CNCelite

8058/8060
8065

Probing (·T· model).

Ref. 2106

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

FAGOR AUTOMATION

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

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

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

INDEX

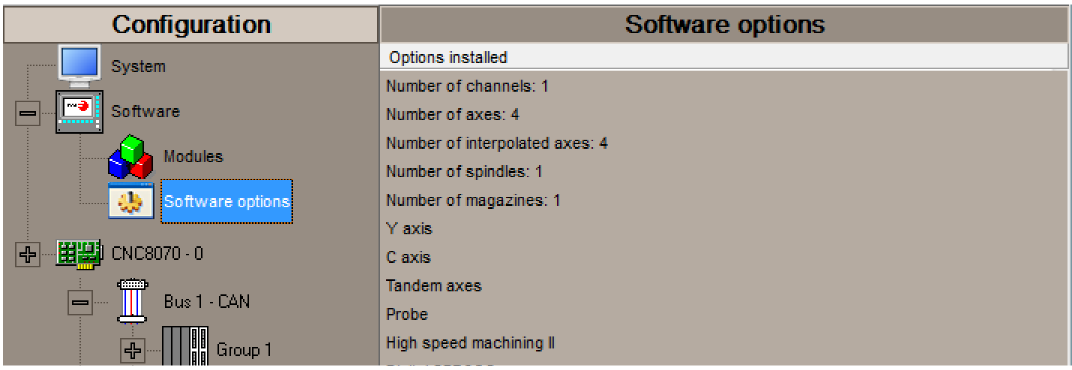
ABOUT THIS MANUAL.

Title.	Probing (·T· 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 T 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_qc_58_60_65_t_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.	v1.10
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

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 m^3. 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 m^3. 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).

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.

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

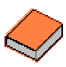
Symbols that may appear in the manual.

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.

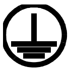
Obligation symbol.
This symbol indicates actions and operations that must be carried out.

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).

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.

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.

NEW FEATURES.

Manual reference: Ref. 2106
Date of publication: June, 2021
Associated software: v1.10

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.

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.

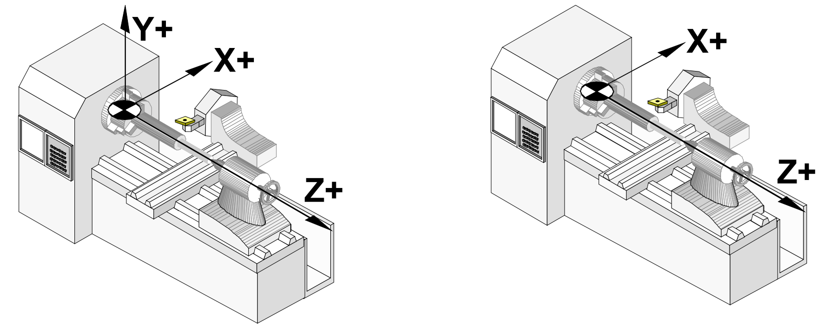
Knowing which is the active probe.

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.

1.2 Geometric configuration of axes and work planes.

The CNC admits two types of geometric configurations; "Trihedron" type and "Plane" type axis configuration.



Trihedron

Plane

Configuration of "Trihedron" type axes.

In this configuration, there are three axes forming a Cartesian XYZ type trihedron like on a milling machine. There may be more axes besides those forming the trihedron.

With this configuration, the planes behave in the same way as on a milling machine except that the usual work plane will be G18 (if it has been configured like that).

All the movements of these cycles are executed on the X Y Z axes; the work plane must be formed by 2 of these axes (XY, XZ, YZ, YX, ZX, ZY). The other axis, that must be perpendicular to that plane must be selected as axis perpendicular to the work plane.

Configuration of "plane" type axes.

In this configuration, there are two axes forming the usual work plane. There may be more axes, but they cannot be part of the trihedron; there must be auxiliary, rotary, etc.

With this configuration, the work plane is always G18 and will be formed by the first two axes defined in the channel. If the X (first) and Z (second) axes have been defined, the work plane will be the ZX (Z as abscissa and X as ordinate).

The probing movements can only be executed in the work plane. The CNC ignores the programmed variables that are related to the axis perpendicular to the work plane.

Configuration of "plane" type axes. Plane selection.

The work plane is always G18; machine parameter IPLANE is not applied and it is not possible to change planes via part-program. The following functions have these effects:

G17 It does not change planes and shows a warning about it.

G18 It has no effect.

G19 It does not change planes and shows a warning about it.

G20 It is permitted if it does not change the main plane; i.e. it can only be used to change the longitudinal axis.

The ·G· functions associated with the work planes are not displayed because it is always the same plane.

1.3 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.

1.4 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).

1.4.1 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
```

1.5 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.

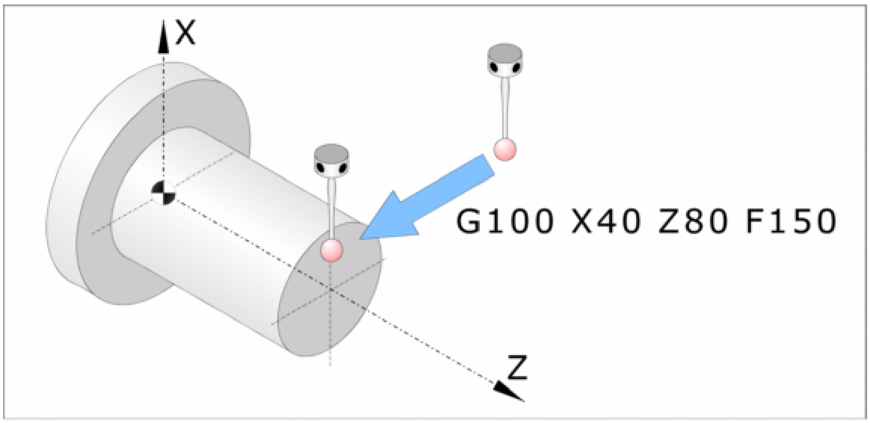
```
G100 X45.23 Z23.45
G100 Z50 F100

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.

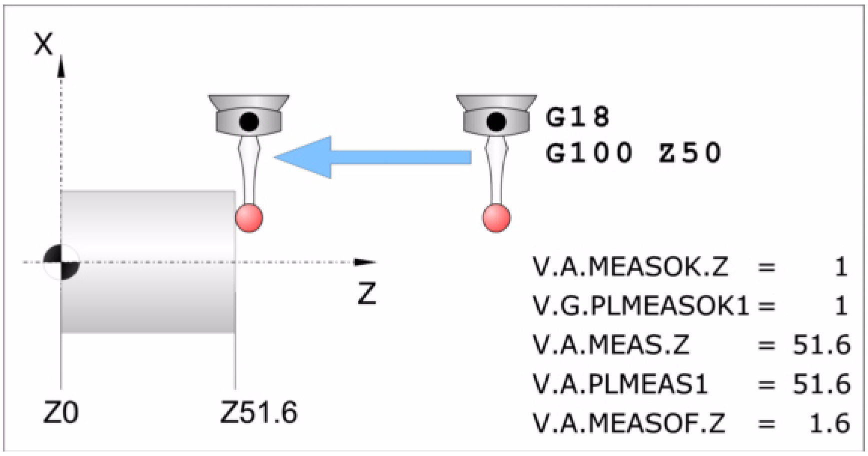
Functions G100 and G103 are not modal. After executing one of these functions, the CNC restores the function G0, G1, G2 ó G3, G33 or G63 that was active.

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.

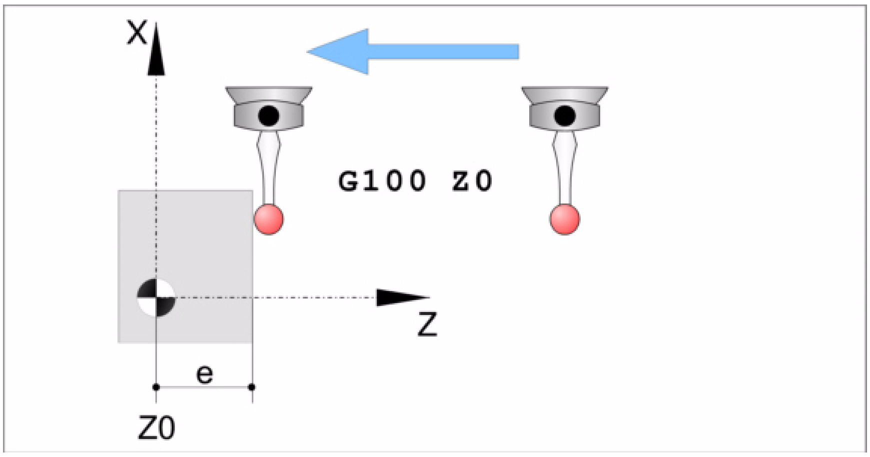
Mnemoni.	Variable.
V.G.MEASOK	The probe has made contact (G100) or stopped making contact (G103). • The variable takes the value of ·1· if the probe has made contact (G100) or has stopped making contact (G103). • The variables takes the value of ·0· if the probe reaches the programmed coordinate.
V.A.MEASOK.xn	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.
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.

Mnemoni.	Variable.
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. The measuring error is given in the active units, radius or diameter.



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.

