CNC 8060 8065

Probing (·T· model)

(Ref: 1603)

FAGOR AUTOMATION

MACHINE SAFETY

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

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

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

HARDWARE EXPANSIONS

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

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

COMPUTER VIRUSES

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

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

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, thus no absolute match is guaranteed. Anyway, the contents of the manual is periodically checked making and including the necessary corrections in a future edition. We appreciate your suggestions for improvement.

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

INDEX

FAGOR

CNC 8060
CNC 8065

(REF: 1603)

ABOUT THE PRODUCT - CNC 8060

BASIC CHARACTERISTICS.

Basic characteristics.	8060 M FL	8060 M	8060 T FL	8060 T	8060 L
PC-based system.	Closed system. Without access to the administrator mode.				
Number of axes.	3 to 4	3 to 6	3 to 4	3 to 6	3 to 4
Number of spindles.	1	1 to 2	1 to 2	1 to 3	1
Number of tool magazines.	1	1	1	1 to 2	1
Number of execution channels.	1	1	1	1 to 2	1
Number of handwheels.	1 to 3				
Type of servo system.	Analog / Sercos Digital				
Communications.	RS485 / RS422 / RS232 Ethernet				
Integrated PLC. PLC execution time. Digital inputs / Digital outputs. Marks / Registers. Timers / Counters. Symbols.	 < 1ms/K 1024 / 1024 8192 / 1024 512 / 256 Unlimited				
Block processing time.	< 2,0 ms	< 1,5 ms	< 2,0 ms	< 1,5 ms	< 2,0 ms

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

DISCONTINUED

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

SOFTWARE OPTIONS.

Bear in mind that some of the features described in this manual depend on the software options that are installed. The information of the following table is informative only; when purchasing the software options, only the information provided in the ordering handbook is valid.

Software options.

	8060 M FL	8060 M	8060 T FL	8060 T	8060 L
Number of execution channels.	1	1	1	1 to 2	1
Number of axes.	3 to 4	3 to 6	3 to 4	3 to 6	3 to 4
Number of spindles.	1	1 to 2	1 to 2	1 to 3	1
Maximun number of axes and spindles.	5	7	5	7	5
Interpolated axes.	4	4	4	4	4
Number of tool magazines.	1	1	1	1 to 2	1
F3D graphics.	- - -	Option	- - -	Option	Option
Conversational IIP.	Option	Option	Option	Option	Option
"C" axis.	Option	Option	Option	Option	Option
"Y" axis for lathe.	- - -	- - -	- - -	Option	- - -
Dynamic RTCP.	- - -	Option	- - -	- - -	- - -
HSSA-I machining system.	Option	- - -	Option	- - -	Option
HSSA-II machining system.	- - -	Option	- - -	Option	- - -
Probing canned cycles.	Option	Option	Option	Option	Option
Tandem axes.	Option	Option	Option	Option	Option
Axis and spindle synchronization.	- - -	- - -	- - -	Option	Option
ProGTL3 language.	- - -	Option	- - -	- - -	Option
Part-program translator.	- - -	Option	- - -	Option	Option
Conversational user cycles.	Option	Option	Option	Option	Option
Kinematics calibration.	- - -	- - -	- - -	- - -	- - -
Third-party CANopen.	Option	Option	Option	Option	Option
MABSYSTEM.	Standard	Option	Standard	Option	- - -
GAP CONTROL	- - -	- - -	- - -	- - -	Option
PWM CONTROL	- - -	- - -	- - -	- - -	Option

ABOUT THE PRODUCT - CNC 8065

BASIC CHARACTERISTICS.

Basic characteristics.	8065 M	8065 T
PC-based system.	Open system. Access to the administrator mode.	
Number of axes.	3 to 28	
Number of spindles.	1 to 4	
Number of tool magazines.	1 to 4	
Number of execution channels.	1 to 4	
Number of handwheels.	1 to 12	
Type of servo system.	Analog / Digital Sercos / Digital Mechatrolink	
Communications.	RS485 / RS422 / RS232 Ethernet	
Integrated PLC. PLC execution time. Digital inputs / Digital outputs. Marks / Registers. Timers / Counters. Symbols.	 < 1ms/K 1024 / 1024 8192 / 1024 512 / 256 Unlimited	
Block processing time.	< 1 ms	

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

DISCONTINUED

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

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

(REF: 1603)

SOFTWARE OPTIONS.

Bear in mind that some of the features described in this manual depend on the software options that are installed. The information of the following table is informative only; when purchasing the software options, only the information provided in the ordering handbook is valid.

Software options (·M· model).

