a ·B· distance from the part.

6 If either C=0 or C=2 has been programmed, approach movement to the third probing point.

Movement of the probe from the second approach point to the center of the part and then to the third point.

7 If either C=0 or C=2 has been programmed, probing of the third point.

It is done like the previous one.

8 Approach movement to the top surface (only if it has been programmed).

When selecting to measure the surface (parameter ·L·), the probe moves the ·D· distance up in rapid (G00) and the probe moves to the center of the part.

9 Measuring the top surface (only if it has been programmed).

The probe makes a first probing move, a withdrawal movement and a second probing move. After the second probing move, the probe withdraws to the approach point.

First probing movement. Probing movement, at feedrate ·H·, until touching the part surface.

Withdrawal movement. Return of the probe in rapid (G00) the distance indicated in ·E·.

Second probe. Probing movement at feedrate ·F· until the probe signal is received.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the safety position.

10 If either C=0 or C=2 has been programmed, approach movement to the fourth probing point.

Movement of the probe from the third approach point to the fourth.

11 If either C=0 or C=2 has been programmed, probing of the fourth point.

It is done like the previous one.

12 Probe positioning at the calculated center.

This movement is carried out in two stages. It first moves along the perpendicular axis in rapid (G00) and then in the plane at feedrate ·Q·.

3.11 #PROBE 11. Circular part centering.

This cycle may be used to calculate the center of a circular part of known dimensions.

Requirements prior to the calibration.

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.

To execute this cycle, the probe must be properly calibrated.

Data returned by the cycle after the measurement.

Once the cycle has ended, the CNC returns the real values obtained in the measurement in the following arithmetic parameters:

P297 Part surface coordinate.

P298 Part center coordinate along the abscissa axis.

P299 Part center coordinate along the ordinate axis.

3.11.1 Programming the cycle.

The programming format for this cycle is. Optional parameters are indicated between angle brackets.

#PROBE 11 <X Y Z> J <K> <L> <B> D E <H> <F> <Q> <C>

X Y Z	Probe position to execute the cycle.
J	Part diameter.
C	Centering type.
K	Optional. Axis and direction of the first probing movement.
L	Optional. Surface coordinate measuring.
B	Optional. Safety distance.
D	Safety distance along Z.
E	Withdrawal distance after initial probing.
H	Optional. Feedrate for the initial probing movement.
F	Optional. Probing feedrate.
Q	Optional. Feedrate for approach movements.

·X Y Z· Probe position to execute the cycle.

Optional parameter; by default the probe position when calling the cycle.

The canned cycles may be programmed in any work plane. See *"Canned cycles and the work planes."* on page 42.

·J· Part diameter.

This parameter only admits positive values greater than 0 (zero).

·C· Centering type.

Optional parameter, by default 0. This parameter indicates on how many axes the part is to be centered.

C=0	Center the part on two axes.
C=1	Part centering on a single axis (defined in parameter K).

·K· Axis and direction of the first probing movement.

Optional parameter, by default 0.

This parameter sets the axis on which the fist probing movement will take place.

K=0	The probe moves in the positive direction of the abscissa axis [G17(X) G18(Z) G19 (Y)].
K=1	The probe moves in the negative direction of the abscissa axis [G17(X) G18(Z) G19 (Y)].
K=2	The probe moves in the positive direction of the ordinate axis [G17(Y) G18(X) G19 (Z)].
K=3	The probe moves in the negative direction of the ordinate axis [G17(Y) G18(X) G19 (Z)].

·L· Surface coordinate measuring.

Optional parameter, by default 0.

This parameter indicates whether the cycle must also measure the position of the top surface of the part.

L=0	The cycle does not measure the surface coordinate.
L=1	The cycle measures the surface coordinate.

·B· Safety distance.

Optional parameter; by default the distance between the part and the probe position when calling the cycle.

Distance with respect to the point to measure, to which the probe approaches before making the probing movement.

·D· Safety distance along Z.

This parameter only admits positive values greater than 0 (zero).

Distance for the probe to go up for its movements over the part.

·E· Withdrawal distance after initial probing.

This parameter only admits positive values greater than 0 (zero).

This parameter sets the distance the probe withdrawals after the first probing movement. Once it withdraws this distance, the CNC makes a second probing movement.

·H· Feedrate for the first probing movement.

Optional parameter; by default the lowest value of machine parameter PROBEFEED of the axes that make up the trihedron. This parameter only admits positive values greater than 0 (zero).

This parameter sets the feedrate for the first probing movement. Then, the CNC will repeat the probing movement at feedrate ·F·.

·F· Probing feedrate.

Optional parameter; by default 10% of the lowest value of machine parameter PROBEFEED of the axes that make up the trihedron. This parameter only admits positive values greater than 0 (zero).

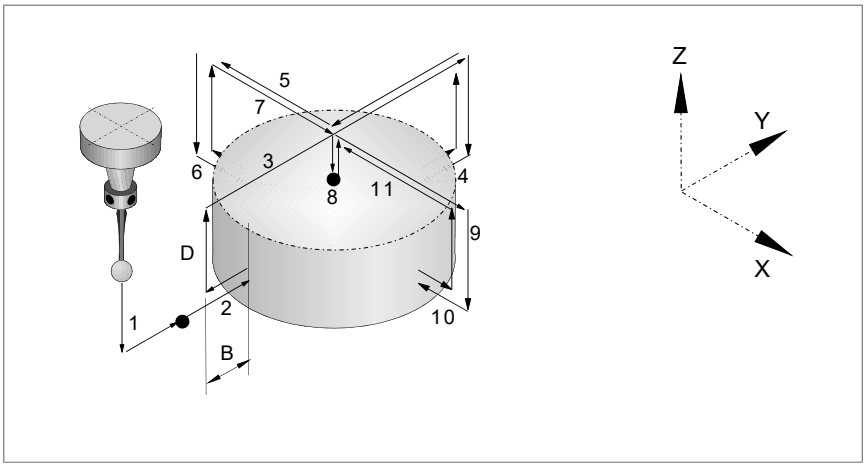
This parameter sets the feedrate for the second probing movement.

·Q· Feedrate for approach movements.

Optional parameter; by default, Feedrate G00.

This parameter sets the approaching feedrate.

3.11.2 Basic operation.



1 Approach movement (if any of parameters ·XYZ· has been defined).

Movement of the probe at feedrate ·Q· from the cycle calling point to the first approach point. defined by parameters ·XYZ·.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.

2 Probing of the first point

The cycles probes in four different points. In each point, it makes a first probing movement, a withdrawal movement and a second probing move. After the second probing move, the probe withdraws to the approach point.

First probing movement. Probe movement along the selected axis at feedrate ·H· until the probe signal is received.

Withdrawal movement. Return of the probe in rapid (G00) the distance indicated in ·E·.

Second probe. Probing movement at feedrate ·F· until the probe signal is received.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the safety position located at a ·B· distance from the part.

3 Approach movement to the second probing point.

Movement of the probe from the first approach point to the second located in front of the first one.

The perpendicular axis moves the ·D· distance up in rapid (G00). Then the probe moves over the part, at feedrate ·Q·, to the next approach point. The probe moves a ·D· distance back down at feedrate ·H·

If in this last movement, the probe touches the part, the CNC shows an error at the cycle parameter value.

4 Probing of the second point.

It is done like the previous one.

5 If C=0 has been programmed, approach movement to the third probing point.

Movement of the probe from the second approach point to the third.

6 If C=0 has been programmed, probing of the third point.

It is done like the previous one.

7 Approach movement to the top surface (only if it has been programmed).

When selecting to measure the surface (parameter ·L·), the probe moves the ·D· distance up in rapid (G00) and the probe moves to the center of the part.

8 Measuring the top surface (only if it has been programmed).

The probe makes a first probing move, a withdrawal movement and a second probing move. After the second probing move, the probe withdraws to the approach point.

First probing movement. Probing movement, at feedrate ·H·, until touching the part surface.

Withdrawal movement. Return of the probe in rapid (G00) the distance indicated in ·E·.

Second probe. Probing movement at feedrate ·F· until the probe signal is received.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the safety position.

9 If C=0 has been programmed, approach movement to the fourth probing point.

Movement of the probe from the third approach point to the fourth.

10 If C=0 has been programmed, probing of the fourth point.

It is done like the previous one.

11 Probe positioning at the calculated center.

This movement is carried out in two stages. It first moves along the perpendicular axis in rapid (G00) and then in the plane at feedrate ·Q·.

3.12 #PROBE 12. Tabletop probe calibration.

This cycle may be used to calibrate a tabletop probe using a tool that has been calibrated in radius and length.

Requirements prior to the calibration.

The tool must be properly calibrated in radius and length. The machine parameters values of the probe must be close to its real values.

Data returned by the cycle after the measurement.

Once the cycle has ended, the CNC returns the real values of the probe obtained in the measurement in the following arithmetic parameters:

P295 Real coordinate of the probe side closest to the origin, along the abscissa axis.

P296 Real coordinate of the probe side furthest from the origin, along the abscissa axis.

P297 Real coordinate of the probe side closest to the origin, along the ordinate axis.

P298 Real coordinate of the probe side furthest from the origin, along the ordinate axis.

P299 Real coordinate of the probe side measured along the longitudinal axis.

The cycle returs the value of parameter PROBEDELAY for all the axes and the spindle in the following arithmetic parameter.

P294 Best value of parameter PROBEDELAY for all the axes and the spindle.

On a tabletop probe, it is also necessary to set the delay of the probe signal (parameter PROBEDELAY) for the spindle.

3.12.1 Programming the cycle.

The programming format for this cycle is. Optional parameters are indicated between angle brackets.

#PROBE 12 B E <H> <F> <I> <X U Y V Z W>

B	Safety distance.
E	Withdrawal distance after initial probing.
H	Optional. Feedrate for the initial probing movement.
F	Optional. Probing feedrate.
I	Optional. Type of calibration
X..W	Optional. Theoretical position of the tabletop probe.

·B· Safety distance.

This parameter only admits positive values greater than 0 (zero).

Distance with respect to the point to measure, to which the probe approaches in G00 before making the probing movement.

·E· Withdrawal distance after initial probing.

This parameter only admits positive values greater than 0 (zero).

This parameter sets the distance the probe withdrawals after the first probing movement. Once it withdraws this distance, the CNC makes a second probing movement.

·H· Feedrate for the first probing movement.

Optional parameter; by default the lowest value of machine parameter PROBEFEED of the axes that make up the trihedron. This parameter only admits positive values greater than 0 (zero).

This parameter sets the feedrate for the first probing movement. Then, the CNC will repeat the probing movement at feedrate ·F·.

·F· Probing feedrate.

Optional parameter; by default 10% of the lowest value of machine parameter PROBEFEED of the axes that make up the trihedron. This parameter only admits positive values greater than 0 (zero).

This parameter sets the feedrate for the second probing movement.

·I· Type of calibration.

Optional parameter, by default 0.

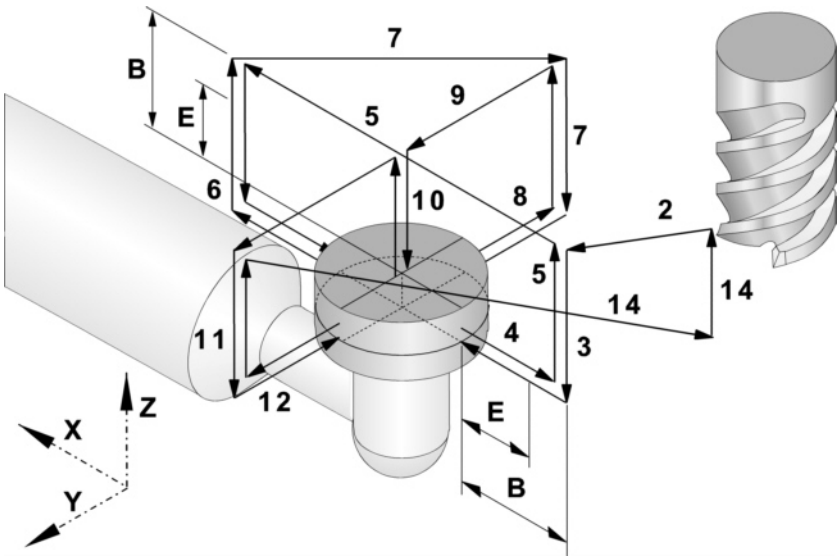
I=0	Simple calibration. The probe calibrates each side of the probe.
I=1	Double calibration. The probe does two calibrations on each side of the probe, one with the spindle oriented to 0º and the other with the spindle oriented to 180º in order to avoid any errors due to a possible tool eccentricity.

·X U Y V Z W· Tabletop probe position.

Parameters X Y Z refer to the minimum coordinates of the probe on the first axis, second axis and on the axis perpendicular to the plane respectively. Parameters U V W refer to the maximum coordinates of the probe on the first axis, second axis and on the axis perpendicular to the plane respectively.

This data does not modify the machine parameters. The CNC takes this data into account only during this calibration. If any of this data is left out, the CNC takes the value assigned to the corresponding machine parameter.

3.12.2 Basic operation.



Initial considerations.

Relative tool position with respect to the probe along the longitudinal axis.

If the Z coordinate of the tool before the cycle is greater than or equal to the maximum Z coordinate of the probe, the Z side of the probe that will be measured first will correspond to its maximum Z coordinate.

If the Z coordinate of the tool before the cycle is less than or equal to the minimum Z coordinate of the probe, the Z side of the probe that will be measured first will correspond to its minimum Z coordinate.

If the Z coordinate of the tool before the cycle is between the maximum and minimum Z coordinates of the probe, the cycle will issue the error "Wrong Z coordinate of the starting plane".

Relative tool position with respect to the probe along the abscissa axis.

If the X coordinate of the tool before the cycle is greater than or equal to the average of the minimum and maximum X coordinate of the probe, the X side of the probe that will be measured first will correspond to the maximum X coordinate. Otherwise, the calibration will begin on the X side corresponding to the minimum X coordinate.

Basic operation.

1. If the cycle is going to do a double calibration, the spindle is oriented to 0º.
2. Rapid probe movement (G00) from the cycle calling point to the safety point. The safety point will be located at a distance ·B· over the probe and at a distance ·B· from the center of the tool to the point to be probed.

   This approach movement is made in two stages. If the tool is separated from the Z side of the probe a shorter distance than the one indicated in parameter ·B·, the movement is carried out first along the perpendicular axis and then in the work plane. Otherwise, It first moves in the work plane and then along the axis perpendicular to the plane.
3. Approach movement. Tool movement along the perpendicular axis at feedrate ·H· up to the probing Z coordinate (in the middle of the probe). If the tool touches the probe, the cycle will display the corresponding error message.

4 Probing of the first point

The probe touches in five different points. In each point, the cycle makes a first probing movement, a withdrawal movement and a second probing move. After the second probing move, the probe withdraws to the approach point.

First probing movement. Tool movement along the abscissa axis at feedrate ·H· until the probe signal is received. Moving direction according to initial considerations.