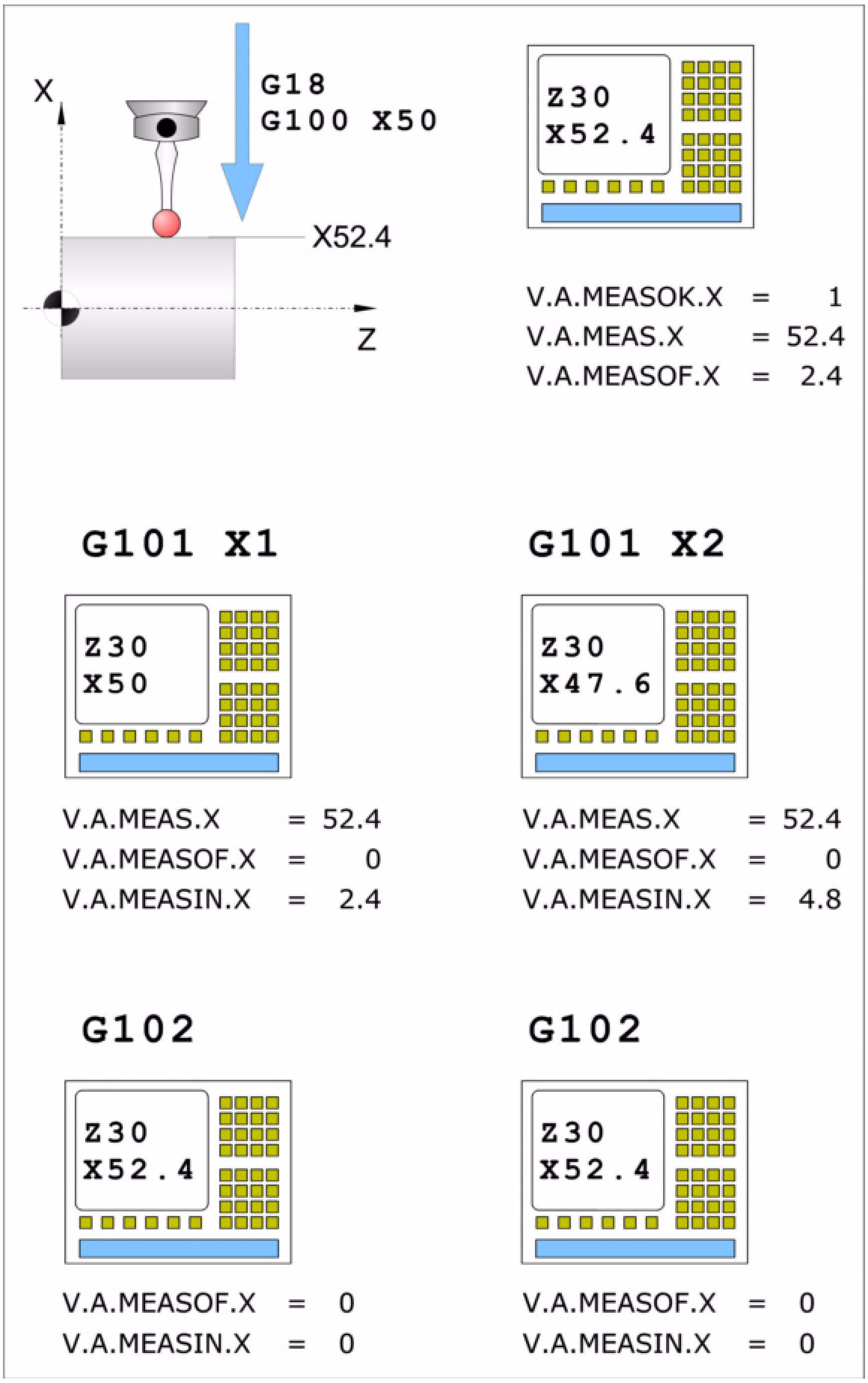
G101 X..C

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

```
G101 X1 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 Z
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. "0" = No "1"= Yes
(V.)[n].G.PLMEASOK2	R	—	—	Probing done on the second axis of the plane. "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

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.

#PROBE 1	Tool calibration.
#PROBE 2	Tabletop probe calibration
#PROBE 3	Part measuring along the ordinate axis.
#PROBE 4	Part measuring along the abscissa axis.

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 Z20 B5 F10
```

```
P1=4 P2=10
#PROBE P1 XP2 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.

A canned cycle may be defined anywhere in the program, that is, in the main program as well as in a subroutine. When working in a plane other than the ZX, the CNC interprets the canned cycle parameters as follows:

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

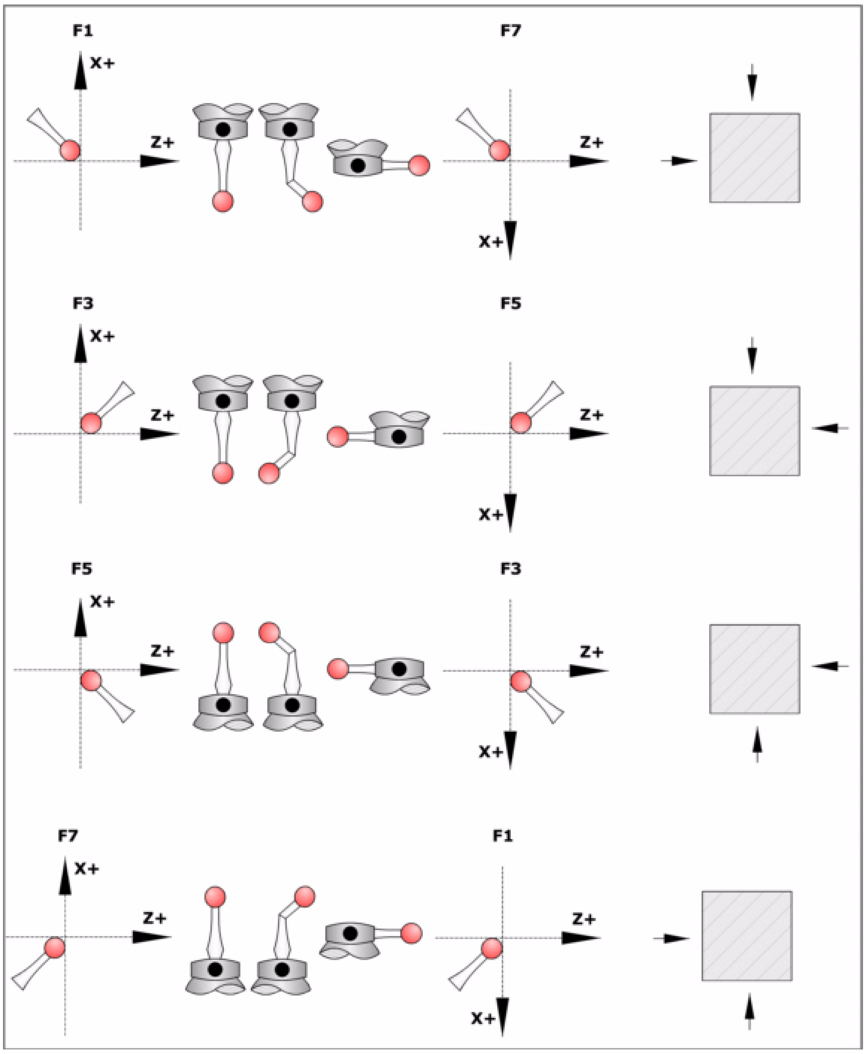
This cycle may be used to calibrate the dimensions of a tool or a touch probe. Once the cycle has concluded, it updates the dimensions in the tool table and initializes the tool wears to 0 (zero).

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 the tool or the probe is being calibrated, enter in the tool table an approximate dimensions, location code and the radius value. If it is a probe, the "R" value will correspond to the radius of the probe ball and the location code will depend on how it has been calibrated.



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 the measurement.

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 real tool length and the value assigned in the table.

P298 Error detected along the abscissa axis.

This value is given in radius.

P299 Error detected along the ordinate axis.

P297 Error detected on the axis perpendicular to the plane.

This value is given in radius.

Once the cycle has concluded, it updates the dimensions in the tool table and initializes the tool wears to 0 (zero).

3.1.1 Programming the cycle.

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

#PROBE 1 B F <K> <X U Z W Y V>

B	Safety distance.
F	Probing feedrate.
K	Optional. Sides of the probe to be used.
X··W	Optional. Tabletop probe position.

·B· Safety distance.

This parameter only admits positive values greater than 0 (zero). Value defined in radius.

Distance with respect to the point to touch, to which the tool approaches in G00 before making the probing movement. When calling the cycle, the tool 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.

·K· Sides of the probe to be used.

Optional parameter, by default 0.

This parameter indicates how many sides of the probe will be used for calibration. In a "Plane" type of axis configuration, two sides of the probe will always be used. In a "Trihedron" type of axis configuration, it is possible to choose to use either two or three sides of the probe.

K=0	Calibration on the X, Z sides.
K=1	Calibration on the X, Z, Y+ sides.
K=2	Calibration on the X, Z, Y- sides.

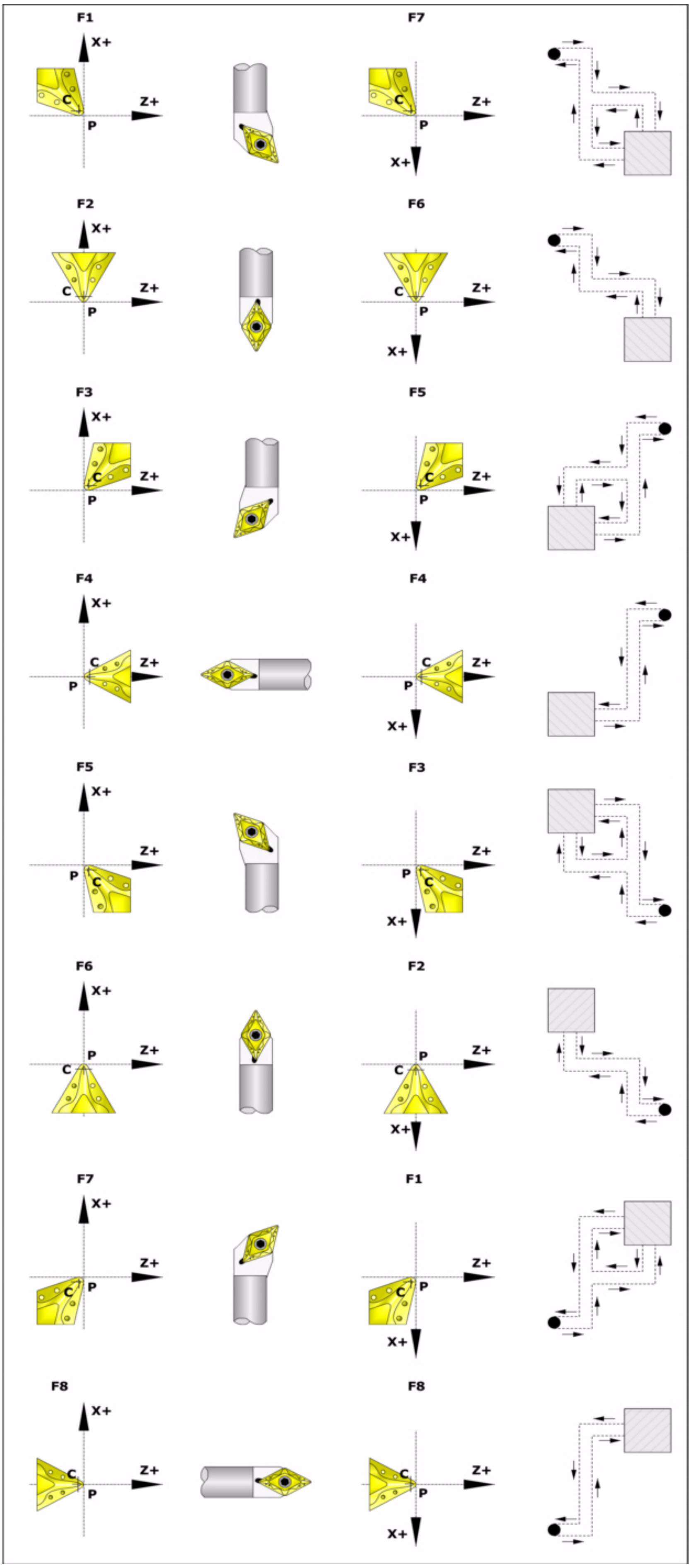
·X U Y V Z W· Tabletop probe position.

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 Z Y 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 W V 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.