	8065 M		8065 M Power	
	Basic	**Pack 1**	**Basic**	**Pack 1**
Open system. Access to the administrator mode.	- - -	- - -	Option	Option
Number of execution channels.	1	1	1	1 to 4
Number of axes.	3 to 6	5 to 8	5 to 12	8 to 28
Number of spindles.	1	1 to 2	1 to 4	1 to 4
Maximun number of axes and spindles.	7	10	16	32
Number of tool magazines.	1	1	1 to 2	1 to 4
Limited to 4 interpolated axes.	Option	Option	Option	Option
IEC 61131 language.	- - -	Option	Option	Option
F3D/HD graphics.	Option	Option	Standard	Standard
Conversational IIP.	Option	Standard	Standard	Standard
Dual-purpose machines (M-T).	- - -	- - -	Option	Standard
"C" axis.	Standard	Standard	Standard	Standard
Dynamic RTCP.	- - -	Option	Option	Standard
HSSA machining system.	Standard	Standard	Standard	Standard
Probing canned cycles.	Option	Standard	Standard	Standard
Tandem axes.	- - -	Option	Standard	Standard
Axis and spindle synchronization.	- - -	- - -	Option	Standard
Tangential control.	- - -	Standard	Standard	Standard
Volumetric compensation (up to 10 m^3).	- - -	- - -	Option	Option
Volumetric compensation (more than 10 m^3).	- - -	- - -	Option	Option
ProGTL3 language.	Option	Option	Option	Option
Part-program translator.	Option	Option	Option	Option
Conversational user cycles.	Option	Option	Option	Option
Kinematics calibration.	Option	Option	Option	Option

Software options (·T· model).

	8065 T		8065 T Power	
	Basic	**Pack 1**	**Basic**	**Pack 1**
Open system. Access to the administrator mode.	- - -	- - -	Option	Option
Number of execution channels.	1	1 to 2	1 to 2	1 to 4
Number of axes.	3 to 5	5 to 7	5 to 12	8 to 28
Number of spindles.	2	2	3 to 4	3 to 4
Maximun number of axes and spindles.	7	9	16	32
Number of tool magazines.	1	1 to 2	1 to 2	1 to 4
Limited to 4 interpolated axes.	Option	Option	Option	Option
IEC 61131 language.	- - -	Option	Option	Option
HD graphics.	Option	Option	Standard	Standard
Conversational IIP.	Option	Standard	Standard	Standard
Dual-purpose machines (T-M).	- - -	- - -	Option	Standard
"C" axis.	Option	Standard	Standard	Standard
Dynamic RTCP.	- - -	- - -	Option	Standard
HSSA machining system.	Option	Standard	Standard	Standard
Probing canned cycles.	Option	Standard	Standard	Standard
Tandem axes.	- - -	Option	Standard	Standard
Axis and spindle synchronization.	- - -	Option	Option	Standard
Tangential control.	- - -	- - -	Option	Standard
Volumetric compensation (up to 10 m^3).	- - -	- - -	Option	Option
Volumetric compensation (more than 10 m^3).	- - -	- - -	Option	Option
ProGTL3 language.	- - -	- - -	- - -	- - -
Part-program translator.	- - -	- - -	- - -	- - -
Conversational user cycles.	Option	Option	Option	Option
Kinematics calibration.	- - -	- - -	- - -	- - -

VERSION HISTORY - CNC 8060

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

Ref. 1402

Software V01.00
First version.

Ref. 1512

Software V01.40	
The software option "probing cycles" is now called "Probe" and it also affects the probing G functions (G100/G101/G102/G103/G104).	• Function: G100, G101, G102, G103 y G104.
First and last routines in tool calibration cycles. • Sub_Probe_Tool_Begin.fst • Sub_Probe_Tool_End.fst	• ISO probing cycles. • Editor cycles
First and last routines in part measuring cycles. • Sub_Probe_Piece_Begin.fst • Sub_Probe_Piece_End.fst	• ISO probing cycles. • Editor cycles

Ref. 1603

Software V01.45	
Cycles of the editor. Programming of M functions to be executed before and after the cycle.	• Probing cycles of the editor.
Safe probing cycles.	

FAGOR

CNC 8060
CNC 8065

(REF: 1603)

VERSION HISTORY - CNC8065

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

Ref. 1103

First version.

Ref. 1309

Software V04.27	
Executing G100/G103/G104 updates variables (V.)A.MEAS.xn, (V.)A.MEASOF.xn and (V.)A.MEASOK.xn of all the axes of the channel.	• Variables: (V.)A.MEAS.xn (V.)A.MEASOF.xn (V.)A.MEASOK.xn
The variable keeps its value after a reset.	• Variable: (V.)A.MEASOK.xn
The CNC lets make a measurement (G100/G103/G104) on any axis of the channel even when function G101 is active.	• Function: G101.
Measuring an axis does not change the G101 of the rest of the axes.	• Function: G101.
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).	• Function: G101.
The CNC lets program any axis of the channel in a G102 block even if it does not have a measuring offset included (G101).	• Function: G102.