Withdrawal movement. Return of the tool in rapid (G00) the distance indicated in ·E·.

Second probe. Tool movement at the indicated feedrate ·F· until the probe signal is received.

Withdrawal movement. Rapid tool movement (G00) from the probing point to the safety position located at a ·B· distance from the probe.

5 Approach movement to the second probing point.

Tool movement from the first approach point to the second located in front of the first one.

The perpendicular axis moves in rapid (G00) up to the approach coordinate. Then, the tool moves in rapid (G00) over the probe up to the next approach point. The tool moves back down to the probing Z coordinate at feedrate ·H·.

For the movement, the cycle takes into consideration the theoretical length of the probe and parameter ·B·. If in this last movement, the tool touches the probe, the tool goes back up to the approach coordinate and moves the safety distance ·B· in the same direction. The cycle repeats this operation until the tool releases the probe.

6 Probing of the second point.

It is done along the abscissa axis same as the previous one.

7 Approach movement to the third probing point.

Tool movement from the second approach point to the third, located in front of the minimum Y side of the probe. Position of the abscissa axis corresponding to the real center coordinate of the probe.

8 Probing of the third point.

It is done along the ordinate axis same as the previous one.

9 Approach movement to the fourth approach point (top surface).

The tool moves up in raid (G00) up to the approach coordinate, it then moves in rapid (G00) along the Y axis up to the theoretical center of the probe.

The perpendicular axis moves in rapid (G00) up to the approach coordinate. Then, the tool moves in rapid (G00) along the ordinate axis and over the probe up to the theoretical center of the probe.

10 Measurement of the top surface.

The tool makes a first probing movement, a withdrawal movement and a second probing move. After the second probing move, the tool withdraws to the approach point.

First probing movement. Tool movement along the perpendicular axis at feedrate ·H· until the probe signal is received.

Withdrawal movement. Return of the tool in rapid (G00) the distance indicated in ·E·.

Second probe. Tool movement at the indicated feedrate ·F· until the probe signal is received.

Withdrawal movement. Rapid tool movement (G00) from the probing point to the safety position located at a ·B· distance from the probe.

11 Approach movement to the fifth probing point.

Tool movement from the forth approach point to the fifth, located in front of the third one.

The tool moves in rapid (G00) to the next approach point. The tool moves back down to the probing Z coordinate at feedrate ·H·.

For the movement, the cycle takes into consideration the theoretical length of the probe and parameter ·B·. If in this last movement, the tool touches the probe, the tool goes back up to the approach coordinate and moves the safety distance ·B· in the same direction. The cycle repeats this operation until the tool releases the probe.

12 Probing of the fifth point.

It is done along the ordinate axis same as the third probing point.

13 Withdrawal movement.

The perpendicular axis moves in rapid (G00) up to the approach coordinate.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2106

14 Withdrawal movement.

Rapid probe movement (G00) up to the cycle calling point.

This withdrawal movement is made in two stages. It first moves in the work plane and then along the axis perpendicular.

15 If the cycle is going to do a double calibration, the spindle is oriented to 180º and the cycle repeats steps 2 through 14.

3.13 Check the data of the canned cycles (variables).

Check which parameters have been programmed

(V.)C.CALLP_a-z

This variable can only be read from the part program or via MDI. The variable is evaluated during block preparation.

This variable indicates whether parameters A-Z have been programmed or not in the calling instruction; value of ·1· if so and ·0· if otherwise.

```
#PROBE 3 X12.5 Y20 Z23.75 B5 F100
    V.C.CALLP_X = 1
    V.C.CALLP_Y = 1
    V.C.CALLP_Z = 1
    V.C.CALLP_B = 1
    V.C.CALLP_K = 0
    V.C.CALLP_F = 1
```

Check the value of the programmed parameters.

(V.)C.a-z

Variable that can be read and written from the part-program or MDI. The variable is evaluated during block preparation.

This variable returns the value of parameters A-Z programmed in the calling instruction.

```
#PROBE 3 X12.5 Y20 Z23.75 B5 F100
    V.C.X = 12.5
    V.C.Y = 20
    V.C.Z = 23.75
    V.C.B = 5
    V.C.F = 100
```


FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2106

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2106

CANNED CYCLES. CYCLE EDITOR.

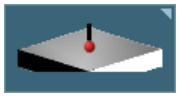
4

Canned cycles may be edited in ISO code (described in this chapter) or with using the cycle editor. See chapter *"3 Canned cycles. ISO coded programming."*.

The cycles may be defined anywhere in the program, that is, in the main program as well as in a subroutine.

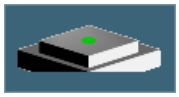
Programming the cycles of the editor.

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



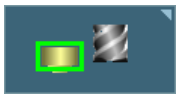
Measurement.

Measurement of the surface, outside or inside corner, of the angle on the abscissa axis, of the outside corner and angle, of circular hole or boss.



Part centering.

Rectangular or circular part centering.



Probe calibration.

Tabletop probe calibration and the measuring probe.

Tool calibration.

Calibrating the length and/or radius of a tool. Measuring the length and/or radius wear of a tool.

Probing canned cycles are not modal; therefore, they must be programmed every time any of them is to be executed. The execution of these cycles does not change the program history.

Cycle data programming.

The cycle number and the rest of parameters may be defined with a number, an arithmetic parameter or expression whose result is a number. See *"4.1 How to define the data of the editor."* on page 95.

Limitations for executing the cycles.

These cycles cannot be executed if tool radius compensation is active.

Canned cycles and the work planes.

Tool and probe calibrations cycles are carried out in the G17, G18 and G19 planes. The rest of the cycles can also be executed in any plane defined with function G20.

When working in a plane other than the XYZ, the CNC interprets the canned cycle parameters as follows:

Parameter	X-Z plane	W-X plane	A-B plane
Parameter X and all related to it, with the abscissa axis.	X axis	W axis	A axis
Parameter ·Y· and all related to it, with the ordinate axis.	Z axis	X axis	B axis
Parameter ·Z· and all related to it, with the axis perpendicular to the plane.			

4.1 How to define the data of the editor.

To enter or modify a data, it must be selected; i.e. it must have the editing focus on it. The parameters of the cycles may be selected with the [⬅] [➡] [⬆] or [⬇] keys, or with the direct access keys. The first data of each group may also be selected by pressing the page-up and page-down keys.

The direct access keys correspond to the name of the parameters; [F] for forward movements, [T] for tools, etc. Each time the same key is pressed, the next value of the same type is selected.

Manual data entry.

- To modify a numerical data, key in the desired value or press [DEL] to leave the data undefined. In either case, press [ENTER] for the cycle to assume the new value.
- Press the [SPACE] key to change the status of this icon.

Leaving some data undefined.

Some data may be left undefined (empty checkbox). In this case, the cycle behaves as follows.

- If the cycle position is not defined, it is executed at the current position the axes when calling the cycle.
- If the tool number is not defined, it will be executed with the tool that is active at the time of execution.

Defining data using arithmetic parameters..

Numerical data may be defined using global arithmetic parameters (P100-P9999) or common ones (P10000-P19999). In this case, when executing the cycle, these data will assume the value that the parameter has at the time.

When using global parameters, bear in mind that some cycles modify the value of these parameters at the end of the execution. Refer to each cycle to see which parameters it modifies.

Teach-in mode for data entry.

The Teach-in mode is activated from the horizontal softkey menu. When the Teach-in mode is active, the bottom of the screen shows a window with the axes of the channel.

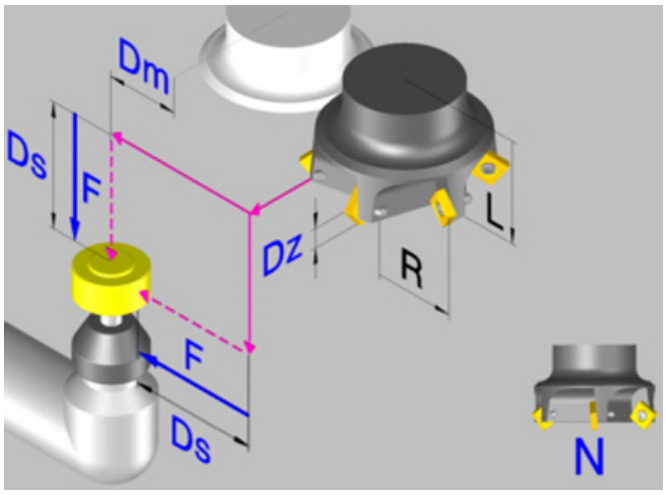
The Teach-in mode may be used to jog the axes and assign their current position to the data that define the position of the cycle. The axes may be jogged using the jog keypad, the handwheels or via MDI.

To assign a value to a data, select it with the cursor (focus on it) and press the [RECALL] key. The data is taken from the channel where the editing-simulation mode is active.

- The X axis related data takes the coordinate of the first axis of the channel.
- The Y axis related data takes the coordinate of the second axis of the channel.
- The Z axis related data takes the coordinate of the third axis of the channel.

4.2 Tool calibration (dimensions and wear)

It is used to calibrate the tool of the spindle in length or in radius. The desired operation is selected from the horizontal softkey menu. The screen will show the help graphics for the selected operation and the cycle will show the necessary data to define it.



Calibrate the length of a tool.

Calibrate the radius of a tool.

Tool length and radius calibration.

Measuring the length wear of a tool.

Measure the radius wear of a tool.

Measuring the length and radius wear of a tool.

Once the cycle has concluded, it updates the dimensions in the tool table and the tool wears.

Requirements prior to the calibration.

If it is the first time that the tool or the probe is being calibrated, you must enter its approximate dimensions in the tool table. If it is a probe, the "R" value will correspond to the radius of the probe ball.

Tabletop probe.

Executing this cycle requires a table-top probe, installed in a fixed position of the machine and with its sides parallel to the axes of the plane. The probe position must be given in absolute coordinates referred to machine reference zero using the machine parameters PRB1MIN, PRB1MAX, PRB2MIN, PRB2MAX, PRB3MIN, PRB3MAX.

Data returned by the cycle after a calibration.

Once the calibration has concluded, the cycle updates the dimensions in the tool table and initializes the tool wears to 0 (zero). The cycle only updates the value of the calibrated dimensions; if only the radius has been calibrated, it does not change the length and vice versa.

Once the cycle is over, the CNC will return the detected error in the following arithmetic parameters. A detected error is the difference between the measured value and the one assigned earlier in the table.

P298 Error detected on the tool radius.

P299 Error detected on the tool length.

If when calling the cycle, the calibration of each cutting edge is requested, (parameter ·N·), the lengths are assigned to arithmetic parameters P271 and on; the radii are assigned to arithmetic parameters P251 and on.

Data returned by the cycle after a wear measurement.

Once the wear measurement has ended, the cycle updates in the tool table the measured wears; it does not modify the length value or the radius value. The cycle only updates the value of the measured wear; if only the radius wear has been measured, it does not change the length wear and vice versa.

Once the cycle has ended and if a maximum wear has been set (parameters ·Lw· and ·Rw·), the CNC compares the measured wear with the maximum allowed.

- If the measured wear exceeds the maximum allowed, the CNC sets the "expired tool" indicator. When defining the cycle, it is possible to define whether the CNC interrupts the cycle or whether it changes the tool with another one of the same family.
- If the measured wear exceeds the maximum allowed, the cycle will returns the measured wear in the following arithmetic parameters.

  P298 Radius wear.

  P299 Length wear.

If when calling the cycle, the wear measurement of each cutting edge was requested (parameter ·N·), the length wears are assigned to arithmetic parameters P271 and on; the radius wears are assigned to arithmetic parameters P251 and on.

4.2.1 Programming the cycle.

Tool to be calibrated.

·T· Tool to be calibrated.

Number of the tool to be calibrated. The tool must be defined in tool table.

·D· Tool offset.

Offset of the tool to be calibrated.

Probing movement.

·icon· Probe side to be used.

This parameter indicates which side of the probe will be used to calibrate or measure the radius wear.

Calibration on the X+ side.

Calibration on the X- side.

Calibration on the Y+ side.

Calibration on the Y- side.

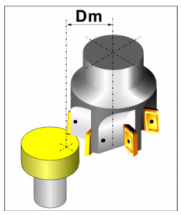
·Ds· Safety distance.

This parameter only admits positive values greater than 0 (zero).

Distance with respect to the probing point that the tool approaches in G00 before touching the probe. The tool must be placed at a distance greater than this value when the cycle is called.

·Dm· Distance from the probing point to the tool shaft.

Optional parameter; by default the tool radius.

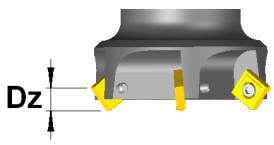


This parameter indicates the distance from the tool shaft to the tool probing point.

To calibrate along the tool shaft, set this parameter to zero.

·Dz· Distance from the probing point to the tool base.

Optional parameter, by default 0.



This parameter indicates the distance from the tool base to the tool probing point.

·F· Probing feedrate.

Optional parameter; by default the feedrate defined in machine parameter PROBEFEED.

This parameter sets the probing feedrate. The rest of the movements will be carried out in G00.

·S· Tool turning speed and direction.

This parameter sets the speed and turning direction of the tool; the sign sets the turning direction (positive if M3 and negative if M4). If this parameter is set, select a climb cutting direction.

Tool length calibration along the tool shaft (parameter Dm = 0) should be done with the spindle stopped.

·N· Number of edges to measure.

This parameter is only valid when parameter ·S· has been defined with a value greater than 0 (zero). This parameter indicates the number of cutting edges to be measured.

N=0 The CNC knows the location of an edge and it only probes once. In this case, the spindle speed must be zero.

N>0 The CNC makes an initial touch to locate an edge; then, stops the spindle and makes a measurement of each edge. In this case, the spindle speed and parameter ·Dm· must be programmed.

Wear measurement.

·Lw· Maximum length wear allowed.

If this parameter is not set or is set to zero, The CNC does not reject the tool because of length wear.

If the measured wear is lower than the value defined in this parameter, the CNC updates the tool table with the measured value. If the measured wear exceeds the defined value, the CNC rejects the tool and acts according to the criteria set by the icon that comes with this parameter.

·Rw· Maximum radius wear allowed.

If this parameter is not set or is set to zero, The CNC does not reject the tool because of radius wear.

If the measured wear is lower than the value defined in this parameter, the CNC updates the tool table with the measured value. If the measured wear exceeds the defined value, the CNC rejects the tool and acts according to the criteria set by the icon that comes with this parameter.

·icon· Behavior when exceeding the maximum of wear.

This parameter is only valid when parameter ·Lw· or ·Rw· has been defined with a value greater than 0 (zero).

This parameter indicates what the CNC does when the measured wear exceeds the defined value. In any case, the CNC sets the "expired tool" indicator in the tool table.

The CNC shows a message of rejected tool and stops the cycle.

The cycle replaces the worn out tool with another one of the same family.