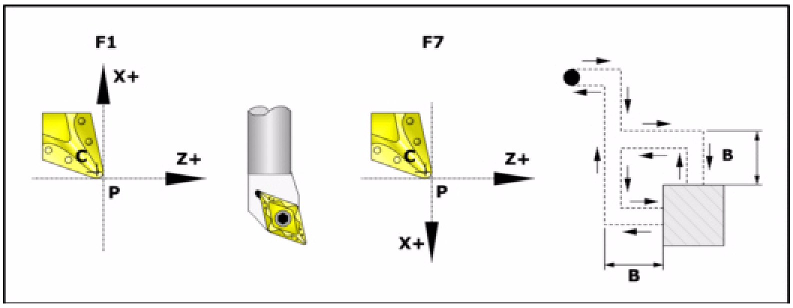
3.1.2 Basic operation.



1 Approach movement.

Rapid probe movement (G00) from the cycle calling point to the approach corner. This point is located in front of the associated probe corner, at a ·B· distance from it.

This approach movement is made in two stages. It first moves along the Z axis and then along the X axis.



2 Probing movement.

The sides of the probe used in this probing move as well as the path traveled by the tool depend on the location code assigned to the selected tool. When having a "Trihedron" type geometrical configuration and the ·K· parameter has been defined with a value other than zero, it will execute an additional probing move on the Y axis.

Each probing move will consist of an approach move, a probing move per se and a withdrawal move.

Approach movement. Rapid probe move (G00) to the approach point located in front of the side to be probed at a ·B· distance from it.

Probing movement. Probing movement at the indicated 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 corner.

3 Withdrawal movement.

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

This withdrawal movement is made in two stages. It first moves along the X axis and then along the Z axis.

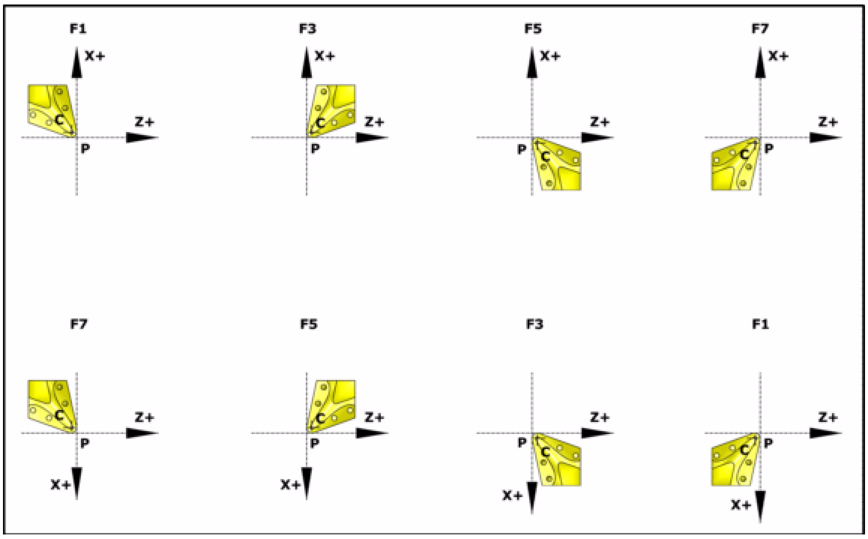
3.2 #PROBE 2. Tabletop probe calibration

This cycle may be used calibrate the sides of the tabletop probe. Once the cycle has ended, the user must enter the data returned by the cycle into the machine parameters that define the position of the probe.

The calibration is carried out with a tool of known dimensions.

Requirements prior to the calibration.

To execute the cycle, use a master tool whose dimensions have already been defined in the tool table. Since the probe needs to be calibrated along the X and Z axes, the location code of the master tool must be F1, F3, F5 or F7.



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: All the values will be given in absolute coordinates referred to machine reference zero.

P298	Real coordinate of the measured side along the abscissa axis.
P299	Real coordinate of the measured side along the ordinate axis. This value is given in radius.
P297	Real coordinate of the measured side along the axis perpendicular to the plane (if it has been measured). This value is given in radius.

Define the probe position.

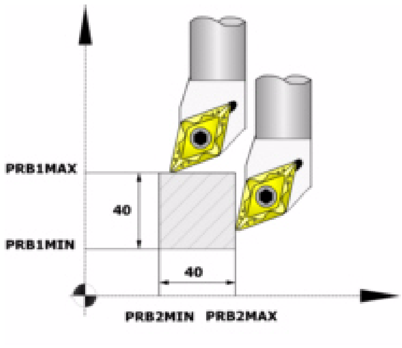
Once the values of these parameters and the probe dimensions are known, the user must calculate the coordinates of the other sides and update the following general machine parameters.

PRB1MIN	Minimum probe coordinate along the first axis of the channel.
PRB1MAX	Maximum probe coordinate along the first axis of the channel.
PRB2MIN	Minimum probe coordinate along the second axis of the channel.
PRB2MAX	Maximum probe coordinate along the second axis of the channel.
PRB3MIN	Minimum probe coordinate along the third axis of the channel.
PRB3MAX	Maximum probe coordinate along the third axis of the channel.

The probe position must be given in absolute coordinates referred to machine reference zero.

Example:

If the tool used has a location code F3 and the probe is square with 40 mm sides, the machine parameters will assume the following values.



PRB1MIN	= P298 - 40
PRB1MAX	= P298
PRB2MIN	= P299 - 40
PRB2MAX	= P299

3.2.1 Programming the cycle.

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