Ref. 1512

Software V05.40	
The software option "probing cycles" is now called "Probe" and it also affects the probing G functions (G100/G101/G102/G103/G104).	• Function: G100, G101, G102, G103 y G104.
First and last routines in tool calibration cycles. • Sub_Probe_Tool_Begin.fst • Sub_Probe_Tool_End.fst	• ISO probing cycles. • Editor cycles
First and last routines in part measuring cycles. • Sub_Probe_Piece_Begin.fst • Sub_Probe_Piece_End.fst	• ISO probing cycles. • Editor cycles

Ref. 1603

Software V05.45	
Cycles of the editor. Programming of M functions to be executed before and after the cycle.	• Probing cycles of the editor.
Safe probing cycles.	

SAFETY CONDITIONS

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

Before start-up, verify that the machine that integrates this CNC meets the 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 manipulate the inside 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 REPAIR

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

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

PRECAUTIONS AGAINST PERSONAL DAMAGE

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 electrical overloads.	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. Before connecting the inputs and outputs of this unit, make sure that all the grounding connections are properly made. In order to avoid electrical shock, before turning the unit on verify that the ground connection is properly made.
Do not work in humid environments.	In order to avoid electrical discharges, always work with a relative humidity (non-condensing).
Do not work in explosive environments.	In order to avoid risks or damages, do no work in explosive environments.

PRECAUTIONS AGAINST PRODUCT DAMAGE

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

Install the unit in the right place.
It is recommended, whenever possible, to install the CNC away from coolants, chemical product, blows, etc. that could damage it.
This unit complies with the European directives on electromagnetic compatibility. Nevertheless, it is recommended to keep it away from sources of electromagnetic disturbance such as:

- Powerful loads connected to the same AC power line as this equipment.
- Nearby portable transmitters (radio-telephones, Ham radio transmitters).
- Nearby radio/TV transmitters.
- Nearby arc welding machines.
- Nearby High Voltage power lines.

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

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

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

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

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

Ambient conditions.
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.

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

SAFETY SYMBOLS

Symbols that may appear on the manual.

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

Warning 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).

WARRANTY TERMS

INITIAL WARRANTY

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

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

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

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

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

EXCLUDING CLAUSES

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

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

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

WARRANTY ON REPAIRS

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

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

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

For sold parts the warranty is 12 moths length.

SERVICE CONTRACTS

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

MATERIAL RETURNING TERMS

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

1. Get a cardboard box whose three inside dimensions are at least 15 cm (6 inches) larger than those of the unit. The cardboard being used to make the box must have a resistance of 170 Kg (375 lb.).
2. Attach a label indicating the owner of the unit, person to contact, type of unit and serial number. In case of malfunction also indicate symptom and a brief description of the problem.
3. Wrap the unit in a polyethylene roll or similar material to protect it. When sending a central unit with monitor, protect especially the screen.
4. Pad the unit inside the cardboard box with poly-utherane foam on all sides.
5. Seal the cardboard box with packing tape or industrial staples.

CNC MAINTENANCE

CLEANING

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

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

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

PRECAUTIONS BEFORE CLEANING THE UNIT

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

- Do not handle the connectors with the unit 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 manipulate the inside of this unit.

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.

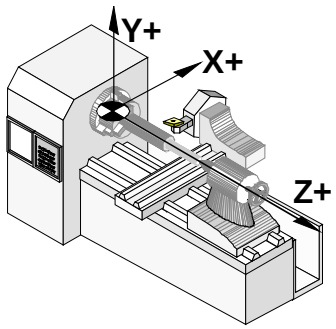
Knowing which is the active probe.

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.

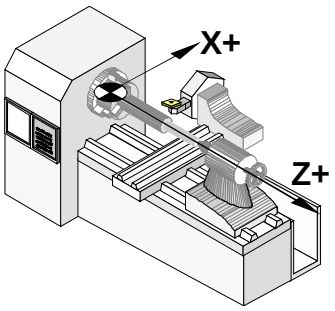
Variable.	Meaning.
(V.)[ch].G.ACTIVPROBE	This variable indicates which one is the active probe in channel n.

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.

1.

PREVIOUS NOTIONS ABOUT THE PROBE.

Safe probing cycles.

FAGOR

CNC 8060
CNC 8065

(REF: 1603)

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.