Probe coordinates.

·icon· Redefine the tabletop probe position.

The probe position is assumed from the machine parameters.

The probe position is defined in the cycle.

When selecting this option, the cycle will show the data necessary to define the probe position.

·PRB1MIN - PRB3MAX· Tabletop probe position.

They are optional parameters that usually need not be defined. On certain machines, due to lack of repeatability in the mechanical positioning of the probe, the probe must be calibrated again before each tool calibration. Instead of re-defining the machine parameters every time the probe is calibrated, those coordinates may be indicated in these parameters.

Parameters PRB1MIN, PRB2MIN and PRB3MIN refer to the minimum coordinates of the probe on the first axis, second axis and on the axis perpendicular to the plane respectively. Parameters PRB1MAX, PRB2MAX and PRB3MAX refer to the maximum coordinates of the probe on the first axis, second axis and on the axis perpendicular to the plane respectively.

This data does not modify the machine parameters. The CNC takes this data into account only during this calibration. If any of this data is left out, the CNC takes the value assigned to the corresponding machine parameter.

4.2.2 Basic operation.

Calibrate or measure tool length wear.

1 Approach movement.

Rapid probe movement (G00) from the cycle calling point to the safety point. The safety point will be located at a distance ·Ds· over the probe and at a distance ·Dm· from the center of the tool to the point to be probed.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.

2 Movement to locate an edge (only if ·N· has been defined with a value other than 0).

The CNC starts the spindle at an ·S· speed and moves the tool at the feedrate ·F· along the axis perpendicular to the active plane until receiving the probe signal. Once it knows the position of an edge, the tool withdraws to the safety point and stops the spindle.

3 Probing movement.

Probing movement along the axis perpendicular to the active plane at the feedrate ·F· until the probe signal is received. Withdrawal to the safety point.

If ·N· has been defined to a value other than 0, this movement is repeated ·N· times.

4 Withdrawal movement.

Movement in the work plane up to the cycle calling point.

Calibrate or measure tool radius wear.

1 Approach movement.

Rapid probe movement (G00) from the cycle calling point to the safety point. The safety point will be located at a ·Ds· distance from the probe.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.

2 Movement to locate an edge (only if ·N· has been defined with a value other than 0).

The CNC starts the spindle at an ·S· speed and moves the tool at the feedrate ·F· along the selected axis until receiving the probe signal. The CNC stops the spindle and carries out some probing moves to locate the edge more precisely. Once it knows the position of an edge, the tool withdraws to the safety point.

3 Probing movement.

Probing movement at the indicated feedrate ·F· until the probe signal is received. Withdrawal to the safety point.

If ·N· has been defined to a value other than 0, this movement is repeated ·N· times.

4 Withdrawal movement.

This movement is carried out in two stages. It first moves in the work plane and then along the perpendicular axis and then in the work plane.

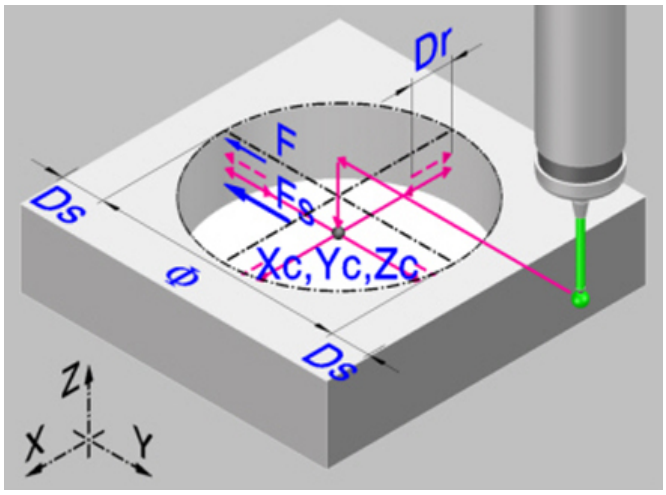
Calibrate or measure tool radius or length wear.

It carries out the movements of the previous sections sequentially; it first calibrates and/or measures the radius and then the length. The tool returns to the cycle calling point only after measuring the tool length.

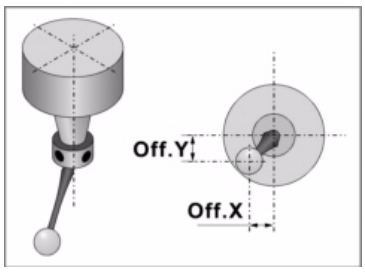
When the number of edges ·N· is other than zero, the position of the edges is known after measuring or calibrating the radius, thus not being necessary to do it when measuring or calibrating the length.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2106

4.3 Measurement probe calibration.



This cycle may be used to calibrate the probe located in the spindle. This probe will be the one used in the tool calibration canned cycles.



The cycle measures the deviation of the probe ball with respect to the tool-holder axis and the electronic radius of the probe. The electronic radius is the deflection of the probe since it makes contact until the probing signal is activated.

For the calibration, a hole previously machined and with known center and dimensions is used.

Requirements prior to the calibration.

The probe tool must be calibrated in length and its dimensions defined in the tool table. The fields of the tool table for each probe will have the following meaning:

R	Probe ball radius. This value will be loaded into the table manually.
L	Probe length. This value will be indicated by the tool length calibration cycle.
Off.X	Deviation of the probe ball shaft with respect to the tool holder along the abscissa axis. This value will be assigned by this cycle.
Off.Y	Deviation of the probe ball shaft with respect to the tool holder along the ordinate axis. This value will be assigned by this cycle.
RW	Electronic radius. Deflection of the probe since it makes contact until the probing signal is activated. The electronic radius is set with a negative value.

The following steps must be followed for calibrating the probe:

1. Once the probe characteristics have been checked, manually enter the ball radius value (R) in the tool table.
2. After selecting the corresponding tool number and tool offset, execute the tool length calibrating cycle. This cycle updates the length value and resets the "Off.Z" value to zero.
3. Execute the probe calibration canned cycle that updates the "Off.X" and "Off.Y" values (deviation of the axis of the probe ball with respect to the axis of the tool holder) and RW (electronic radius).

Data returned by the cycle after the measurement.

Once the cycle has concluded, it updates the tool table with the value tool offset that is currently active. Offsets "Off.X" and "Off.Y" for the abscissa and ordinate axes respectively indicate the deviation of the probe ball. "RW" indicates the deflection of the probe since it makes contact until the probing signal is activated.

Likewise, the CNC returns the best value to be assigned to axis machine parameter PROBEDELAY in the following arithmetic parameters.

P299 Best value of parameter PROBEDELAY for all the axes.

4.3.1 Programming the cycle.

Probe data.

·Tp· Number of the tool that identifies the probe.

Optional parameter, if not defined, use the probe that is in the spindle when executing the cycle.

Number of the tool used to define the probe in the tool table.

·Dp· Number of the tool offset that identifies the probe.

Offset associated with the probe, used to execute the cycle.

Probing movement.

·Xc Yc· Real coordinates of the hole center.

Optional parameter; if not programmed, the cycle takes the probe position as theoretical center of the hole.

The canned cycles may be programmed in any work plane. See *"Canned cycles and the work planes."* on page 93.

·Zc· Coordinates along the longitudinal axis where probing takes place.

Optional parameter; if not programmed, the cycle takes the probe position as the coordinate where probing will take place.

·Ø· Real diameter of the hole.

This parameter only admits positive values greater than 0 (zero).

This parameter sets the real diameter of the hole. This cycle may be used to measure holes with diameters no larger than "Ø+Ds".

·Ds· Safety distance.

This parameter only admits positive values greater than 0 (zero).

·Dr· Withdrawal distance after initial probing.

This parameter only admits positive values greater than 0 (zero).

This parameter sets the distance the probe withdrawals after the first probing movement. Once it withdraws this distance, the CNC makes a second probing movement.

·Fs· Feedrate for the first probing movement.

This parameter sets the feedrate for the first probing movement. Then, the CNC will repeat the probing movement at feedrate ·F·.

·F· Probing feedrate.

This parameter sets the feedrate for the second probing movement. The rest of the movements are carried out in G00 except the first probing movement which is carried out at feedrate ·Fs·.

Programming of M functions.

·M before· M functions to be executed before the cycle.

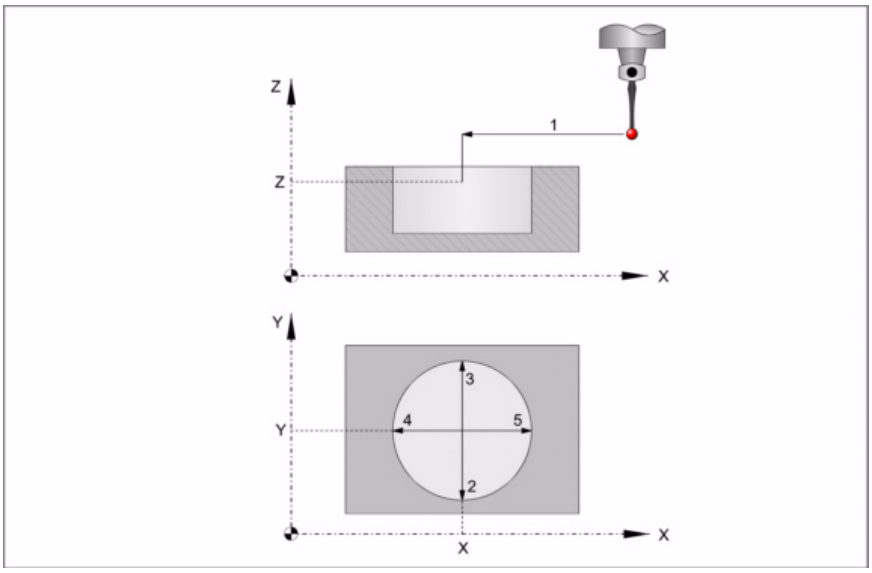
The cycle allows executing up to 4 M functions before the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

·M after· M functions to be executed after the cycle.

The cycle allows executing up to 4 M functions after the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

We recommend using these functions, for example, to manage wireless probes. Wireless probes are not always active, they have to be turned on before using the probing cycles and turned off afterwards. For this type of probes, set an M function to turn the probe on and another one to turn it off. Having the probe turn on/off programmed with M functions inside the cycle avoids executing the cycle without having the probe active or leaving the probe always active after executing the cycle.

4.3.2 Basic operation.



1 If the approach coordinates Xc Yc Zc have been programmed, approach movement. Probe's rapid movement (G00) from the cycle calling point to the center of the hole.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.

2 Probing of the first point

The probe touches in four different points. In each point, it makes a first probing movement, a withdrawal movement and a second probing move. After the second probing move, the probe returns to the center of the hole.

First probing movement. Probe movement along the ordinate axis at feedrate ·Fs· until the probe signal is received. The maximum probing distance is "Ds+Ø/2". If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.

Withdrawal movement. Probe's rapid movement (G00) the distance indicated by ·Dr·.

Second probe. Probe movement along the ordinate axis at feedrate ·F· until the probe signal is received.

Withdrawal movement to the center of the hole. Rapid probe movement (G00) from the probing point to the center of the hole.

3 Probing of the second point.

It is done along the ordinate axis same as the previous one.

4 Probing of the third point.

It is done along the abscissa axis same as the previous one.

5 Probing of the fourth point.

It is done along the abscissa axis same as the previous one.

6 180º probe rotation. The probe touches again in the previous four points.

It is up to the user to make sure that the probe can rotate without danger.

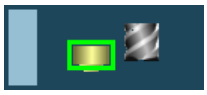
- *The hole must be large enough to let the probe rotate taking into account its real offsets.*
- *If the probe has a cable, check that the cable does not hinder the rotation of the probe.*

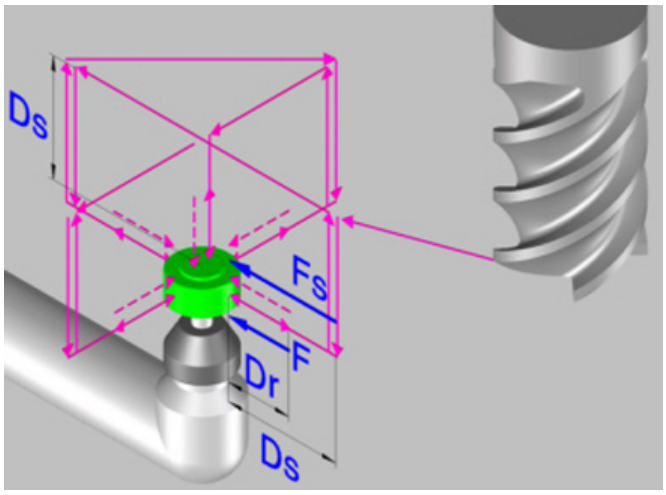
7 Rotation of the probe to its previous position.

8 Withdrawal movement. Rapid probe movement (G00) from the center of the hole to the cycle calling point.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane and then in the work plane.

4.4 Tabletop probe calibration.





This cycle may be used to calibrate a tabletop probe using a tool that has been calibrated in radius and length.

Requirements prior to the calibration.

The tool must be properly calibrated in radius and length. The values of the machine parameters of the probe must be close to their real values.

Data returned by the cycle after the measurement.

Once the cycle has ended, the CNC returns the real values of the probe obtained in the measurement in the following arithmetic parameters:

P295 Real coordinate of the probe side closest to the origin, along the abscissa axis.

P296 Real coordinate of the probe side furthest from the origin, along the abscissa axis.

P297 Real coordinate of the probe side closest to the origin, along the ordinate axis.

P298 Real coordinate of the probe side furthest from the origin, along the ordinate axis.

P299 Real coordinate of the probe side measured along the longitudinal axis.

The cycle returs the value of parameter PROBEDELAY for all the axes and the spindle in the following arithmetic parameter.

P294 Best value of parameter PROBEDELAY for all the axes and the spindle.

On a tabletop probe, it is also necessary to set the delay of the probe signal (parameter PROBEDELAY) for the spindle.

4.4.1 Programming the cycle.

Tool data.

·Tp· Tool number.

Tool number. Optional parameter, if not defined or set to 0, use the tool that is in the spindle when executing the cycle.

·Dp· Number of the tool offset that identifies the probe.

Offset associated with the tool, used to execute the cycle.

Probing movement.

·Ds· Safety distance.

This parameter only admits positive values greater than 0 (zero).

·Dr· Withdrawal distance after initial probing.

This parameter only admits positive values greater than 0 (zero).

This parameter sets the distance the probe withdrawals after the first probing movement. Once it withdraws this distance, the CNC makes a second probing movement.

·Fs· Feedrate for the first probing movement.

Optional parameter; by default the lowest value of machine parameter PROBEFEED of the axes that make up the trihedron.

This parameter sets the feedrate for the first probing movement. Then, the CNC will repeat the probing movement at feedrate ·F·.

·F· Probing feedrate.

Optional parameter; by default 10% of the lowest value of machine parameter PROBEFEED of the axes that make up the trihedron.