```
#PROBE 2 B F <K> <X U Z W Y V>
```

B	Safety distance.
F	Probing feedrate.
K	Optional. Sides of the probe to be used.
X··W	Optional. Tabletop probe position.

·B· Safety distance.

This parameter only admits positive values greater than 0 (zero). Value defined in radius.

Distance with respect to the point to touch, to which the tool approaches in G00 before making the probing movement. When calling the cycle, the tool 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.

·K· Sides of the probe to be used.

Optional parameter, by default 0.

This parameter indicates how many sides of the probe will be used for calibration. In a "Plane" type of axis configuration, two sides of the probe will always be used. In a "Trihedron" type of axis configuration, it is possible to choose to use either two or three sides of the probe.

K=0	Calibration on the X, Z sides.
K=1	Calibration on the X, Z, Y+ sides.
K=2	Calibration on the X, Z, Y- sides.

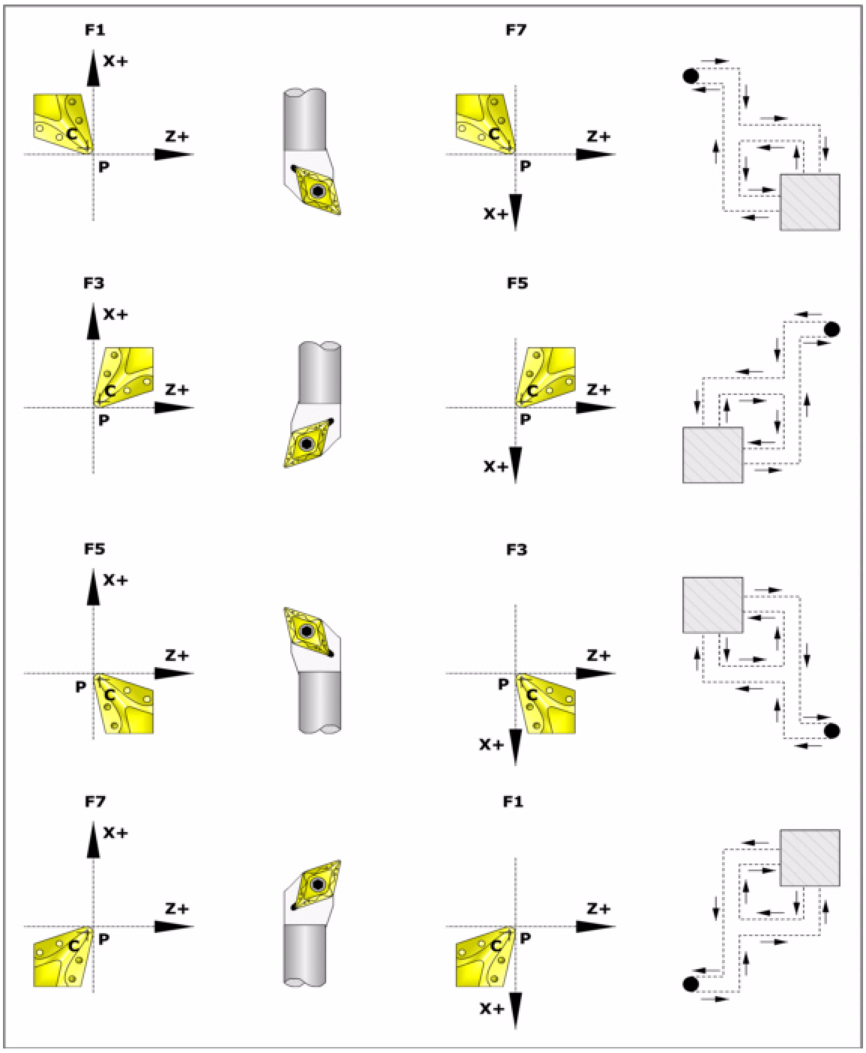
·X U Y V Z W· Tabletop probe position.

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 Z Y 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 W V 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.

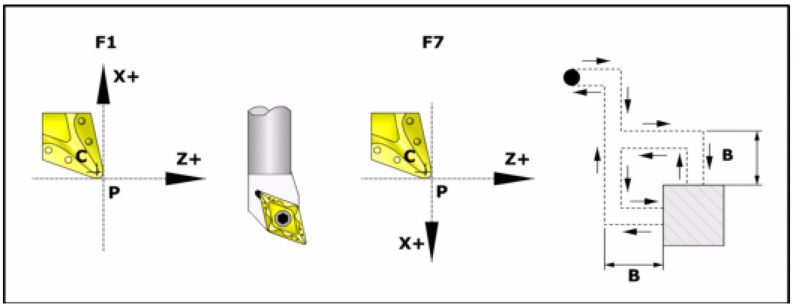
3.2.2 Basic operation.



1 Approach movement.

Rapid probe movement (G00) from the cycle calling point to the approach corner. This point is located in front of the associated probe corner, at a ·B· distance from it.

This approach movement is made in two stages. It first moves along the Z axis and then along the X axis.



2 Probing movement.

The sides of the probe used in this probing move as well as the path traveled by the tool depend on the location code assigned to the selected tool. When having a "Trihedron" type geometrical configuration and the ·K· parameter has been defined with a value other than zero, it will execute an additional probing move on the Y axis.

Each probing move will consist of an approach move, a probing move per se and a withdrawal move.

Approach movement. Rapid probe move (G00) to the approach point located in front of the side to be probed at a ·B· distance from it.

Probing movement. Probing movement at the indicated 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 corner.

3 Withdrawal movement.

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

This withdrawal movement is made in two stages. It first moves along the X axis and then along the Z axis.

3.3 #PROBE 3. Part measuring along the ordinate axis.

This cycle measures the part along the ordinate 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.

For this cycle, a probe mounted in the tool holding spindle must be used, it must be previously calibrated with the tool calibration canned cycle.

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.

This value is given in the active units, radius or diameter.

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

This value is given in radius.

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.

3.3.1 Programming the cycle.

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

#PROBE 3 X Z B F <L> <T D>

X Z	Theoretical coordinates of the measuring point.
B	Safety distance.
F	Probing feedrate.
L	Optional. Tolerance for the measuring error.
T	Optional. Tool to be corrected.
D	Optional. Tool offset to be corrected.

·X· Theoretical coordinate of the probing point along the ordinate axis.

Theoretical ordinate coordinate of the point being measured. This value is given in the active units, radius or diameter.

·Z· Theoretical coordinate of the probing point along the abscissa axis.

Theoretical abscissa coordinate of the point being measured.

·B· Safety distance.

This parameter only admits positive values greater than 0 (zero). Value defined in radius.

Distance with respect to the point to measure and along the ordinate 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.

·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.

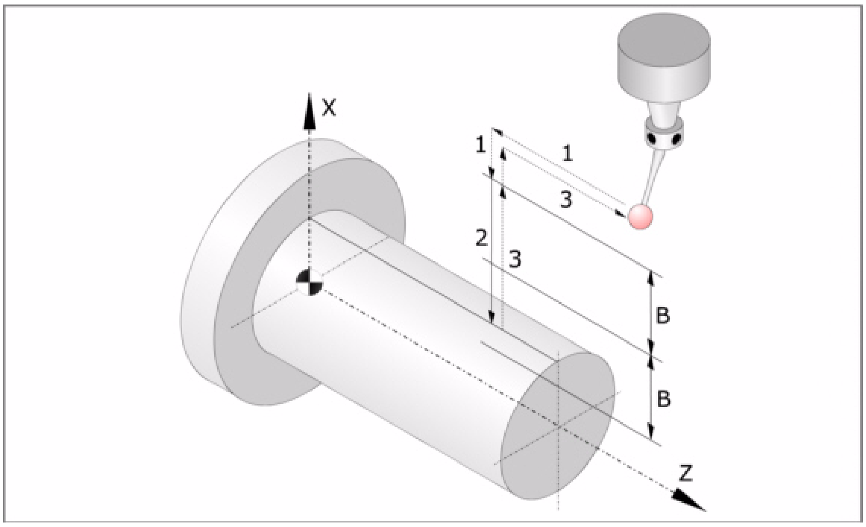
Tool whose wear is to be corrected, which will be the tool used to machine the surface.

·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.

3.3.2 Basic operation.



In the following description, the Z axis is the abscissa axis and the X axis is the ordinate axis.

1 Approach movement.

Rapid probe movement (G00) from the cycle calling point to the approach point. This point is located in front of the point being measured, at a ·B· distance from it.

This approach movement is made in two stages. It first moves along the Z axis and then along the X axis.

2 Probing movement.

Probing movement along the X axis at the indicated feedrate (F) until the probe signal is received. Once probing is over, the CNC will assume the actual position of the axes when the probe signal is received as their theoretical position.

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.

3 Withdrawal movement.

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

This withdrawal movement is made in two stages. It first moves along the X axis and then along the Z axis.

3.4 #PROBE 4. Part measuring along the abscissa axis.

This cycle measures the part along the abscissa 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.

For this cycle, a probe mounted in the tool holding spindle must be used, it must be previously calibrated with the tool calibration canned cycle.

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.

3.4.1 Programming the cycle.

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

#PROBE 4 X Z B F <L> <T D>

X Z	Theoretical coordinates of the measuring point.
B	Safety distance.
F	Probing feedrate.
L	Optional. Tolerance for the measuring error.
T	Optional. Tool to be corrected.
D	Optional. Tool offset to be corrected.

·X· Theoretical coordinate of the probing point along the ordinate axis.

Theoretical ordinate coordinate of the point being measured. This value is given in the active units, radius or diameter.

·Z· Theoretical coordinate of the probing point along the abscissa axis.

Theoretical abscissa coordinate of the point being measured.

·B· Safety distance.

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

Distance with respect to the point to measure and along the abscissa 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.

·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.

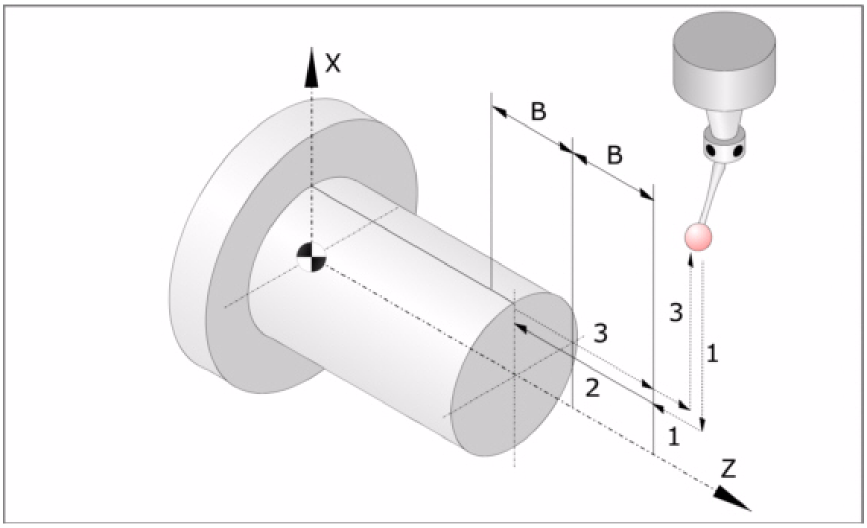
Tool whose wear is to be corrected, which will be the tool used to machine the surface.

·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.

3.4.2 Basic operation.



In the following description, the Z axis is the abscissa axis and the X axis is the ordinate axis.

1 Approach movement.

Rapid probe movement (G00) from the cycle calling point to the approach point. This point is located in front of the point being measured, at a ·B· distance from it.

This approach movement is made in two stages. It first moves along the X axis and then along the Z axis.

2 Probing movement.

Probing movement along the Z axis at the indicated feedrate (F) until the probe signal is received. Once probing is over, the CNC will assume the actual position of the axes when the probe signal is received as their theoretical position.

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.

3 Withdrawal movement.

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

This withdrawal movement is made in two stages. It first moves along the Z axis and then along the X axis.

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

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 4 X12.5 Z23.75 B5 F10
    V.C.X = 12.5
    V.C.Z = 23.75
    V.C.B = 5
    V.C.F = 10
```