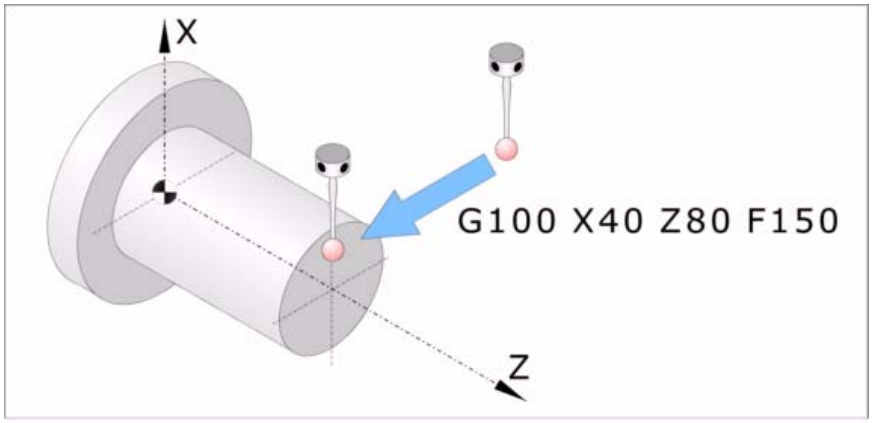
```
G100 X45.23 Z23.45
G100 Z50 F100
G103 X2.6 Z3 F20
G103 Z1 F20
```

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.

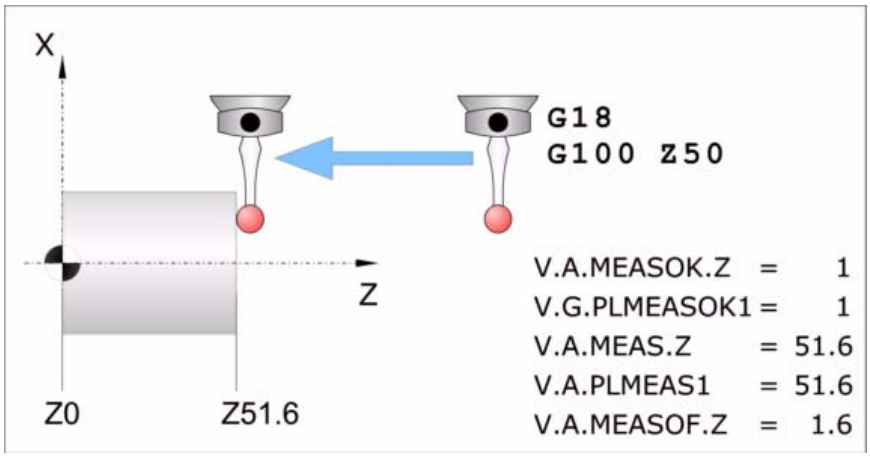
Functions G100 and G103 are not modal. After executing one of these functions, the CNC restores the function G0, G1, G2 ó G3, G33 or G63 that was active.

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.

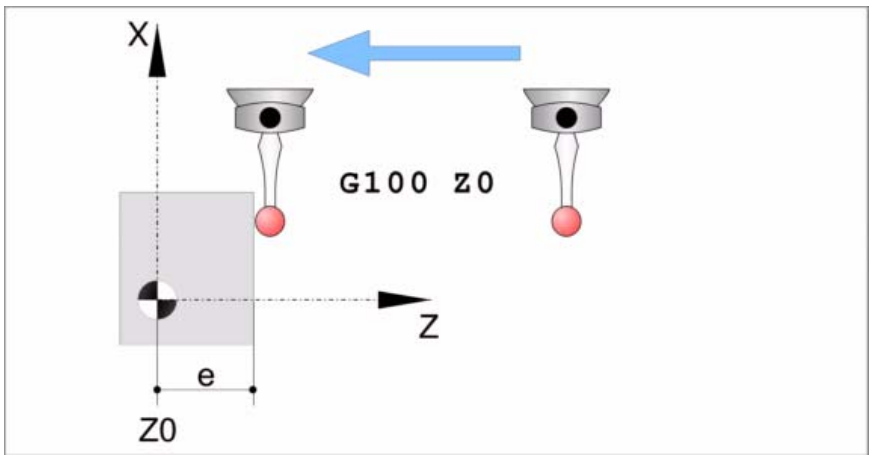
Mnemoni.	Variable.
V.G.MEASOK	The probe has made contact (G100) or stopped making contact (G103). • The variable takes the value of ·1· if the probe has made contact (G100) or has stopped making contact (G103). • The variables takes the value of ·0· if the probe reaches the programmed coordinate.
V.A.MEASOK.xn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