This parameter sets the feedrate for the second probing movement. The rest of the movements are carried out in G00 except the first probing movement which is carried out at feedrate ·Fs·.

·icon· Probing type.

Simple calibration. The probe calibrates each side of the probe.

Double calibration. The probe does two calibrations on each side of the probe, one with the spindle oriented to 0º and the other with the spindle oriented to 180º in order to avoid any errors due to a possible tool eccentricity.

Probe coordinates.

·icon· Redefine the tabletop probe position.

The probe position is assumed from the machine parameters.

The probe position is defined in the cycle.

When selecting this option, the cycle will show the data necessary to define the probe position.

·PRB1MIN - PRB3MAX· Tabletop probe position.

They are optional parameters that usually need not be defined. On certain machines, due to lack of repeatability in the mechanical positioning of the probe, the probe must be

calibrated again before each tool calibration. Instead of re-defining the machine parameters every time the probe is calibrated, those coordinates may be indicated in these parameters.

Parameters PRB1MIN, PRB2MIN and PRB3MIN refer to the minimum coordinates of the probe on the first axis, second axis and on the axis perpendicular to the plane respectively. Parameters PRB1MAX, PRB2MAX and PRB3MAX refer to the maximum coordinates of the probe on the first axis, second axis and on the axis perpendicular to the plane respectively.

This data does not modify the machine parameters. The CNC takes this data into account only during this calibration. If any of this data is left out, the CNC takes the value assigned to the corresponding machine parameter.

Programming of M functions.

·M before· M functions to be executed before the cycle.

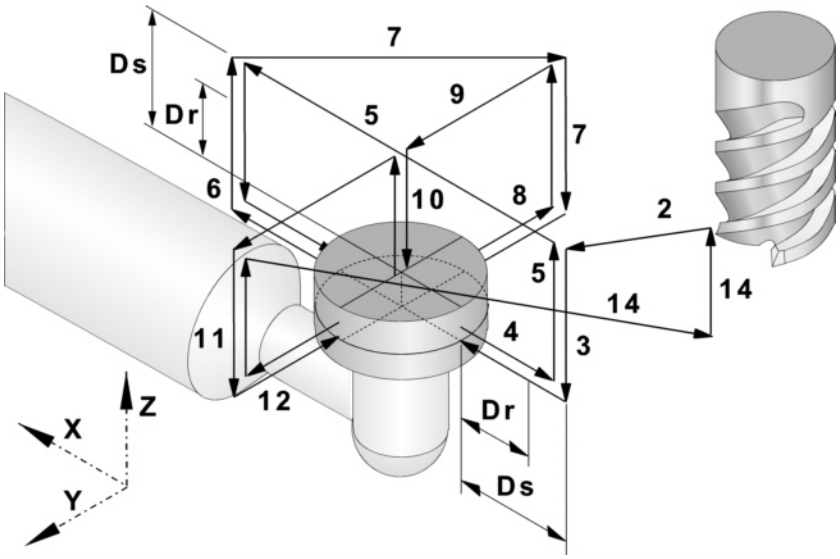
The cycle allows executing up to 4 M functions before the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

·M after· M functions to be executed after the cycle.

The cycle allows executing up to 4 M functions after the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

We recommend using these functions, for example, to manage wireless probes. Wireless probes are not always active, they have to be turned on before using the probing cycles and turned off afterwards. For this type of probes, set an M function to turn the probe on and another one to turn it off. Having the probe turn on/off programmed with M functions inside the cycle avoids executing the cycle without having the probe active or leaving the probe always active after executing the cycle.

4.4.2 Basic operation.



Initial considerations.

Relative tool position with respect to the probe along the longitudinal axis.

If the Z coordinate of the tool before the cycle is greater than or equal to the maximum Z coordinate of the probe, the Z side of the probe that will be measured first will correspond to its maximum Z coordinate.

If the Z coordinate of the tool before the cycle is less than or equal to the minimum Z coordinate of the probe, the Z side of the probe that will be measured first will correspond to its minimum Z coordinate.

If the Z coordinate of the tool before the cycle is between the maximum and minimum Z coordinates of the probe, the cycle will issue the error "Wrong Z coordinate of the starting plane".

Relative tool position with respect to the probe along the abscissa axis.

If the X coordinate of the tool before the cycle is greater than or equal to the average of the minimum and maximum X coordinate of the probe, the X side of the probe that will be measured first will correspond to the maximum X coordinate. Otherwise, the calibration will begin on the X side corresponding to the minimum X coordinate.

Basic operation.

1. If the cycle is going to do a double calibration, the spindle is oriented to 0º.
2. Rapid probe movement (G00) from the cycle calling point to the safety point. The safety point will be located at a distance ·Ds· over the probe and at a distance ·Ds· from the center of the tool to the point to be probed.

   This approach movement is made in two stages. If the tool is separated from the Z side of the probe a shorter distance than the one indicated in parameter ·Ds·, the movement is carried out first along the perpendicular axis and then in the work plane. Otherwise, It first moves in the work plane and then along the axis perpendicular to the plane.
3. Approach movement. Tool movement along the perpendicular axis at feedrate ·Fs· up to the probing Z coordinate (in the middle of the probe). If the tool touches the probe, the cycle will display the corresponding error message.

4 Probing of the first point

The probe touches in five different points. In each point, the cycle makes a first probing movement, a withdrawal movement and a second probing move. After the second probing move, the probe withdraws to the approach point.

First probing movement. Tool movement along the abscissa axis at feedrate ·Fs· until the probe signal is received. Moving direction according to initial considerations.

Withdrawal movement. Return of the tool in rapid (G00) the distance indicated in ·Dr·.

Second probe. Tool movement at the indicated feedrate ·F· until the probe signal is received.

Withdrawal movement. Rapid tool movement (G00) from the probing point to the safety position located at a ·Ds· distance from the probe.

5 Approach movement to the second probing point.

Tool movement from the first approach point to the second located in front of the first one.

The perpendicular axis moves in rapid (G00) up to the approach coordinate. Then, the tool moves in rapid (G00) over the probe up to the next approach point. The tool moves back down to the probing Z coordinate at feedrate ·Fs·.

For the movement, the cycle takes into consideration the theoretical length of the probe and parameter ·Ds·. If in this last movement, the tool touches the probe, the tool goes back up to the approach coordinate and moves the safety distance ·Ds· in the same direction. The cycle repeats this operation until the tool releases the probe.

6 Probing of the second point.

It is done along the abscissa axis same as the previous one.

7 Approach movement to the third probing point.

Tool movement from the second approach point to the third, located in front of the minimum Y side of the probe. Position of the abscissa axis corresponding to the real center coordinate of the probe.

8 Probing of the third point.

It is done along the ordinate axis same as the previous one.

9 Approach movement to the fourth approach point (top surface).

The tool moves up in raid (G00) up to the approach coordinate, it then moves in rapid (G00) along the Y axis up to the theoretical center of the probe.

The perpendicular axis moves in rapid (G00) up to the approach coordinate. Then, the tool moves in rapid (G00) along the ordinate axis and over the probe up to the theoretical center of the probe.

10 Measurement of the top surface.

The tool makes a first probing movement, a withdrawal movement and a second probing move. After the second probing move, the tool withdraws to the approach point.

First probing movement. Tool movement along the perpendicular axis at feedrate ·Fs· until the probe signal is received.

Withdrawal movement. Return of the tool in rapid (G00) the distance indicated in ·Dr·.

Second probe. Tool movement at the indicated feedrate ·F· until the probe signal is received.

Withdrawal movement. Rapid tool movement (G00) from the probing point to the safety position located at a ·Ds· distance from the probe.

11 Approach movement to the fifth probing point.

Tool movement from the forth approach point to the fifth, located in front of the third one.

The tool moves in rapid (G00) to the next approach point. The tool moves back down to the probing Z coordinate at feedrate ·Fs·.

For the movement, the cycle takes into consideration the theoretical length of the probe and parameter ·Ds·. If in this last movement, the tool touches the probe, the tool goes back up to the approach coordinate and moves the safety distance ·Ds· in the same direction. The cycle repeats this operation until the tool releases the probe.

12 Probing of the fifth point.

It is done along the ordinate axis same as the third probing point.

13 Withdrawal movement.

The perpendicular axis moves in rapid (G00) up to the approach coordinate.

14 Withdrawal movement.

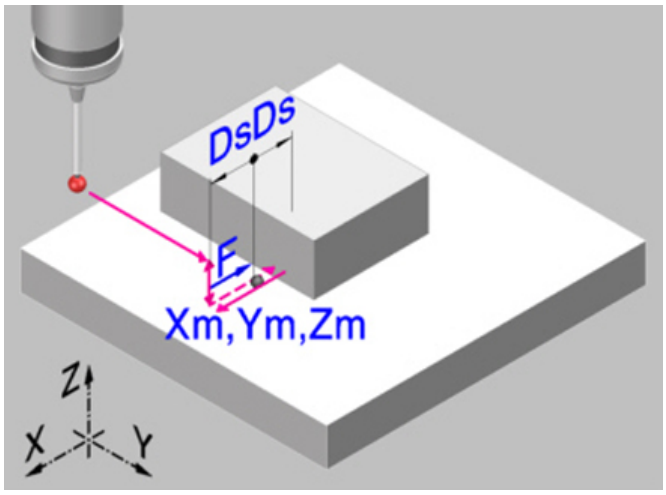
Rapid probe movement (G00) up to the cycle calling point.

This withdrawal movement is made in two stages. It first moves in the work plane and then along the axis perpendicular.

15 If the cycle is going to do a double calibration, the spindle is oriented to 180º and the cycle repeats steps 2 through 14.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2106

4.5 Surfacing measuring.



This cycle measures the part along the selected axis. With this cycle, it is also possible to correct the value of the wear of the tool used to machine the surface. The wear correction only takes place when the measuring error exceeds a programmed value.

The probing movement may be programmed in the three axes of the active channel.

Data returned by the cycle after the measurement.

Once the cycle has ended, the CNC returns the real values obtained in the measurement in the following arithmetic parameters:

P298 Actual (real) surface coordinate.

P299 Detected error. Difference between the actual surface coordinate and the programmed theoretical coordinate.

If wear correction is enabled in the calling instruction, the CNC updates those values in the programmed tool. This correction is applied only if the measuring error is equal to or greater than the programmed tolerance.

Depending on the axis used for measuring, the cycle will correct either the tool length value or the tool radius value.

- If the measurement is made on the axis perpendicular to the work plane, it will correct the length wear.
- If the measurement is made with one of the axis forming the plane, it will correct the tool radius wear.

4.5.1 Programming the cycle.

Probe data.

·Tp· Number of the tool that identifies the probe.

Number of the tool used to define the probe in the tool table. Optional parameter, if not defined or set to 0, use the probe that is in the spindle when executing the cycle.

·Dp· Number of the tool offset that identifies the probe.

Offset associated with the probe, used to execute the cycle.

Probing movement.

·icon· Probing axis.

This parameter indicates the axis to be used to measure the surface.

Probing with the abscissa axis.

Probing with the ordinate axis.

Probing with the axis perpendicular to the plane.

·icon· Cycle ending point (calling point or measured point).

This parameter indicates where the probing cycle ends.

The probe returns to the point from where the cycle was called.

The cycle ends with the probe positioned over the measured point. The longitudinal axis returns to the point where the cycle was called.

·Xm Ym Zm· Theoretical coordinates of the probing point.

Optional parameter; if any of them is not programmed, the cycle assumes the current position of the axis as theoretical coordinate of the probing point. The axes that are not programmed do not execute the positioning movement before probing.

Xm Theoretical coordinate of the probing point along the abscissa axis.

Ym Theoretical coordinate of the probing point along the ordinate axis.

Zm Theoretical coordinate of the probing point along the axis perpendicular to the plane.

The canned cycles may be programmed in any work plane.

·icon· Moving direction for axis that has not been programmed.

When not programming the theoretical coordinate of the probing point on some axis, the cycle shows an icon to define the moving direction of that axis.

The probe moves in the positive direction of the X axis.

The probe moves in the negative direction of the X axis.

The probe moves in the positive direction of the Y axis.

The probe moves in the negative direction of the Y axis.

The probe moves in the positive direction of the Z axis.

The probe moves in the negative direction of the Z axis.

·Ds· Safety distance.

This parameter only admits positive values greater than 0 (zero).

Distance with respect to the point to measure and along the probing axis, to which the probe approaches in G00 before making the probing movement. When calling the cycle, the probe must be located, with respect to the point to be measured, at a greater distance than this value

·F· Probing feedrate.

This parameter sets the probing feedrate. The rest of the movements will be carried out in G00.

Wear measurement.

Tool wear correction is optional. If it is activated, the correction only takes place when the measuring error exceeds the programmed value.

Wear correction active.

No tool wear correction is applied if this box is not selected.

·T· Tool to be corrected.

Optional parameter; by default, undefined. If T=0 (or not programmed), the CNC interprets that it is the tool used for machining.

Tool whose wear is to be corrected, which will be the tool used to machine the surface.

·D· Tool offset to be corrected.

Tool offset whose wear is to be corrected, which will be the tool offset used to machine the surface.

·WT· Tolerance for the measuring error.

Optional parameter, by default 0. This parameter only admits positive values.

If the measuring error (difference between the theoretical and the real values) is within this tolerance, the CNC does not change the tool data. If the measuring error is equal to or greater than this tolerance, the CNC corrects the data of the tool defined in parameters ·T· and ·D·.

Save the result of the measurement.

·icon· Save the result of the measurement in a zero offset G159.

This parameter indicates whether or not the result of the measurement will be saved in a zero offset G159. Regardless of the selected option, the cycle always saves the result of the measurements in the corresponding arithmetic parameters.

Save the result of the measurement in a zero offset G159.

Do not save the result of the measurement in a zero offset G159.

·OffsetG159· Zero offset number

Number of the zero offset into which to save the result of a measurement. If it is set to 0, the cycle does not save any data into the zero offset.

Programming of M functions.

·M before· M functions to be executed before the cycle.

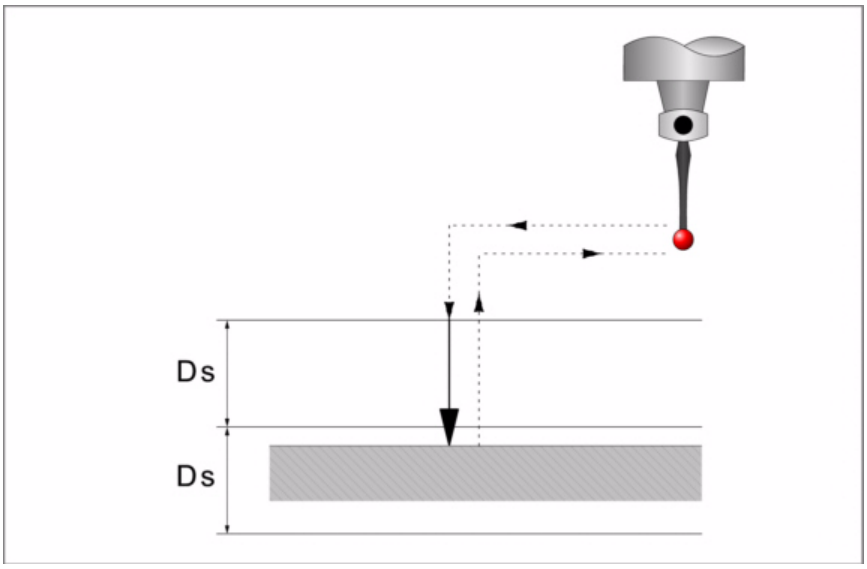
The cycle allows executing up to 4 M functions before the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

·M after· M functions to be executed after the cycle.