CANNED CYCLES. CYCLE EDITOR.

4

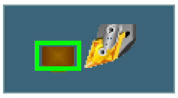
The cycles may be defined anywhere in the program, that is, in the main program as well as in a subroutine.

Programming the cycles of the editor.

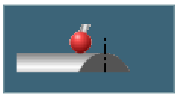
Using the configuration softkey, the user can select the graphics for vertical lathes. By default, it will show the graphics for horizontal lathes.

The cycles of the editor are accessed with the following softkeys:

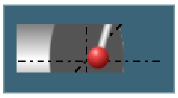
Tool calibration.



Tabletop probe calibration.



Part measurement along the ordinate axis.



Part measurement along the abscissa axis.

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 64.

Limitations for executing the cycles.

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

Canned cycles and the work planes.

A canned cycle may be defined anywhere in the program, that is, in the main program as well as in a subroutine. When working in a plane other than the ZX, the CNC interprets the canned cycle parameters as follows:

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

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.

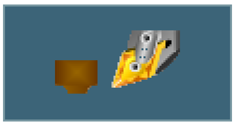
The Teach-in mode is activated from the horizontal softkey menu. When the Teach-in mode is active, the bottom of the screen shows a window with the axes of the channel.

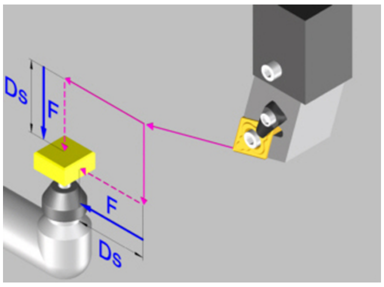
The Teach-in mode may be used to jog the axes and assign their current position to the data that define the position of the cycle. The axes may be jogged using the jog keypad, the handwheels or via MDI.

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.

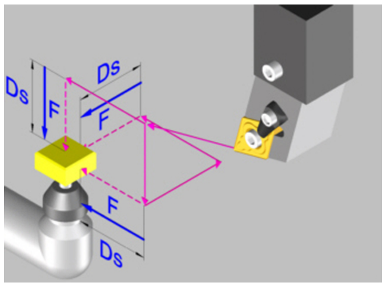
- The X axis related data takes the coordinate of the first axis of the channel.
- The Z axis related data takes the coordinate of the second axis of the channel, if the channel has only two axes. If there are three or more axes, the data takes the coordinate of the first axis of the channel.

4.2 Tool calibration.





Geometrical configuration of "Plane" type axes.



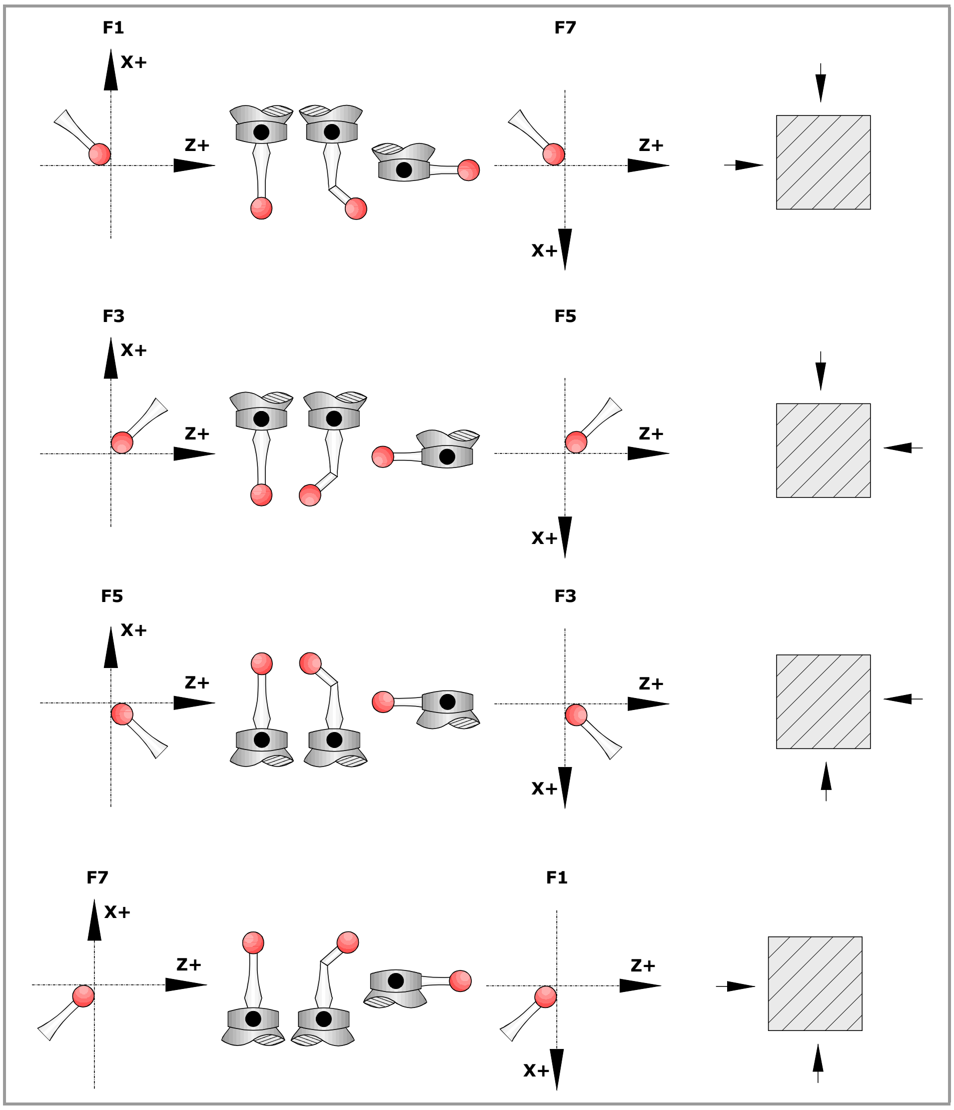
Geometrical configuration of "trihedron" type axes.

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 initializes the tool wears to 0 (zero).

The calibration is done using a tabletop probe.

Requirements prior to the calibration.

If it is the first time the tool or the probe is being calibrated, enter in the tool table an approximate dimensions, location code and the radius value. If it is a probe, the "R" value will correspond to the radius of the probe ball and the location code will depend on how it has been calibrated.



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 the measurement.

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 real tool length and the value assigned in the table.

P298 Error detected along the abscissa axis.

This value is given in radius.

P299 Error detected along the ordinate axis.

P297 Error detected on the axis perpendicular to the plane.

This value is given in radius.

Once the cycle has concluded, it updates the dimensions in the tool table and initializes the tool wears to 0 (zero).

4.2.1 Programming the cycle.

Tool to be calibrated.

·Tp· Tool to be calibrated.

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

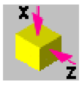
·Dp· Tool offset

Offset of the tool to be calibrated.

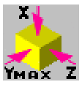
Probing movement.

·icon· Axes along which calibration takes place.

This parameter indicates how many sides of the probe will be used for calibration. In a "Plane" type of axis configuration, two sides of the probe will always be used. In a "Trihedron" type of axis configuration, it is possible to choose to use either two or three sides of the probe.



Calibration along the abscissa and ordinate axes of the work plane.



Calibration along the abscissa and ordinate axes of the work plane. Additional calibration along the axis perpendicular to the plane, in the negative direction (Y+ side).

Calibration along the abscissa and ordinate axes of the work plane. Additional calibration along the axis perpendicular to the plane, in the positive direction (Y- side).

·Ds· Safety distance.