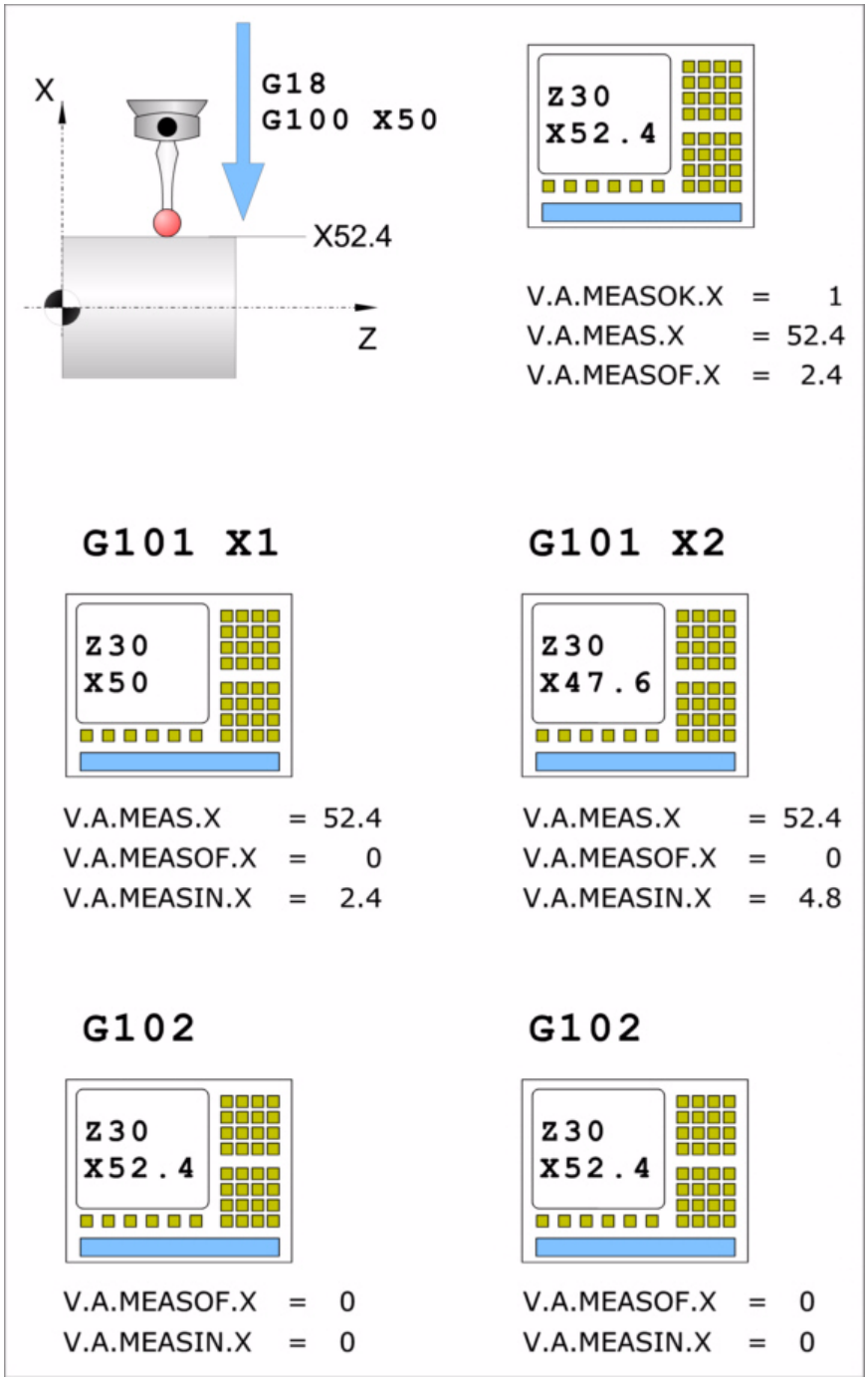
```
G101 X..C
```

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

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

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

Programming ISO coded cycles.

ISO coded cycles are defined with the #PROBE instruction followed by the number of the cycle to be executed and the call parameters.

#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.