The cycle allows executing up to 4 M functions after the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

We recommend using these functions, for example, to manage wireless probes. Wireless probes are not always active, they have to be turned on before using the probing cycles and turned off afterwards. For this type of probes, set an M function to turn the probe on and another one to turn it off. Having the probe turn on/off programmed with M functions inside the cycle avoids executing the cycle without having the probe active or leaving the probe always active after executing the cycle.

4.5.2 Basic operation.



1 The cycle executes the "M-before" functions.

2 Approach movement (if any of parameters ·XYZ· has been defined).

Rapid probe movement (G00) from the cycle calling point to the approach point located at a distance ·Ds· from the probing point and along the probing axis.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.

3 Probing movement.

Probing movement. Probing movement along the selected axis at the indicated feedrate ·F· until the probe signal is received. The maximum probing distance is ·2Ds·. If once this distance has been reached, the CNC has not yet received the probe signal, it will issue the relevant error code and stop the movement of the axes.

Once probing is over, the CNC will assume the actual position of the axes when the probe signal is received as their theoretical position.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the approach point.

4 Withdrawal movement.

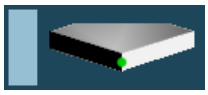
Rapid probe movement (G00) from the probing point to the cycle calling point.

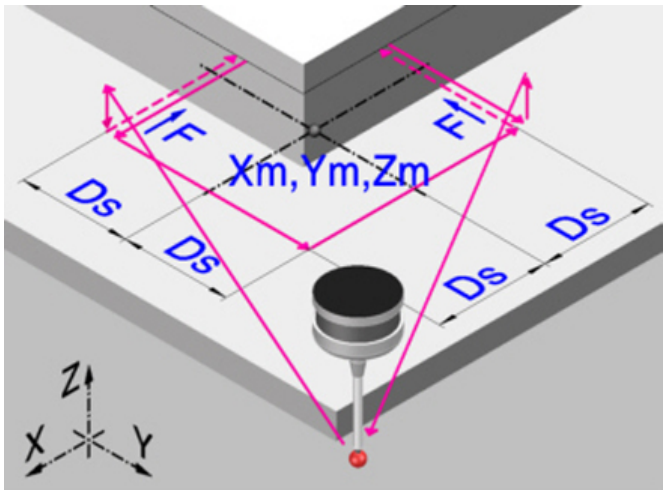
It first moves along the perpendicular axis up to the coordinate of the cycle calling point along that axis. If programmed, it makes a movement in the main plane to the cycle calling point.

5 If programmed, the cycle saves the result of the measurement in the selected G159 zero offset and in the corresponding arithmetic parameters.

6 The cycle executes the "M-after" functions.

4.6 Outside corner measuring.





This cycle calculates the position of an outside corner.

Data returned by the cycle after the measurement.

Once the cycle has ended, the CNC returns the real values obtained in the measurement in the following arithmetic parameters:

P296 Real coordinate of the corner along the abscissa axis.

P297 Real coordinate of the corner along the ordinate axis.

P298 Error detected along the abscissa axis. Difference between the real coordinate of the corner and the theoretical programmed coordinate.

P299 Error detected along the ordinate axis. Difference between the real coordinate of the corner and the theoretical programmed coordinate.

4.6.1 Programming the cycle.

Probe data.

·Tp· Number of the tool that identifies the probe.

Number of the tool used to define the probe in the tool table. Optional parameter, if not defined or set to 0, use the probe that is in the spindle when executing the cycle.

·Dp· Number of the tool offset that identifies the probe.

Offset associated with the probe, used to execute the cycle.

Probing movement.

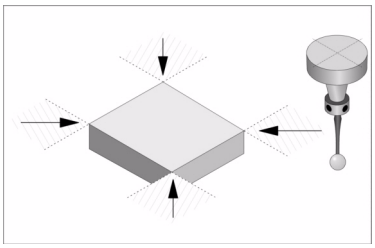
·Xm Ym Zm· Theoretical coordinates of the corner to be measured.

Xm Theoretical coordinate of the corner along the abscissa axis.

Ym Theoretical coordinate of the corner along the ordinate axis.

Zm Theoretical coordinate of the corner along the axis perpendicular to the plane.

The canned cycles may be programmed in any work plane. See *"Canned cycles and the work planes."* on page 93.



Depending on the corner of the part it is required to measure, the probe must be placed in the corresponding shaded area (see figure) before calling the cycle.

·Ds· Safety distance.

This parameter only admits positive values greater than 0 (zero).

Distance with respect to the point to measure, to which the probe approaches in G00 before making the probing movement. The probe must be placed, with respect to the point to be measured, at a distance greater than this value when the cycle is called.

·F· Probing feedrate.

This parameter sets the probing feedrate. The rest of the movements will be carried out in G00.

Save the result of the measurement.

·icon· Save the result of the measurement in a zero offset G159.

This parameter indicates whether or not the result of the measurement will be saved in a zero offset G159. Regardless of the selected option, the cycle always saves the result of the measurements in the corresponding arithmetic parameters.

Save the result of the measurement in a zero offset G159.

Do not save the result of the measurement in a zero offset G159.

·OffsetG159· Zero offset number

Number of the zero offset into which to save the result of a measurement. If it is set to 0, the cycle does not save any data into the zero offset.

Programming of M functions.

·M before· M functions to be executed before the cycle.

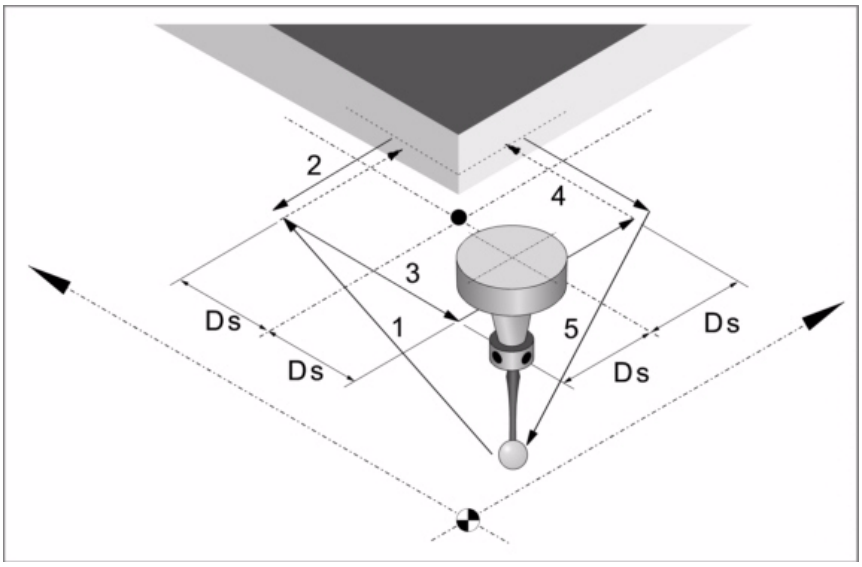
The cycle allows executing up to 4 M functions before the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

·M after· M functions to be executed after the cycle.

The cycle allows executing up to 4 M functions after the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

We recommend using these functions, for example, to manage wireless probes. Wireless probes are not always active, they have to be turned on before using the probing cycles and turned off afterwards. For this type of probes, set an M function to turn the probe on and another one to turn it off. Having the probe turn on/off programmed with M functions inside the cycle avoids executing the cycle without having the probe active or leaving the probe always active after executing the cycle.

4.6.2 Basic operation.



1 Approach movement to the first side to be probed.

Rapid probe movement (G00) from the cycle calling point to the first approach point located at a ·Ds· distance from the side to be probed.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.

2 Probing on the first side.

The probing on each side consists of a probing move and a withdrawal move.

Probing movement. Probe movement along the abscissa axis at feedrate ·F· until the probe signal is received. The maximum probing distance is ·2Ds·. If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the first approach point.

3 Approach movement to the second side to be probed.

Rapid probe move (G00) from the first approach point to the second.

This approach movement is made in two stages. It first moves in the work plane and then along the ordinate axis and then along the abscissa axis.

4 Probing on the second side.

Probing movement. Probe movement along the ordinate axis at feedrate ·F· until the probe signal is received. The maximum probing distance is ·2Ds·. If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.

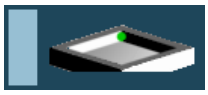
Withdrawal movement. Rapid probe movement (G00) from the probing point to the second approach point.

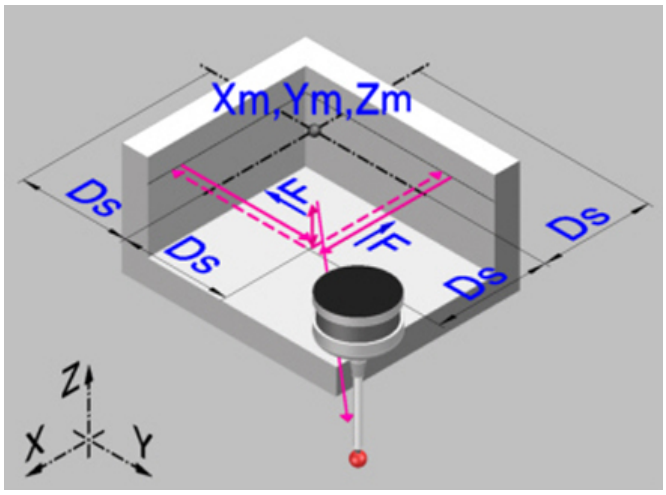
5 Withdrawal movement.

Rapid probe movement (G00) from the second approach point to the cycle calling point.

This withdrawal movement is made in two stages. It first moves in the work plane and then along the perpendicular axis and then in the work plane.

4.7 Inside corner measuring.





This cycle calculates the position of an inside corner.

Data returned by the cycle after the measurement.

Once the cycle has ended, the CNC returns the real values obtained in the measurement in the following arithmetic parameters:

P296 Real coordinate of the corner along the abscissa axis.

P297 Real coordinate of the corner along the ordinate axis.

P298 Error detected along the abscissa axis. Difference between the real coordinate of the corner and the theoretical programmed coordinate.

P299 Error detected along the ordinate axis. Difference between the real coordinate of the corner and the theoretical programmed coordinate.

4.7.1 Programming the cycle.

Probe data.

·Tp· Number of the tool that identifies the probe.

Number of the tool used to define the probe in the tool table. Optional parameter, if not defined or set to 0, use the probe that is in the spindle when executing the cycle.

·Dp· Number of the tool offset that identifies the probe.

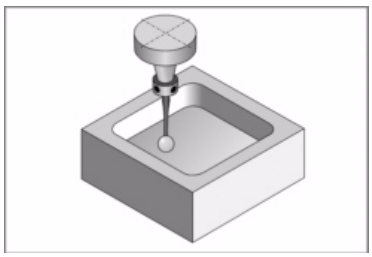
Offset associated with the probe, used to execute the cycle.

Probing movement.

·Xm Ym Zm· Theoretical coordinates of the corner to be measured.

Xm	Theoretical coordinate of the corner along the abscissa axis.
Ym	Theoretical coordinate of the corner along the ordinate axis.
Zm	Theoretical coordinate of the corner along the axis perpendicular to the plane.

The canned cycles may be programmed in any work plane. See *"Canned cycles and the work planes."* on page 93.



The probe must be placed within the pocket before calling the cycle.

·Ds· Safety distance.

This parameter only admits positive values greater than 0 (zero).

Distance with respect to the point to measure, to which the probe approaches in G00 before making the probing movement. The probe must be placed, with respect to the point to be measured, at a distance greater than this value when the cycle is called.

·F· Probing feedrate.

This parameter sets the probing feedrate. The rest of the movements will be carried out in G00.

Save the result of the measurement.

·icon· Save the result of the measurement in a zero offset G159.

This parameter indicates whether or not the result of the measurement will be saved in a zero offset G159. Regardless of the selected option, the cycle always saves the result of the measurements in the corresponding arithmetic parameters.

Save the result of the measurement in a zero offset G159.

Do not save the result of the measurement in a zero offset G159.

·OffsetG159· Zero offset number

Number of the zero offset into which to save the result of a measurement. If it is set to 0, the cycle does not save any data into the zero offset.

Programming of M functions.

·M before· M functions to be executed before the cycle.

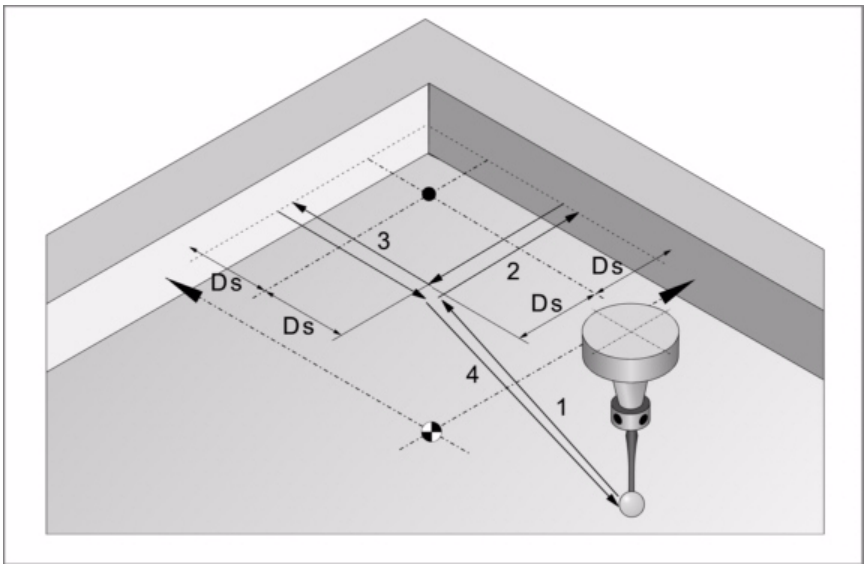
The cycle allows executing up to 4 M functions before the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

·M after· M functions to be executed after the cycle.

The cycle allows executing up to 4 M functions after the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

We recommend using these functions, for example, to manage wireless probes. Wireless probes are not always active, they have to be turned on before using the probing cycles and turned off afterwards. For this type of probes, set an M function to turn the probe on and another one to turn it off. Having the probe turn on/off programmed with M functions inside the cycle avoids executing the cycle without having the probe active or leaving the probe always active after executing the cycle.