This parameter only admits positive values greater than 0 (zero). Value defined in radius.

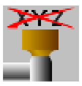
Distance with respect to the point to touch, to which the tool approaches in G00 before making the probing movement. When calling the cycle, the tool must be located, with respect to the point to be measured, at a greater distance than this value

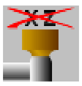
·F· Probing feedrate.

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

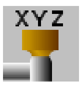
Probe coordinates.

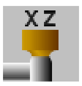
·icon· Redefine the tabletop probe position.





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.

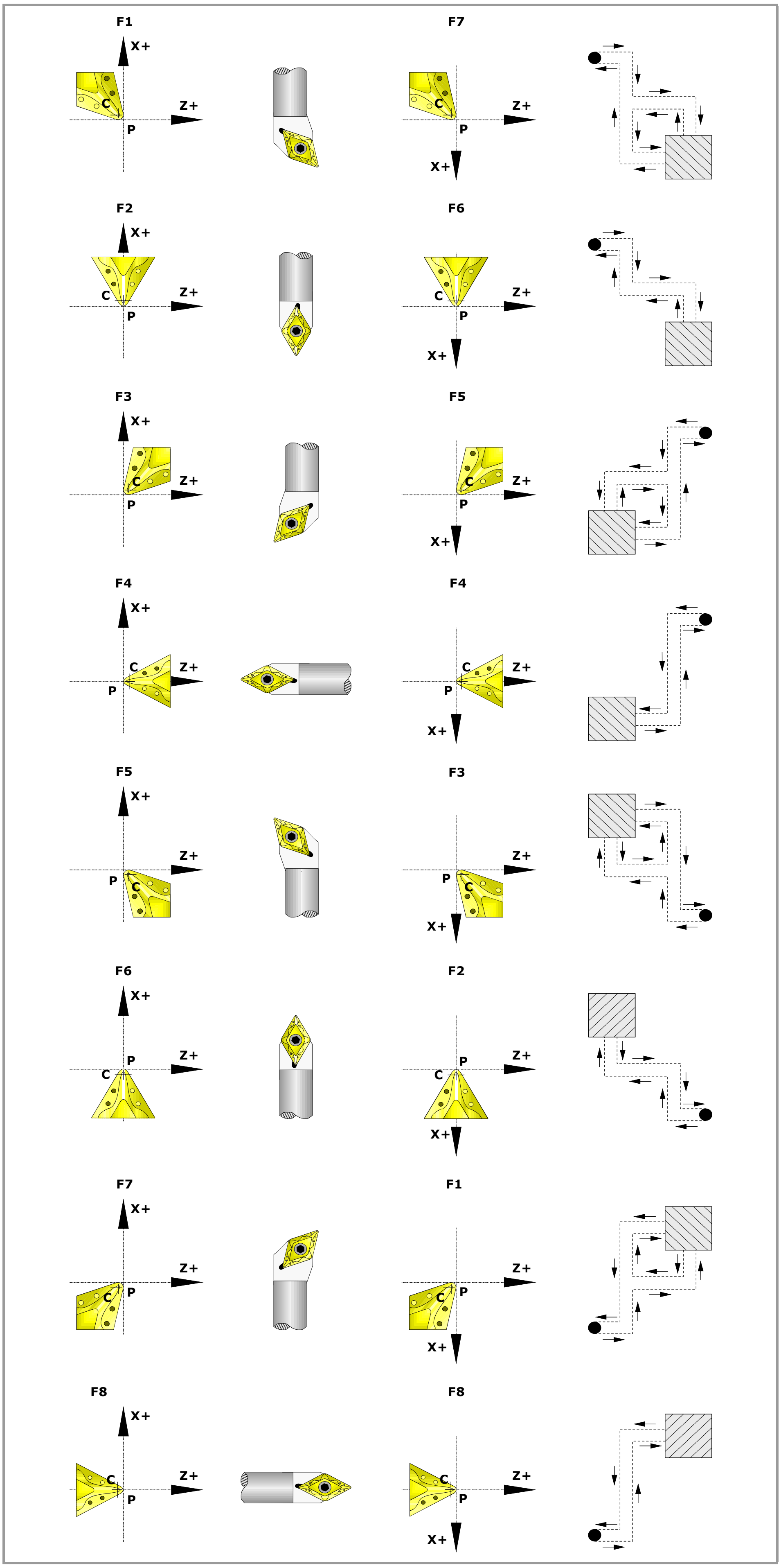
·M after· M functions to be executed after the cycle.

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.

4.2.2 Basic operation.



1 The cycle selects the programmed tool.

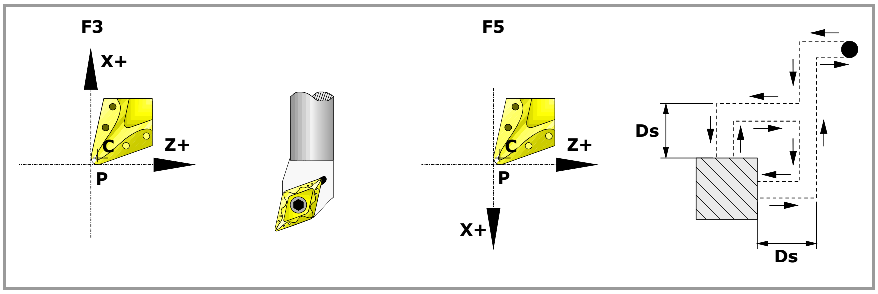
2 The CNC runs the subroutine Sub_Probe_Tool_Begin.fst, defined by the OEM.

3 The cycle executes the "M-before" functions.

4 Approach movement.

Rapid probe movement (G00) from the cycle calling point to the approach corner. This point is located in front of the associated probe corner, at a ·Ds· distance from it.

This approach movement is made in two stages. It first moves along the Z axis and then along the X axis.



5 Probing movement.

The sides of the probe used in this probing move as well as the path traveled by the tool depend on the location code assigned to the selected tool. When having a "Trihedron" type geometrical configuration and three-axis probing has been defined, it will execute an additional probing move on the Y axis.

Each probing move will consist of an approach move, a probing move per se and a withdrawal move.

Approach movement. Rapid probe move (G00) to the approach point located in front of the side to be probed at a ·Ds· distance from it.

Probing movement. Probing movement at the indicated 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 corner.

6 Withdrawal movement.

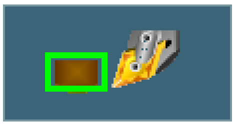
Rapid probe movement (G00) from the approach corner to the cycle calling point.

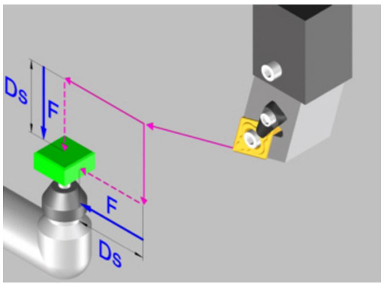
This withdrawal movement is made in two stages. It first moves along the X axis and then along the Z axis.

7 The cycle executes the "M-after" functions.

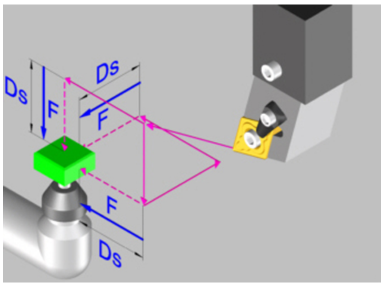
8 The CNC runs the subroutine Sub_Probe_Tool_End.fst, defined by the OEM.

4.3 Tabletop probe calibration.





Geometrical configuration of "Plane" type axes.



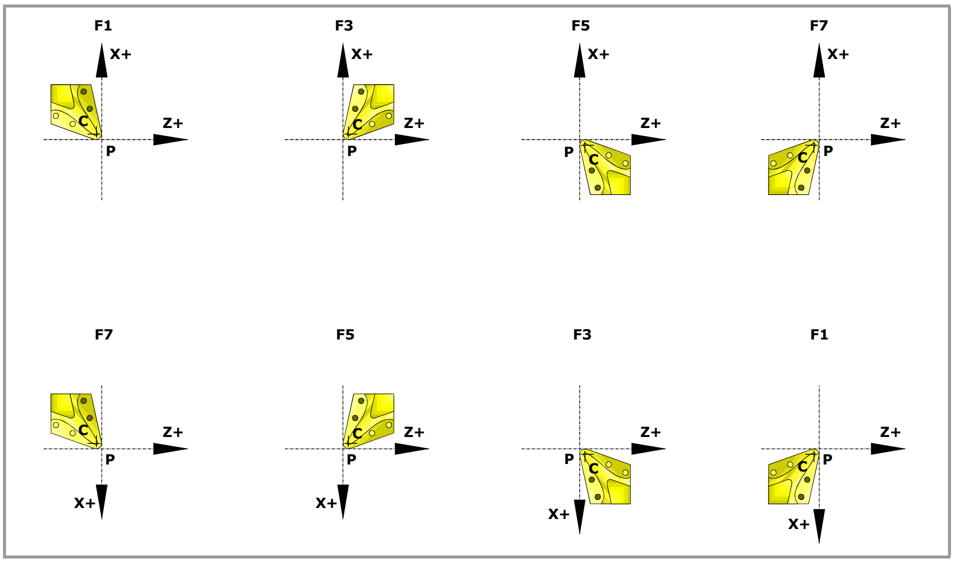
Geometrical configuration of "trihedron" type axes.