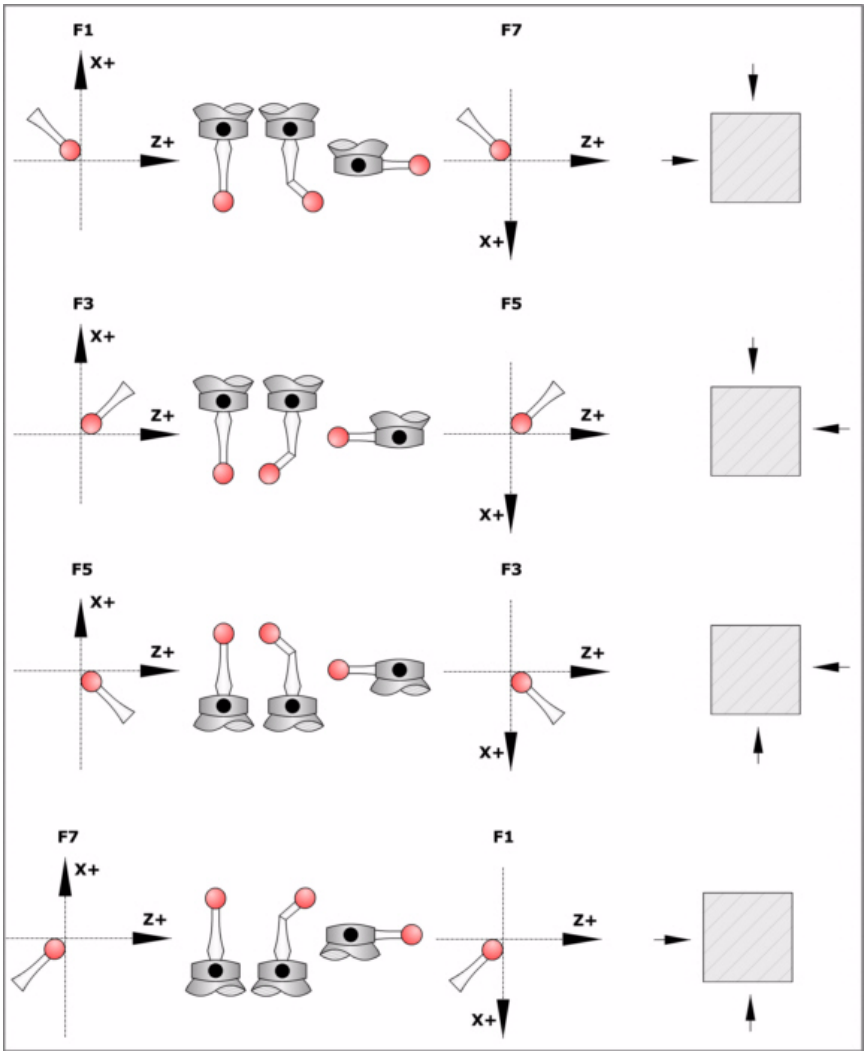
This cycle may be used to calibrate the dimensions of a tool or a touch probe. Once the cycle has concluded, it updates the dimensions in the tool table and initializes the tool wears to 0 (zero).

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.

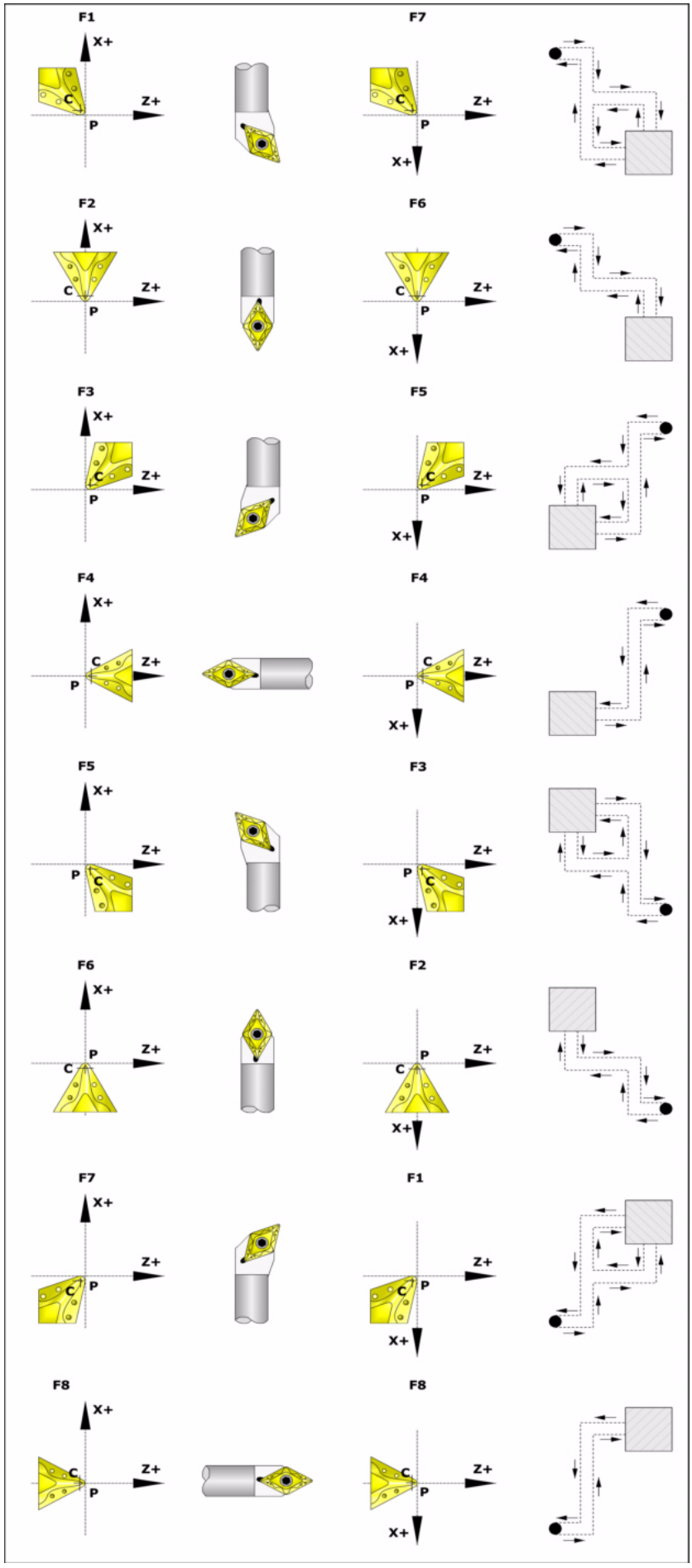
·X U Y V Z W· Tabletop probe position.