4.7.2 Basic operation.



1 Approach movement.

Rapid probe movement (G00) from the cycle calling point to the first approach point located at a ·Ds· distance from the two sides to be probed.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.

2 Probing on the first side.

The probing on each side consists of a probing move and a withdrawal move.

Probing movement. Probe movement along the abscissa axis at feedrate ·F· until the probe signal is received. The maximum probing distance is ·2Ds·. If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the approach point.

3 Probing on the second side.

Probing movement. Probe movement along the ordinate axis at feedrate ·F· until the probe signal is received. The maximum probing distance is ·2Ds·. If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.

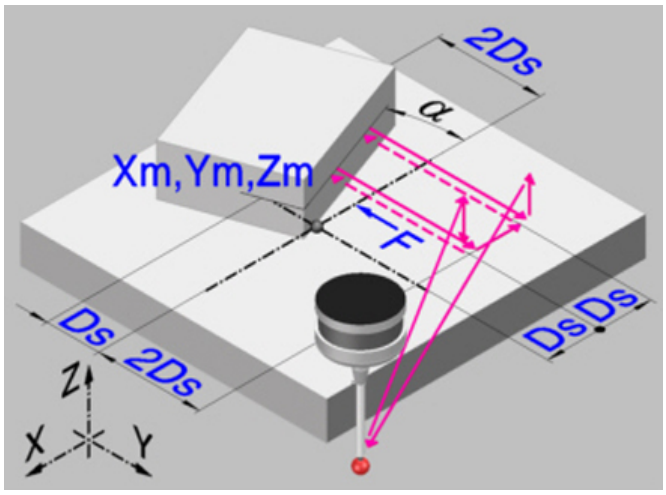
Withdrawal movement. Rapid probe movement (G00) from the probing point to the approach point.

4 Withdrawal movement.

Rapid probe movement (G00) from the approach point to the cycle calling point.

This withdrawal movement is made in two stages. It first moves in the work plane and then along the perpendicular axis and then in the work plane.

4.8 Measuring the angle with the abscissa axis.



This cycle calculates the angle of the part with respect to the abscissa axis. This cycle may be used to measure angles between ±45º.

- If the angle to be measured is equal to or greater than 45º, the CNC will display the relevant error.
- If the angle to be measured is equal to or smaller than –45º the probe will collide with the part.

Data returned by the cycle after the measurement.

Once the cycle has ended, the CNC returns the real values obtained in the measurement in the following arithmetic parameters:

P295 Inclination angle between the part and the abscissa axis.

4.8.1 Programming the cycle.

Probe data.

·Tp· Number of the tool that identifies the probe.

Number of the tool used to define the probe in the tool table. Optional parameter, if not defined or set to 0, use the probe that is in the spindle when executing the cycle.

·Dp· Number of the tool offset that identifies the probe.

Offset associated with the probe, used to execute the cycle.

Probing movement.

·Xm Ym Zm· Theoretical coordinates of the corner to be measured.

Xm Theoretical coordinate of the corner along the abscissa axis.

Ym Theoretical coordinate of the corner along the ordinate axis.

Zm Theoretical coordinate of the corner along the axis perpendicular to the plane.

The canned cycles may be programmed in any work plane. See *"Canned cycles and the work planes."* on page 93.

·Ds· Safety distance.

This parameter only admits positive values greater than 0 (zero).

Distance with respect to the point to measure, to which the probe approaches in G00 before making the probing movement. When calling the cycle, the probe must be located at a distance twice this value with respect to the point to be measured,

·F· Probing feedrate.

This parameter sets the probing feedrate. The rest of the movements will be carried out in G00.

Programming of M functions.

·M before· M functions to be executed before the cycle.

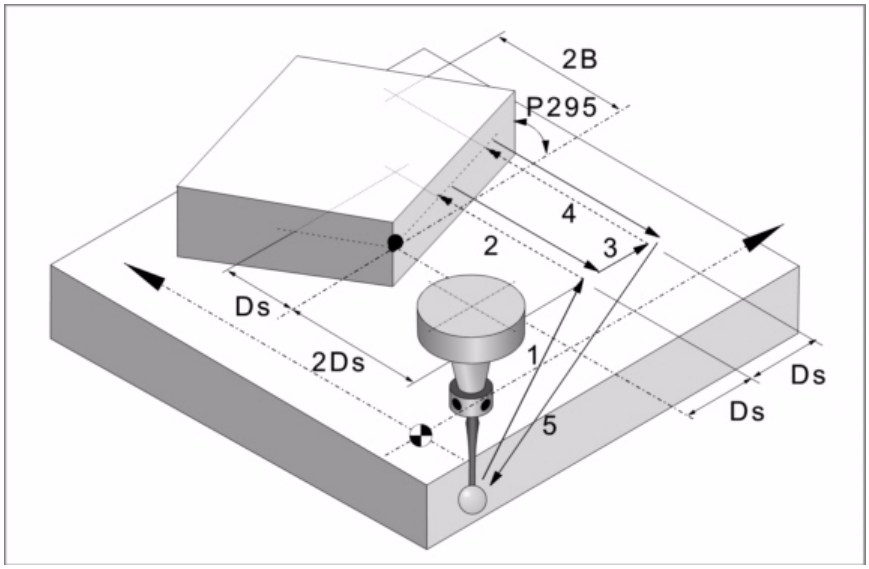
The cycle allows executing up to 4 M functions before the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

·M after· M functions to be executed after the cycle.

The cycle allows executing up to 4 M functions after the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

We recommend using these functions, for example, to manage wireless probes. Wireless probes are not always active, they have to be turned on before using the probing cycles and turned off afterwards. For this type of probes, set an M function to turn the probe on and another one to turn it off. Having the probe turn on/off programmed with M functions inside the cycle avoids executing the cycle without having the probe active or leaving the probe always active after executing the cycle.

4.8.2 Basic operation.



1 First approach movement.

Rapid probe movement (G00) from the cycle calling point to the first approach point located at a distance ·2Ds· from the side to be probed.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.

2 First probing move.

Probing movement. Probe movement along the ordinate axis at feedrate ·F· until the probe signal is received. The maximum probing distance is ·3Ds·. If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the first approach point.

3 Second approach movement.

Rapid probe move (G00) from the first approach point to the second. The second approach point is located at a distance ·Ds· from the first one.

4 Second probe.

Probing movement. Probe movement along the ordinate axis at feedrate ·F· until the probe signal is received. The maximum probing distance is ·4Ds·. If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.

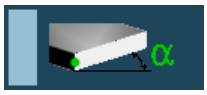
Withdrawal movement. Rapid probe movement (G00) from the probing point to the second approach point.

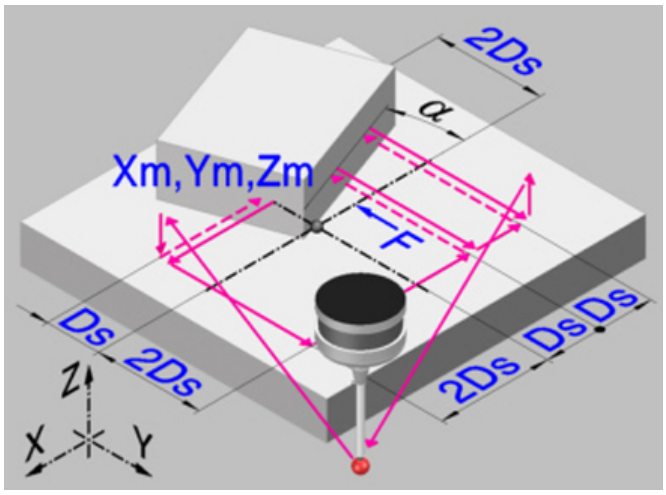
5 Withdrawal movement.

Rapid probe movement (G00) from the approach point to the cycle calling point.

This withdrawal movement is made in two stages. It first moves in the work plane and then along the perpendicular axis and then in the work plane.

4.9 Outside corner and angle measuring.





This cycle calculates the position of an outside corner and the angle of the part with respect to the abscissa axis. This cycle may be used to measure angles between ±45º.

- If the angle to be measured is equal to or greater than 45º, the CNC will display the relevant error.
- If the angle to be measured is equal to or smaller than –45º the probe will collide with the part.

Data returned by the cycle after the measurement.

Once the cycle has ended, the CNC returns the real values obtained in the measurement in the following arithmetic parameters:

P295	Inclination angle between the part and the abscissa axis.
P296	Real coordinate of the corner along the abscissa axis.
P297	Real coordinate of the corner along the ordinate axis.
P298	Error detected along the abscissa axis. Difference between the real coordinate of the corner and the theoretical programmed coordinate.
P299	Error detected along the ordinate axis. Difference between the real coordinate of the corner and the theoretical programmed coordinate.

4.9.1 Programming the cycle.

Probe data.

·Tp· Number of the tool that identifies the probe.

Number of the tool used to define the probe in the tool table. Optional parameter, if not defined or set to 0, use the probe that is in the spindle when executing the cycle.

·Dp· Number of the tool offset that identifies the probe.

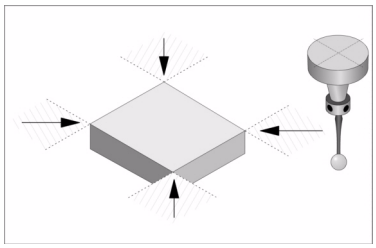
Offset associated with the probe, used to execute the cycle.

Probing movement.

·Xm Ym Zm· Theoretical coordinates of the corner to be measured.

Xm	Theoretical coordinate of the corner along the abscissa axis.
Ym	Theoretical coordinate of the corner along the ordinate axis.
Zm	Theoretical coordinate of the corner along the axis perpendicular to the plane.

The canned cycles may be programmed in any work plane. See *"Canned cycles and the work planes."* on page 93.



Depending on the corner of the part it is required to measure, the probe must be placed in the corresponding shaded area (see figure) before calling the cycle.

·Ds· Safety distance.

This parameter only admits positive values greater than 0 (zero).

Distance with respect to the point to measure, to which the probe approaches in G00 before making the probing movement. When calling the cycle, the probe must be located at a distance twice this value with respect to the point to be measured,

·F· Probing feedrate.

This parameter sets the probing feedrate. The rest of the movements will be carried out in G00.

Save the result of the measurement.

·icon· Save the result of the measurement in a zero offset G159.

This parameter indicates whether or not the result of the measurement will be saved in a zero offset G159. Regardless of the selected option, the cycle always saves the result of the measurements in the corresponding arithmetic parameters.

Save the result of the measurement in a zero offset G159.

Do not save the result of the measurement in a zero offset G159.

·OffsetG159· Zero offset number

Number of the zero offset into which to save the result of a measurement. If it is set to 0, the cycle does not save any data into the zero offset.

Programming of M functions.

·M before· M functions to be executed before the cycle.

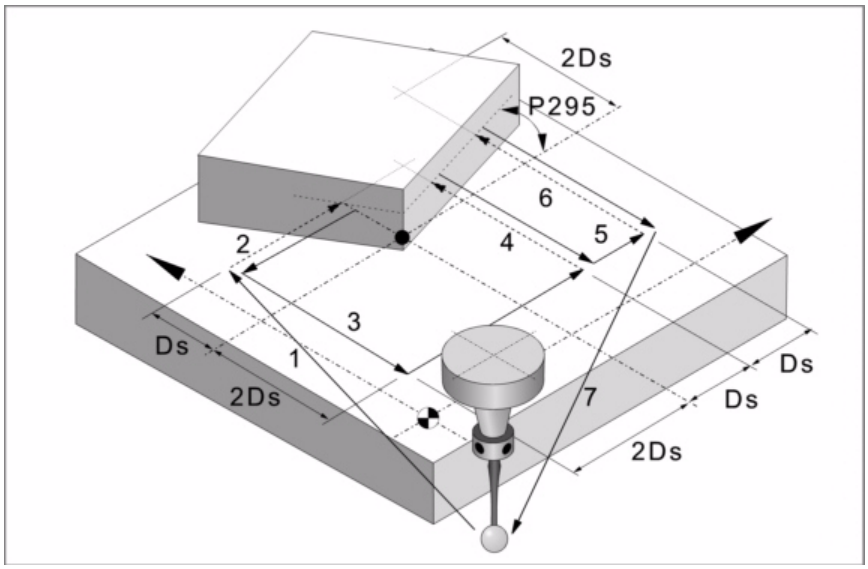
The cycle allows executing up to 4 M functions before the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

·M after· M functions to be executed after the cycle.

The cycle allows executing up to 4 M functions after the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

We recommend using these functions, for example, to manage wireless probes. Wireless probes are not always active, they have to be turned on before using the probing cycles and turned off afterwards. For this type of probes, set an M function to turn the probe on and another one to turn it off. Having the probe turn on/off programmed with M functions inside the cycle avoids executing the cycle without having the probe active or leaving the probe always active after executing the cycle.

4.9.2 Basic operation.



1 First approach movement.

Rapid probe movement (G00) from the cycle calling point to the first approach point located at a ·2Ds· distance from the side to be probed.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.

2 First probing move.

Probing movement. Probe movement along the ordinate axis at feedrate ·F· until the probe signal is received. The maximum probing distance is ·3Ds·. If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the first approach point.

3 Second approach movement.

Rapid probe movement (G00) from the first approach point to the second, located at a ·2Ds· distance from the second side to be probed.

This approach movement is made in two stages. It first moves in the work plane and then along the ordinate axis and then along the abscissa axis.

4 Second probe.

Probing movement. Probe movement along the ordinate axis at feedrate ·F· until the probe signal is received. The maximum probing distance is ·3Ds·. If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the second approach point.

5 Third approach movement.

Rapid probe move (G00) from the second approach point to the third. It is located at a ·Ds· distance from the previous one.

6 Third probing move.

Probing movement. Probe movement along the ordinate axis at feedrate ·F· until the probe signal is received. The maximum probing distance is ·4Ds·. If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the third approach point.

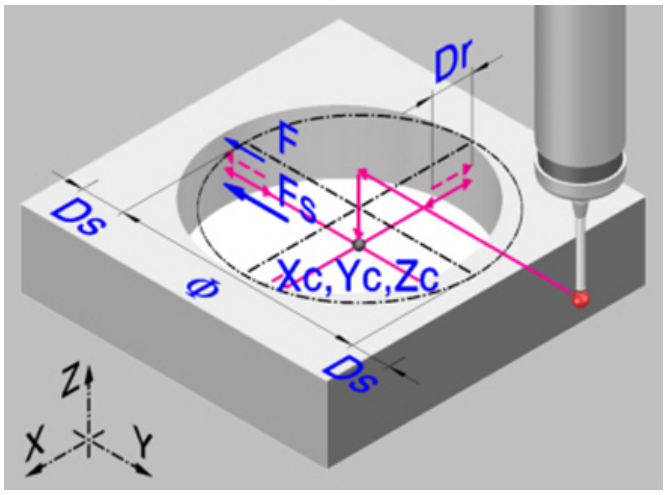
7 Withdrawal movement.