This cycle may be used calibrate the sides of the tabletop probe. Once the cycle has ended, the user must enter the data returned by the cycle into the machine parameters that define the position of the probe.

The calibration is carried out with a tool of known dimensions.

Requirements prior to the calibration.

To execute the cycle, use a master tool whose dimensions have already been defined in the tool table. Since the probe needs to be calibrated along the X and Z axes, the location code of the master tool must be F1, F3, F5 or F7.



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: All the values will be given in absolute coordinates referred to machine reference zero.

P298 Real coordinate of the measured side along the abscissa axis.

P299 Real coordinate of the measured side along the ordinate axis.

This value is given in radius.

P297 Real coordinate of the measured side along the axis perpendicular to the plane (if it has been measured).

This value is given in radius.

Define the probe position.

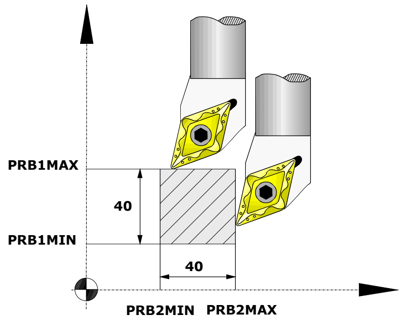
Once the values of these parameters and the probe dimensions are known, the user must calculate the coordinates of the other sides and update the following general machine parameters.

PRB1MIN	Minimum probe coordinate along the first axis of the channel.
PRB1MAX	Maximum probe coordinate along the first axis of the channel.
PRB2MIN	Minimum probe coordinate along the second axis of the channel.
PRB2MAX	Maximum probe coordinate along the second axis of the channel.
PRB3MIN	Minimum probe coordinate along the third axis of the channel.
PRB3MAX	Maximum probe coordinate along the third axis of the channel.

The probe position must be given in absolute coordinates referred to machine reference zero.

Example:

If the tool used has a location code F3 and the probe is square with 40 mm sides, the machine parameters will assume the following values.



PRB1MIN = P298 - 40

PRB1MAX = P298

PRB2MIN = P299 - 40

PRB2MAX = P299

4.3.1 Programming the cycle.

Tool to be calibrated.

·Tp· Tool to be used in the calibration.

Number of the tool used to calibrate the tabletop probe.

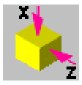
·Dp· Tool offset

Offset of the tool to be calibrated.

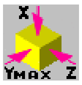
Probing movement.

·icon· Axes along which calibration takes place.

This parameter indicates how many sides of the probe will be used for calibration. In a "Plane" type of axis configuration, two sides of the probe will always be used. In a "Trihedron" type of axis configuration, it is possible to choose to use either two or three sides of the probe.



Calibration along the abscissa and ordinate axes of the work plane.



Calibration along the abscissa and ordinate axes of the work plane. Additional calibration along the axis perpendicular to the plane, in the negative direction (Y+ side).

Calibration along the abscissa and ordinate axes of the work plane. Additional calibration along the axis perpendicular to the plane, in the positive direction (Y- side).

·Ds· Safety distance.

This parameter only admits positive values greater than 0 (zero). Value defined in radius.

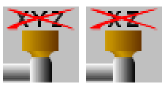
Distance with respect to the point to touch, to which the tool approaches in G00 before making the probing movement. When calling the cycle, the tool must be located, with respect to the point to be measured, at a greater distance than this value

·F· Probing feedrate.

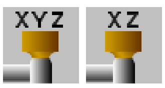
This parameter sets the probing feedrate. The rest of the movements will be carried out in G00.

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.

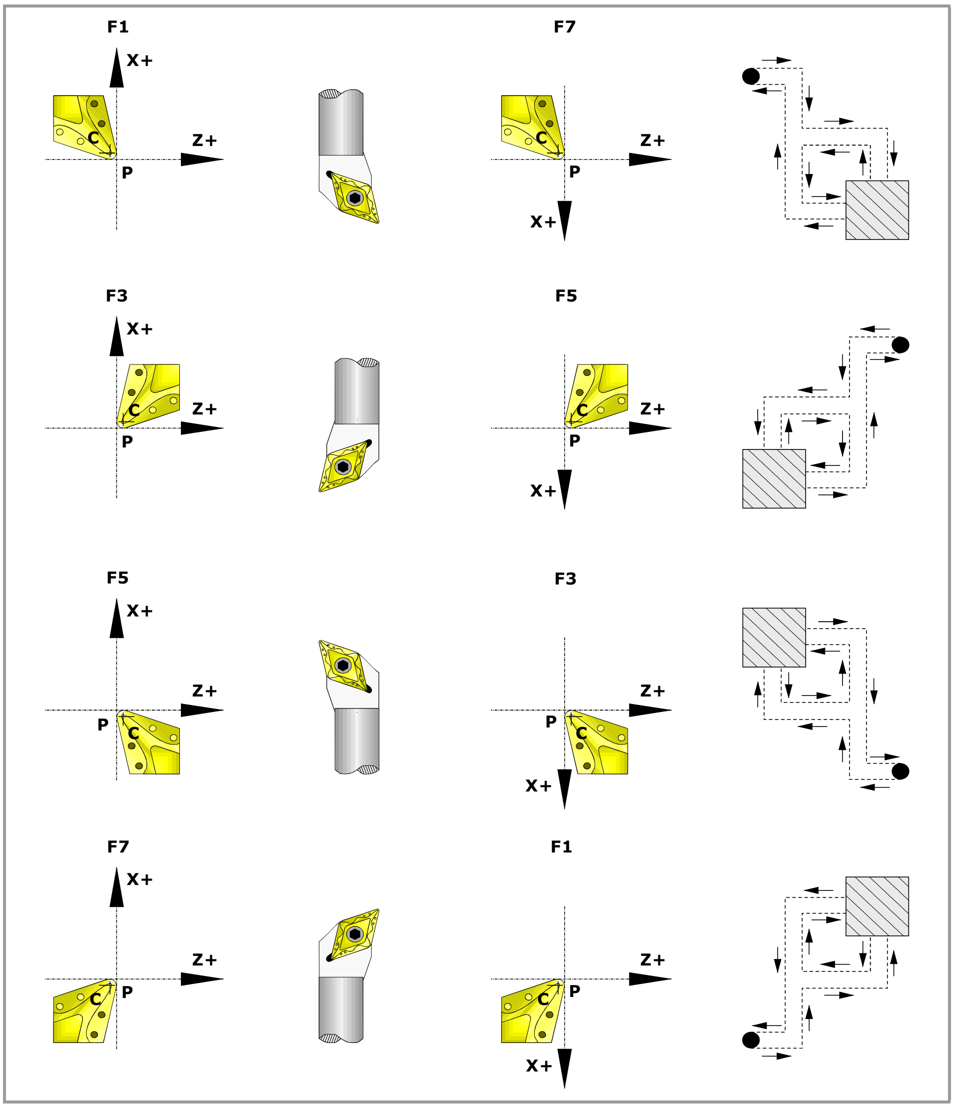
The cycle allows executing up to 4 M functions before the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

·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.

4.3.2 Basic operation.



1 The cycle selects the programmed tool.

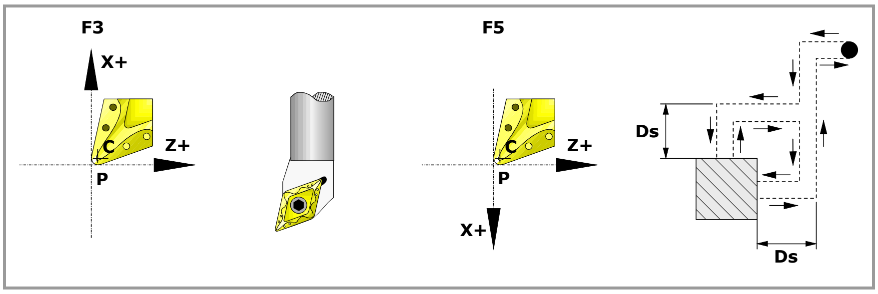
2 The CNC runs the subroutine Sub_Probe_Tool_Begin.fst, defined by the OEM.

3 The cycle executes the "M-before" functions.

4 Approach movement.

Rapid probe movement (G00) from the cycle calling point to the approach corner. This point is located in front of the associated probe corner, at a ·Ds· distance from it.

This approach movement is made in two stages. It first moves along the Z axis and then along the X axis.



5 Probing movement.

The sides of the probe used in this probing move as well as the path traveled by the tool depend on the location code assigned to the selected tool. When having a "Trihedron" type geometrical configuration and three-axis probing has been defined, it will execute an additional probing move on the Y axis.

Each probing move will consist of an approach move, a probing move per se and a withdrawal move.

Approach movement. Rapid probe move (G00) to the approach point located in front of the side to be probed at a ·Ds· distance from it.

Probing movement. Probing movement at the indicated 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 corner.

6 Withdrawal movement.

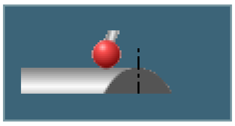
Rapid probe movement (G00) from the approach corner to the cycle calling point.