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.

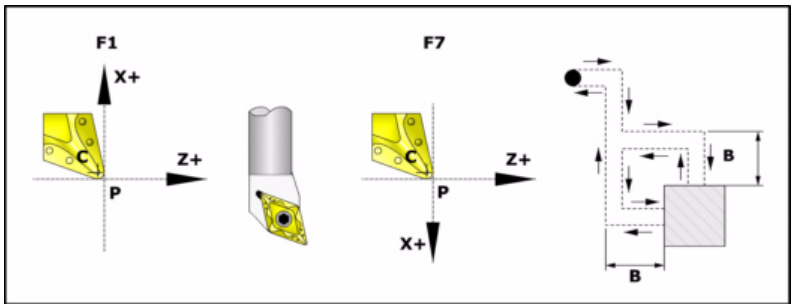
3.1.2 Basic operation.



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.

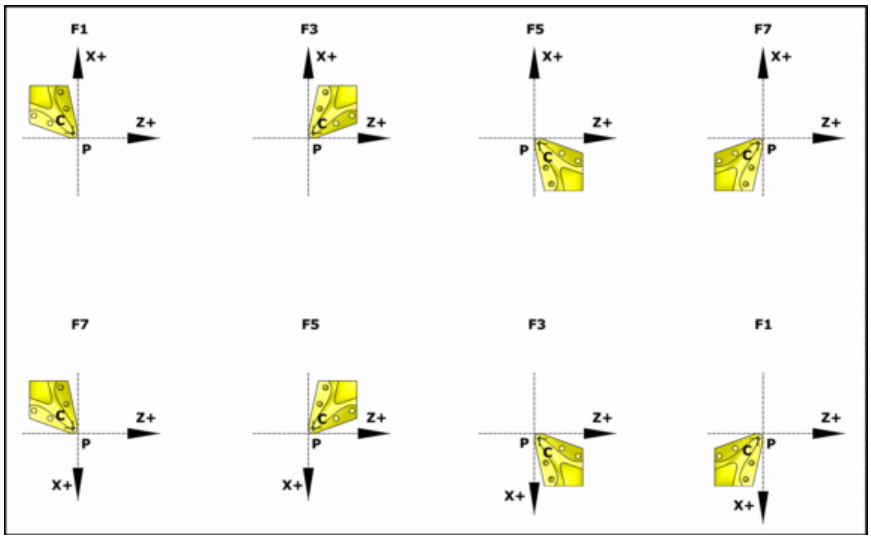
3.2 #PROBE 2. Tabletop probe calibration

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.

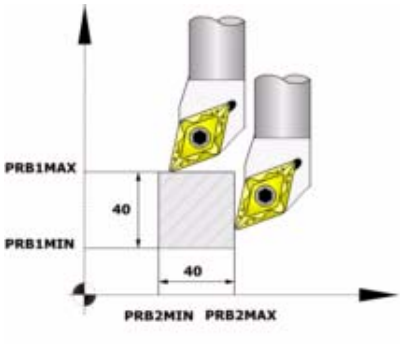
PRB2MAX Maximum probe coordinate along the second axis of the channel.

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.