Rapid probe movement (G00) from the approach point to the cycle calling point.

This withdrawal movement is made in two stages. It first moves in the work plane and then along the perpendicular axis and then in the work plane.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2106

4.10 Hole measuring.



This cycle calculates the diameter of a hole and the real position of the center.

Data returned by the cycle after the measurement.

Once the cycle has ended, the CNC returns the real values obtained in the measurement in the following arithmetic parameters:

P294	Hole diameter.
P295	Error detected in the hole diameter. Difference between the real diameter and the programmed theoretical diameter.
P296	Real coordinate of the center along the abscissa axis.
P297	Real center coordinate along the ordinate axis.
P298	Error detected along the abscissa axis. Difference between the real coordinate of the center and the programmed theoretical coordinate.
P299	Error detected along the ordinate axis. Difference between the real coordinate of the center and the programmed theoretical coordinate.

4.10.1 Programming the cycle.

Probe data.

·Tp· Number of the tool that identifies the probe.

Number of the tool used to define the probe in the tool table. Optional parameter, if not defined or set to 0, use the probe that is in the spindle when executing the cycle.

·Dp· Number of the tool offset that identifies the probe.

Offset associated with the probe, used to execute the cycle.

Probing movement.

·icon· Cycle ending point (calling point or measured point).

This parameter indicates where the probing cycle ends.

The probe returns to the point from where the cycle was called.

The cycle ends with the probe positioned over the measured point. The longitudinal axis returns to the point where the cycle was called.

·Xm Ym Zm· Theoretical coordinates of the hole center.

Xm	Theoretical hole center coordinate along the abscissa axis.
Ym	Theoretical hole center coordinate along the ordinate axis.
Zm	Theoretical hole center coordinate along the axis perpendicular to the plane.

The canned cycles may be programmed in any work plane. See *"Canned cycles and the work planes."* on page 93.

·Ø· Theoretical diameter of the hole.

This parameter only admits positive values greater than 0 (zero).

This parameter sets the theoretical diameter of the hole. This cycle may be used to measure holes with diameters no larger than "Ø+Ds".

·Ds· Safety distance.

This parameter only admits positive values greater than 0 (zero).

·Dr· Withdrawal distance after initial probing.

This parameter only admits positive values greater than 0 (zero).

This parameter sets the distance the probe withdrawals after the first probing movement. Once it withdraws this distance, the CNC makes a second probing movement.

·Fs· Feedrate for the first probing movement.

This parameter sets the feedrate for the first probing movement. Then, the CNC will repeat the probing movement at feedrate ·F·.

·F· Probing feedrate.

This parameter sets the feedrate for the second probing movement. The rest of the movements are carried out in G00 except the first probing movement which is carried out at feedrate ·Fs·.

Save the result of the measurement.

·icon· Save the result of the measurement in a zero offset G159.

This parameter indicates whether or not the result of the measurement will be saved in a zero offset G159. Regardless of the selected option, the cycle always saves the result of the measurements in the corresponding arithmetic parameters.

Save the result of the measurement in a zero offset G159.

Do not save the result of the measurement in a zero offset G159.

·OffsetG159· Zero offset number

Number of the zero offset into which to save the result of a measurement. If it is set to 0, the cycle does not save any data into the zero offset.

Programming of M functions.

·M before· M functions to be executed before the cycle.

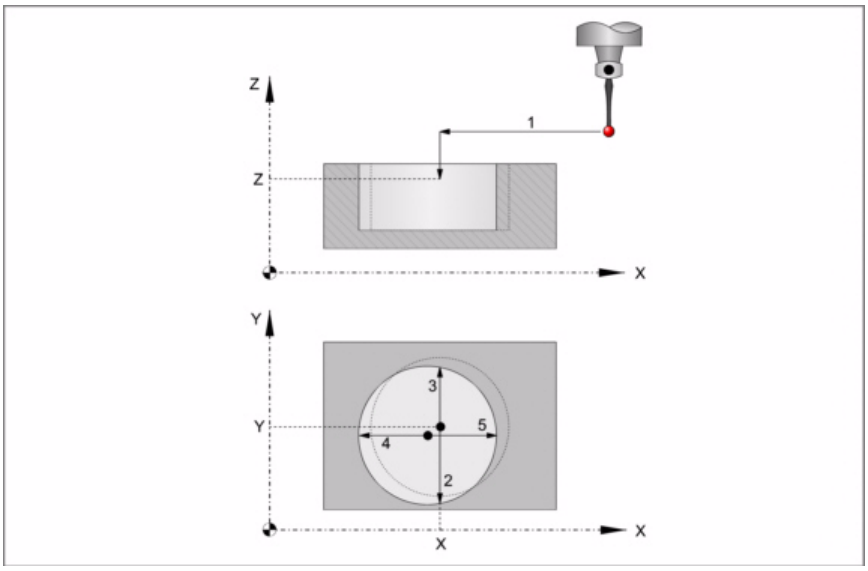
The cycle allows executing up to 4 M functions before the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

·M after· M functions to be executed after the cycle.

The cycle allows executing up to 4 M functions after the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

We recommend using these functions, for example, to manage wireless probes. Wireless probes are not always active, they have to be turned on before using the probing cycles and turned off afterwards. For this type of probes, set an M function to turn the probe on and another one to turn it off. Having the probe turn on/off programmed with M functions inside the cycle avoids executing the cycle without having the probe active or leaving the probe always active after executing the cycle.

4.10.2 Basic operation.



1 Approach movement.

Probe's rapid movement (G00) from the cycle calling point to the center of the hole.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.

2 Probing of the first point

The probe touches in four different points. In each point, it makes a first probing movement, a withdrawal movement and a second probing move. After the second probing move, the probe returns to the center of the hole.

First probing movement. Probe movement along the ordinate axis at feedrate ·Fs· until the probe signal is received. The maximum probing distance is "Ds+Ø/2". If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.

Withdrawal movement. Probe's rapid movement (G00) the distance indicated by ·Dr·.

Second probe. Probe movement along the ordinate axis at feedrate ·F· until the probe signal is received.

Withdrawal movement to the center of the hole. Rapid probe movement (G00) from the probing point to the center of the hole.

3 Probing of the second point.

It is done along the ordinate axis same as the previous one.

4 Probing of the third point.

It is done along the abscissa axis same as the previous one.

5 Probing of the fourth point.

It is done along the abscissa axis same as the previous one.

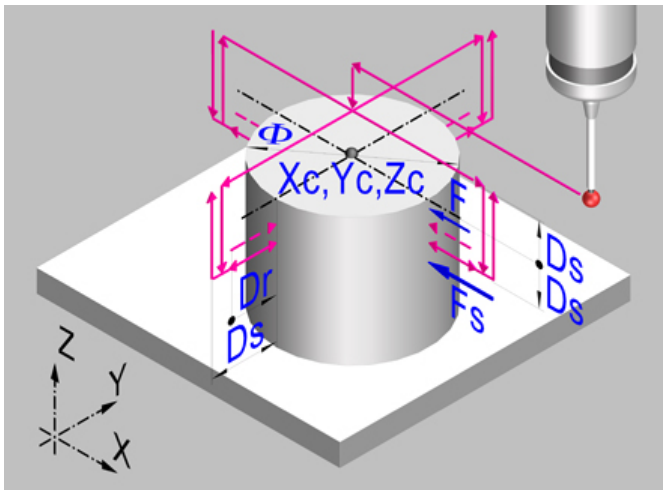
The probe moves to the real hole center (the calculated one).

6 Withdrawal movement (if it has been defined).

If it has been defined, the probe moves in rapid (G00) from the center of the hole to the cycle calling point.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane and then in the work plane.

4.11 Circular boss measuring.



This cycle calculates the diameter of a boss and the real position of the center.

Data returned by the cycle after the measurement.

Once the cycle has ended, the CNC returns the real values obtained in the measurement in the following arithmetic parameters:

P294	Boss diameter.
P295	Error detected in the boss diameter. Difference between the real diameter and the programmed theoretical diameter.
P296	Real coordinate of the center along the abscissa axis.
P297	Real center coordinate along the ordinate axis.
P298	Error detected along the abscissa axis. Difference between the real coordinate of the center and the programmed theoretical coordinate.
P299	Error detected along the ordinate axis. Difference between the real coordinate of the center and the programmed theoretical coordinate.

4.11.1 Programming the cycle.

Probe data.

·Tp· Number of the tool that identifies the probe.

Number of the tool used to define the probe in the tool table. Optional parameter, if not defined or set to 0, use the probe that is in the spindle when executing the cycle.

·Dp· Number of the tool offset that identifies the probe.

Offset associated with the probe, used to execute the cycle.

Probing movement.

·icon· Cycle ending point (calling point or measured point).

This parameter indicates where the probing cycle ends.

The probe returns to the point from where the cycle was called.

The cycle ends with the probe positioned over the measured point. The longitudinal axis returns to the point where the cycle was called.

·Xm Ym Zm· Theoretical coordinates of the boss center.

Xm	Theoretical boss center coordinate along the abscissa axis.
Ym	Theoretical boss center coordinate along the ordinate axis.
Zm	Theoretical boss center coordinate along the axis perpendicular to the plane.

The canned cycles may be programmed in any work plane. See *"Canned cycles and the work planes."* on page 93.

·Ø· Theoretical diameter of the boss.

This parameter only admits positive values greater than 0 (zero).

This parameter sets the theoretical diameter of the boss. This cycle may be used to measure bosses with diameters no larger than "Ø+Ds".

·Ds· Safety distance.

This parameter only admits positive values greater than 0 (zero).

·Dr· Withdrawal distance after initial probing.

This parameter only admits positive values greater than 0 (zero).

This parameter sets the distance the probe withdrawals after the first probing movement. Once it withdraws this distance, the CNC makes a second probing movement.

·Fs· Feedrate for the first probing movement.

This parameter sets the feedrate for the first probing movement. Then, the CNC will repeat the probing movement at feedrate ·F·.

·F· Probing feedrate.

This parameter sets the feedrate for the second probing movement. The rest of the movements are carried out in G00 except the first probing movement which is carried out at feedrate ·Fs·.

Save the result of the measurement.

·icon· Save the result of the measurement in a zero offset G159.

This parameter indicates whether or not the result of the measurement will be saved in a zero offset G159. Regardless of the selected option, the cycle always saves the result of the measurements in the corresponding arithmetic parameters.

Save the result of the measurement in a zero offset G159.

Do not save the result of the measurement in a zero offset G159.

·OffsetG159· Zero offset number

Number of the zero offset into which to save the result of a measurement. If it is set to 0, the cycle does not save any data into the zero offset.

Programming of M functions.

·M before· M functions to be executed before the cycle.

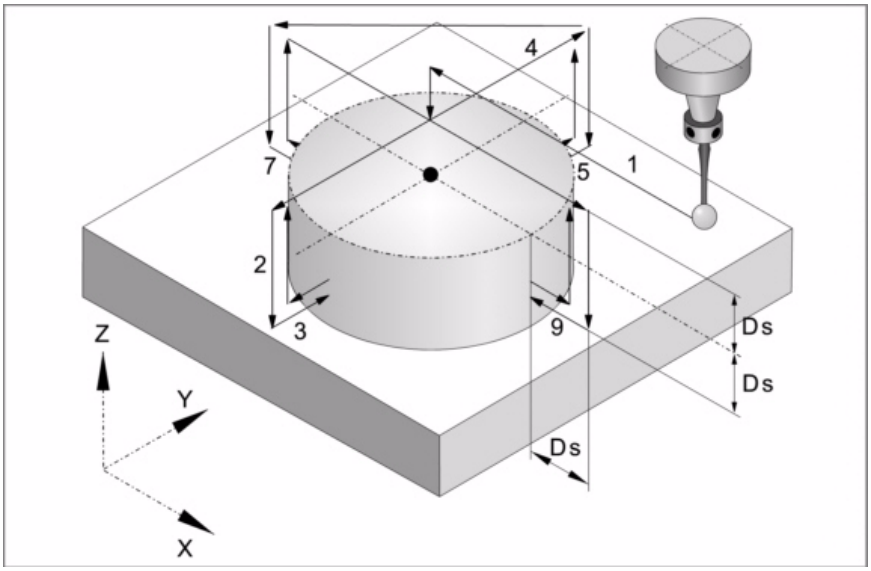
The cycle allows executing up to 4 M functions before the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

·M after· M functions to be executed after the cycle.

The cycle allows executing up to 4 M functions after the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

We recommend using these functions, for example, to manage wireless probes. Wireless probes are not always active, they have to be turned on before using the probing cycles and turned off afterwards. For this type of probes, set an M function to turn the probe on and another one to turn it off. Having the probe turn on/off programmed with M functions inside the cycle avoids executing the cycle without having the probe active or leaving the probe always active after executing the cycle.

4.11.2 Basic operation.



1 Positioning over the boss center.

Probe's rapid movement (G00) from the cycle calling point to the center of the boss. The probe positions over the boss at a distance ·Ds· from the surface.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.

2 First approach movement.

Rapid probe movement (G00) from the center of the boss to the first approach point.

This approach movement is made in two stages. It first moves in the work plane and then along the ordinate axis and then along the perpendicular axis a distance ·2Ds·.

3 Probing of the first point

The probe touches in four different points. In each point, it makes a first probing movement, a withdrawal movement and a second probing move. After the second probing move, the probe withdraws to the approach point.

First probing movement. Probe movement along the ordinate axis at feedrate ·Fs· until the probe signal is received. The maximum probing distance is "Ds+Ø/2". If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.

Withdrawal movement. Probe's rapid movement (G00) the distance indicated by ·Dr·.

Second probe. Probe movement along the ordinate axis at feedrate ·F· until the probe signal is received.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the approach point.

4 Second approach movement.

Rapid probe move (G00) from the first approach point to the second.

The perpendicular axis moves at a distance ·Ds· above the boss, the probe moves to the next approach point above the boss and it goes back down to a distance ·Ds· from the surface.

5 Probing of the second point.

It is done along the ordinate axis same as the previous one.

6 Third approach movement.

It is done like the previous one.

7 Probing of the third point.

It is done along the abscissa axis same as the previous one.

8 Fourth approach movement.

It is done like the previous one.

9 Probing of the fourth point.

It is done along the abscissa axis same as the previous one.

The probe moves in rapid (G00) at a distance ·Ds· over the boss up to the calculated real boss center.

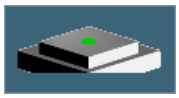
10 Withdrawal movement (if it has been defined).