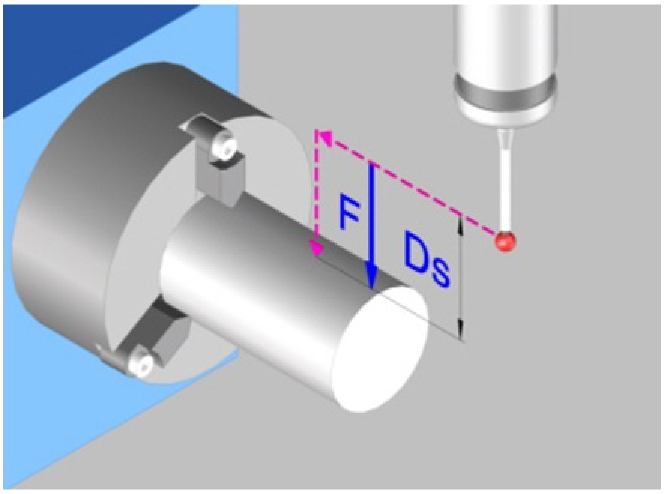
This withdrawal movement is made in two stages. It first moves along the X axis and then along the Z axis.

7 The cycle executes the "M-after" functions.

8 The CNC runs the subroutine Sub_Probe_Tool_End.fst, defined by the OEM.

4.4 Part measuring along the ordinate axis.





This cycle measures the part along the ordinate 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.

For this cycle, a probe mounted in the tool holding spindle must be used, it must be previously calibrated with the tool calibration canned cycle.

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.

This value is given in the active units, radius or diameter.

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

This value is given in radius.

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.

4.4.1 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.

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

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

Probing movement.

·X· Theoretical coordinate of the probing point along the ordinate axis.

Theoretical ordinate coordinate of the point being measured. This value is given in the active units, radius or diameter.

·Z· Theoretical coordinate of the probing point along the abscissa axis.

Theoretical abscissa coordinate of the point being measured.

·Ds· Safety distance.

This parameter only admits positive values greater than 0 (zero). Value defined in radius.

Distance with respect to the point to measure and along the ordinate axis, 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.

·TW· 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·.

Tool wear compensation.

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 does not correct any tool wear.

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·.

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.

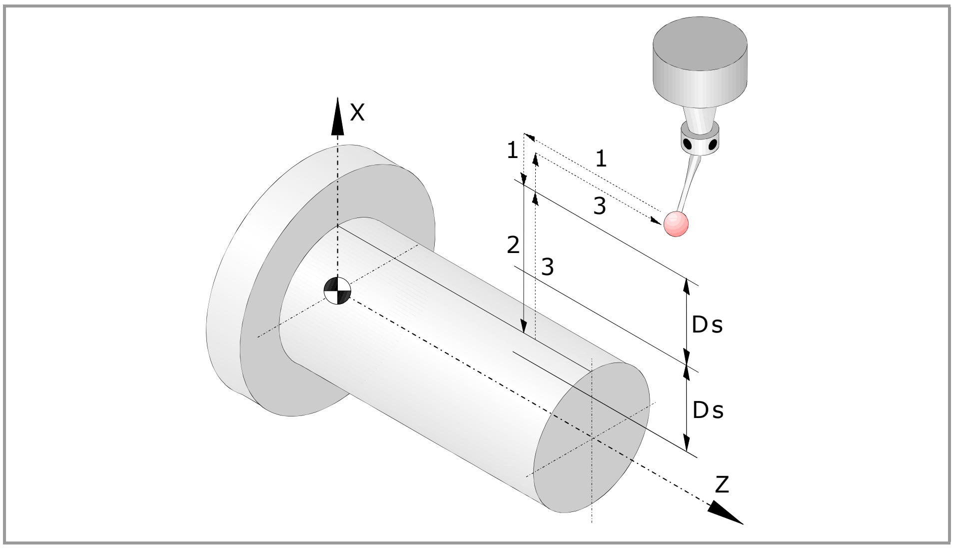
i

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.

In the following description, the Z axis is the abscissa axis and the X axis is the ordinate axis.



1 The cycle selects the programmed tool.

2 The CNC runs the subroutine Sub_Probe_Piece_Begin.fst, defined by the OEM.

3 The cycle executes the "M-before" functions.

4 Approach movement.

Rapid probe movement (G00) from the cycle calling point to the approach point. This point is located in front of the point being measured, at a ·Ds· distance from it.

This approach movement is made in two stages. It first moves along the Z axis and then along the X axis.

5 Probing movement.

Probing movement along the X axis at the indicated feedrate (F) until the probe signal is received. Once probing has been made, the CNC will assume as their theoretical position the real position of the axes when 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.

6 Withdrawal movement.

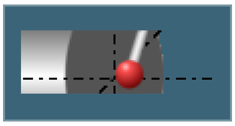
Rapid probe movement (G00) from the probing point to the cycle calling point.

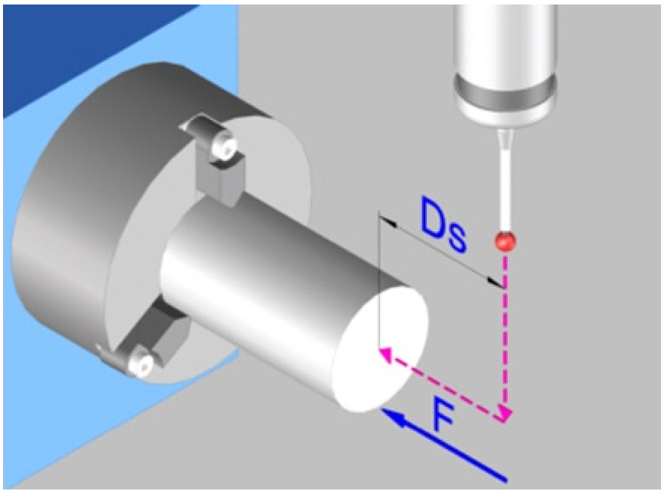
This withdrawal movement is made in two stages. It first moves along the X axis and then along the Z axis.

7 The cycle executes the "M-after" functions.

8 The CNC runs the subroutine Sub_Probe_Piece_End.fst, defined by the OEM.

4.5 Part measuring along the abscissa axis.





This cycle measures the part along the abscissa 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.

For this cycle, a probe mounted in the tool holder must be used, it must be previously calibrated with the tool calibration canned cycle.

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.

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.

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

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

Probing movement.

·X· Theoretical coordinate of the probing point along the ordinate axis.

Theoretical ordinate coordinate of the point being measured. This value is given in the active units, radius or diameter.

·Z· Theoretical coordinate of the probing point along the abscissa axis.

Theoretical abscissa coordinate of the point being measured.

·Ds· Safety distance.

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

Distance with respect to the point to measure and along the abscissa axis, 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.

Tool wear compensation.

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 does not correct any tool wear.

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·.

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.

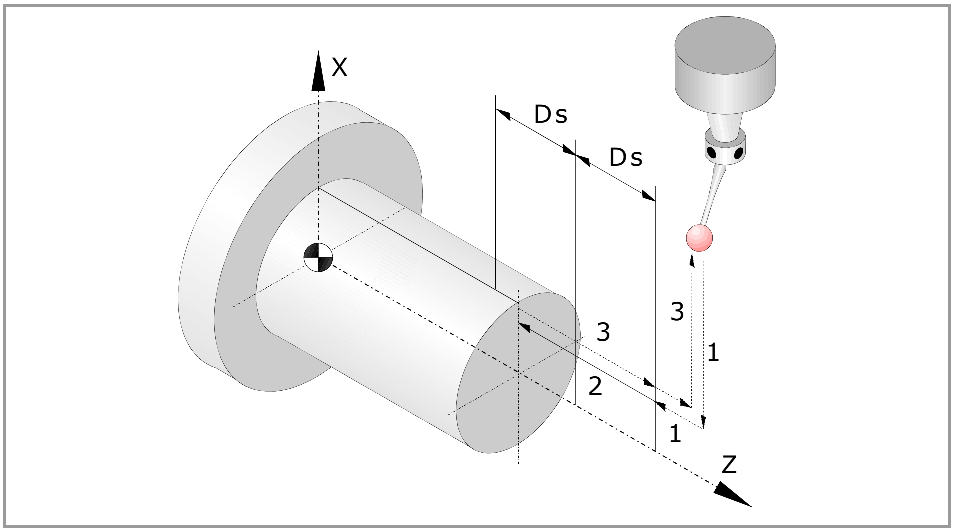
i

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.

In the following description, the Z axis is the abscissa axis and the X axis is the ordinate axis.



1 The cycle selects the programmed tool.

2 The CNC runs the subroutine Sub_Probe_Piece_Begin.fst, defined by the OEM.

3 The cycle executes the "M-before" functions.

4 Approach movement.

Rapid probe movement (G00) from the cycle calling point to the approach point. This point is located in front of the point being measured, at a ·Ds· distance from it.

This approach movement is made in two stages. It first moves along the X axis and then along the Z axis.

5 Probing movement.

Probing movement along the Z axis at the indicated feedrate (F) until the probe signal is received. Once probing has been made, the CNC will assume as their theoretical position the real position of the axes when 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.

6 Withdrawal movement.

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

This withdrawal movement is made in two stages. It first moves along the Z axis and then along the X axis.

7 The cycle executes the "M-after" functions.

8 The CNC runs the subroutine Sub_Probe_Piece_End.fst, defined by the OEM.

4.6 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] [G]	It shows the graphics window when a simulation is running and the parameter editing window is active.
[ESC]	If the graphics are shown at full screen, it shows the cycle editor screen.

User notes:

User notes:

User notes:

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
Empresa Registrada
ER-073/1994