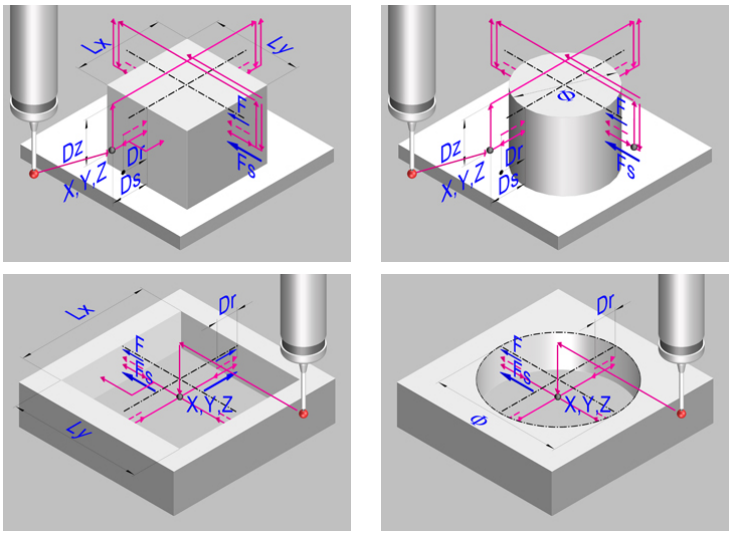
If it has been defined, the probe moves in rapid (G00) from the center of the boss to the cycle calling point.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane and then in the work plane.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2106

4.12 Rectangular or circular part/pocket centering.





This cycle may be used to calculate the center of a rectangular or circular part or pocket of known dimensions as well as, in rectangular parts, the inclination of the part with respect to the abscissa axis.

Requirements prior to the calibration.

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.

To execute this cycle, the probe must be properly calibrated.

Data returned by the cycle after the measurement.

Once the cycle has ended, the CNC returns the real values obtained in the measurement in the following arithmetic parameters:

P296 Inclination angle between the part and the abscissa axis (rectangular part).

P297 Part surface coordinate.

P298 Part center coordinate along the abscissa axis.

P299 Part center coordinate along the ordinate axis.

Optionally, in this cycle, it is possible to preset the coordinates to select a new part zero and, in rectangular parts, rotate the coordinate axes to align the axes with the part.

4.12.1 Programming the cycle.

Probe data.

·Tp· Number of the tool that identifies the probe.

Number of the tool used to define the probe in the tool table. Optional parameter, if not defined or set to 0, use the probe that is in the spindle when executing the cycle.

·Dp· Number of the tool offset that identifies the probe.

Offset associated with the probe, used to execute the cycle.

Probing movement.

·X Y Z· Probe position to execute the cycle.

X Probe position along the abscissa axis

Y Probe position along the ordinate axis.

Z Probe position along the axis perpendicular to the plane.

The canned cycles may be programmed in any work plane.

·icon· Geometry of the part to be centered.

This parameter indicates the type of part/pocket to be centered.

Rectangular part centering.

Circular part centering.

Rectangular pocket centering.

Circular pocket centering.

·icon· Axis and direction of the first probing movement.

This parameter sets the axis on which the fist probing movement will take place.

The probe moves in the positive direction of the X axis.

The probe moves in the negative direction of the X axis.

The probe moves in the positive direction of the Y axis.

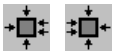
The probe moves in the negative direction of the Y axis.

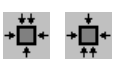
·icon· Center the part/pocket on one or two axes with one or two probings on the first side.

This parameter indicates on how many axes the part is centered and the number of probing movements to carry out on the first side.

Rectangular part. Center the part on one axis.

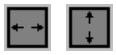
Rectangular part. Center the part on both axes, with one probing on the first side.


Rectangular part. Center the part on both axes, with two probings on the first side.



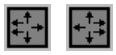
Circular part. Center the part on one axis.

Circular part. Center the part on both axes, with one probing on the first side.

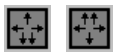


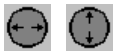
Rectangular pocket. Center the pocket on one axis.

Rectangular pocket. Center the pocket on both axes, with one probing on the first side.



Rectangular pocket. Center the pocket on both axes, with two probings on the first side.





Circular pocket. Center the pocket on one axis.

Circular pocket. Center the pocket on both axes, with one probing on the first side.

·icon· Surface coordinate measuring.

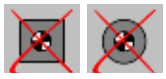
This parameter indicates whether the cycle must also measure the position of the top surface of the part. This icon is only available when the part is centered on two axes.

The cycle does not measure the coordinate of the boss surface or of the bottom of the pocket.

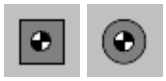
The cycle measures the coordinate of the boss surface or of the bottom of the pocket.

·icon· Preset the part zero after ending the cycle.

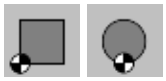
This parameter indicates whether the part zero is to be preset or not, if so, the point taken as reference. This point may be preset with any value using parameters ·Px Py Pz·.



Do not preset the part zero.



Preset the part zero at the center of the part.



Preset the part zero at one of the corners (the cycle shows an icon for each corner).

·icon· Pattern rotation.

For rectangular parts, this parameter indicates whether a coordinate rotation is to be applied or not with the measured angle. This icon is only available when the part is centered on two axes, with two probings on the first side.

Do not rotate the coordinates (pattern).

Rotate the coordinates (pattern).

·Lx Ly Ø· Part dimensions.

These parameters establish the dimensions of the rectangular or circular part.

Lx	Rectangular part. Approximate part length along the abscissa axis.
Ly	Rectangular part. Approximate part length along the ordinate axis.
Ø	Circular part. Part diameter.

·Ds· Safety distance.

Optional parameter; if not defined, it assumes the distance between the part and the probe position when calling the cycle.

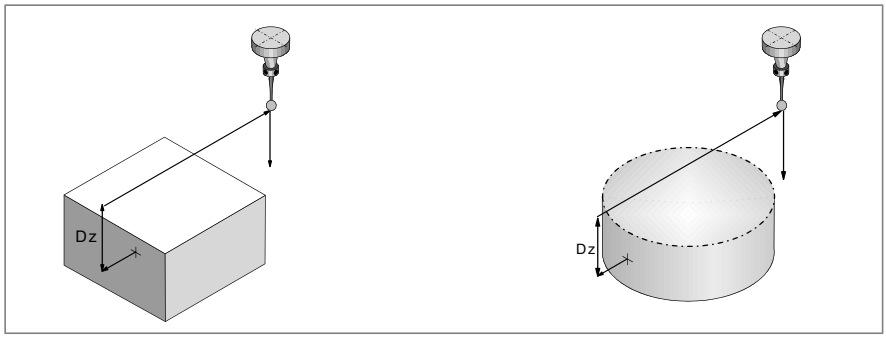
Distance with respect to the point to measure, to which the probe approaches before making the probing movement.

·Dr· Withdrawal distance after initial probing.

This parameter sets the distance the probe withdrawals after the first probing movement. Once it withdraws this distance, the CNC makes a second probing movement.

·Dz· Z safety distance from the probing coordinate (position).

Distance for the probe to go up from the probing coordinate (position) for its movements over the part. This parameter must have enough value to prevent the probe from colliding when moving over the part.



·Fs· Feedrate for the first probing movement.

Optional parameter; by default the lowest value of machine parameter PROBEFEED of the axes that make up the trihedron.

This parameter sets the feedrate for the first probing movement. Then, the CNC will repeat the probing movement at feedrate ·F·.

·F· Probing feedrate.

Optional parameter; by default 10% of the lowest value of machine parameter PROBEFEED of the axes that make up the trihedron.

This parameter sets the feedrate for the second probing movement.

·icon· Feedrate for approach movements.

This parameter sets the type of feedrate for the movements to the approach points.

The movements are carried out in rapid.

The movements are carried out at work feedrate.

Coordinate preset value.

·Px Py Pz· Coordinate preset value on each axis.

Px Preset value on the abscissa axis.

Py Preset value on the ordinate axis.

Pz Preset value on the axis perpendicular to the plane.

These parameters are only valid when presetting the part zero and any this point may be assigned any value.

Save the result of the measurement.

·icon· Save the result of the measurement in a zero offset G159.

This parameter indicates whether or not the result of the measurement will be saved in a zero offset G159. Regardless of the selected option, the cycle always saves the result of the measurements in the corresponding arithmetic parameters.

Save the result of the measurement in a zero offset G159.

Do not save the result of the measurement in a zero offset G159.

·OffsetG159· Zero offset number

Number of the zero offset into which to save the result of a measurement. If it is set to 0, the cycle does not save any data into the zero offset.

Programming of M functions.

·M before· M functions to be executed before the cycle.

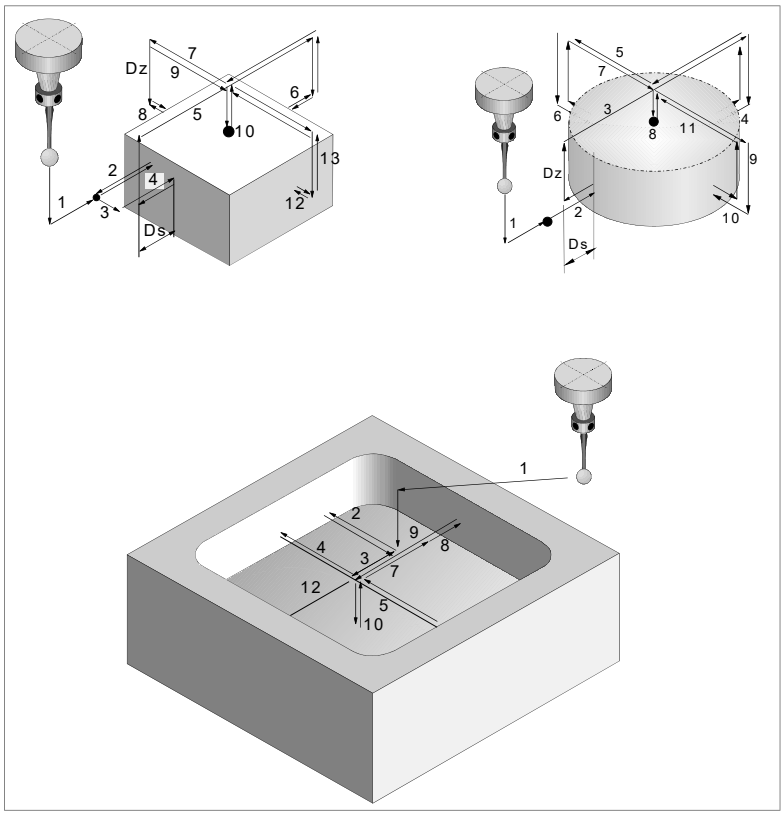
The cycle allows executing up to 4 M functions before the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

·M after· M functions to be executed after the cycle.

The cycle allows executing up to 4 M functions after the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

We recommend using these functions, for example, to manage wireless probes. Wireless probes are not always active, they have to be turned on before using the probing cycles and turned off afterwards. For this type of probes, set an M function to turn the probe on and another one to turn it off. Having the probe turn on/off programmed with M functions inside the cycle avoids executing the cycle without having the probe active or leaving the probe always active after executing the cycle.

4.12.2 Basic operation.



1 The cycle executes the "M-before" functions.

2 Approach movement (if any of parameters ·XYZ· has been defined).

Movement of the probe at the selected feedrate from the cycle calling point to the first approach point defined by parameters ·XYZ·.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.

3 Probing of the first point

The cycle makes a first probing movement, a withdrawal movement and a second probing move. After the second probing move, the probe withdraws to the approach point.

First probing movement. Probe movement along the selected axis at feedrate ·Fs· until the probe signal is received.

Withdrawal movement. Return of the probe in rapid (G00) the distance indicated in ·Dr·.

Second probe. Probing movement at feedrate ·F· until the probe signal is received.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the starting point.

4 If the part is rectangular and probing on two points has been programmed, repeat the probing in a different point of the surface. The cycle uses this second probing to calculate the inclination of the part.

Approach movement. Movement parallel to the probed side to touch a different point of the same side, located at 1/4 the length programmed in ·Ly·.

Probing movement. Probing movement at feedrate ·F· until the probe signal is received.

Withdrawal movement. The probe returns to the safety position located at a ·Ds· distance from the part.

5 Approach movement to the second probing point.

Movement of the probe from the first approach point to the second located in front of the first one.

The perpendicular axis moves the ·Dz· distance up in rapid (G00). Then the probe moves over the part, at the selected feedrate, to the next approach point. The probe moves a ·Dz· distance back down at feedrate ·Fs·

For the movement, the cycle takes into consideration the dimensions of the part and, if it is rectangular, its inclination angle. If in this last movement, the probe touches the part, the CNC shows an error at the cycle parameter value.

6 Probing of the second point.

It is done like the previous one. The probe returns to the safety position located at a ·Ds· distance from the part.

7 If part centering on two axes has been programmed, approach movement to the third probing point.

Movement of the probe from the second approach point to the center of the part and then to the third point.

8 If part centering on two axes has been programmed, probing of the third point.

It is done like the previous one.

9 Approach movement to the top surface (only if it has been programmed).

When selecting to measure the surface, the probe moves the ·Dz· distance up in rapid (G00) and the probe moves to the center of the part.

10 Measuring the top surface of the boss or the bottom of the pocket (only if it has been programmed).

The probe makes a first probing move, a withdrawal movement and a second probing move. After the second probing move, the probe withdraws to the approach point.

First probing movement. Probing movement, at feedrate ·Fs·, until touching the part surface.

Withdrawal movement. Return of the probe in rapid (G00) the distance indicated in ·Dr·.

Second probe. Probing movement at feedrate ·F· until the probe signal is received.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the safety position.

11 If part centering on two axes has been programmed, approach movement to the fourth probing point.

Movement of the probe from the third approach point to the fourth.

12 If part centering on two axes has been programmed, probing of the four point.

It is done like the previous one.

13 Probe positioning at the calculated center.

This movement is carried out in two stages. It first moves along the perpendicular axis in rapid (G00) and then in the plane.

14 If programmed, the cycle saves the result of the measurement in the selected G159 zero offset and in the corresponding arithmetic parameters.

15 The cycle executes the "M-after" functions.

4.13 Simulating a cycle from the editor.

At the canned cycle editor, it is possible to simulate the cycle being edited without having to simulate the whole part-program. During simulation, another canned cycle may be viewed and edited and it is also possible to return to the program editor.

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

Simulating a cycle.

Pressing the [START] softkey begins the simulation of the cycle that is being edited. The simulation may be interrupted with the [STOP] softkey or canceled with the [RESET] softkey. The simulation graphics is always superimposed on the help graphics of the main cycle.

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

Cycle simulation window.

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

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

Configuring the graphic environment.

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

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

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

Best area for displaying the graphics.

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

While the graphics window is visible, the key combination [CTRL]+[D] activates the calculation of the best area. From that moment on and until quitting the cycle editor, the CNC periodically calculates the best display are for the graphics.

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

Window for simulation and data editing.

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

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

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

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

Summary of the quick keyboard methods.

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

User notes:

User notes:

FAGOR

FAGOR AUTOMATION

Fagor Automation S. Coop.

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

AENOR
ER
Empresa Registrada
ER-0731/1994

CERTIFIED
IQNet
MANAGEMENT SYSTEM