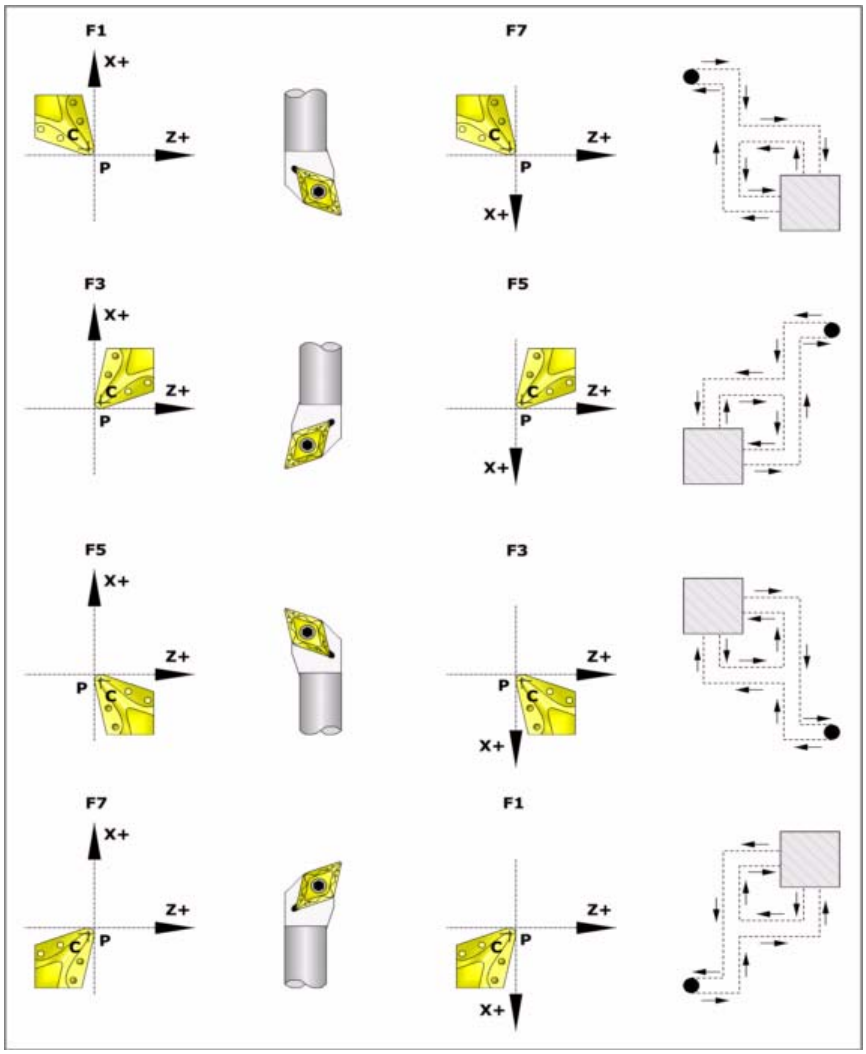
·X U Y V Z W· Tabletop probe position.

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.

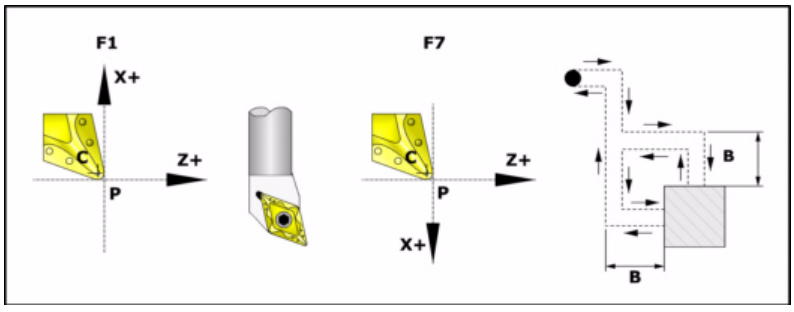
3.2.2 Basic operation.



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.

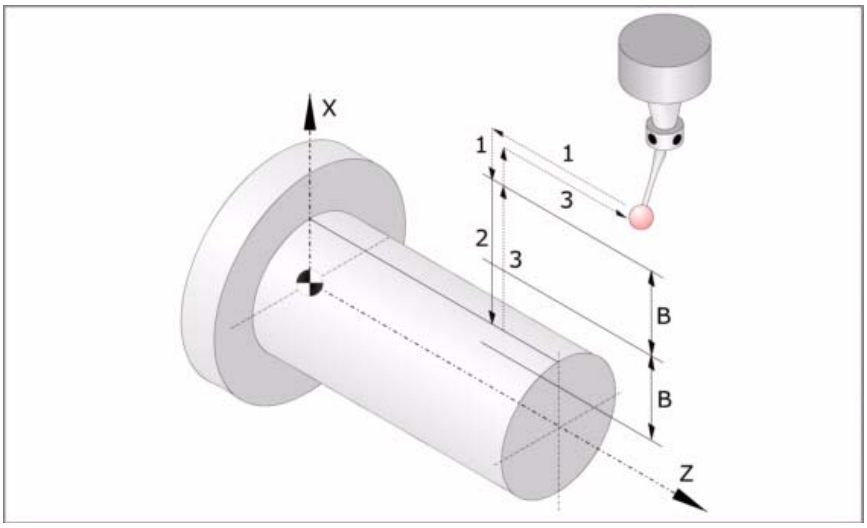
Tool whose wear is to be corrected, which will be the tool used to machine the surface.

·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.

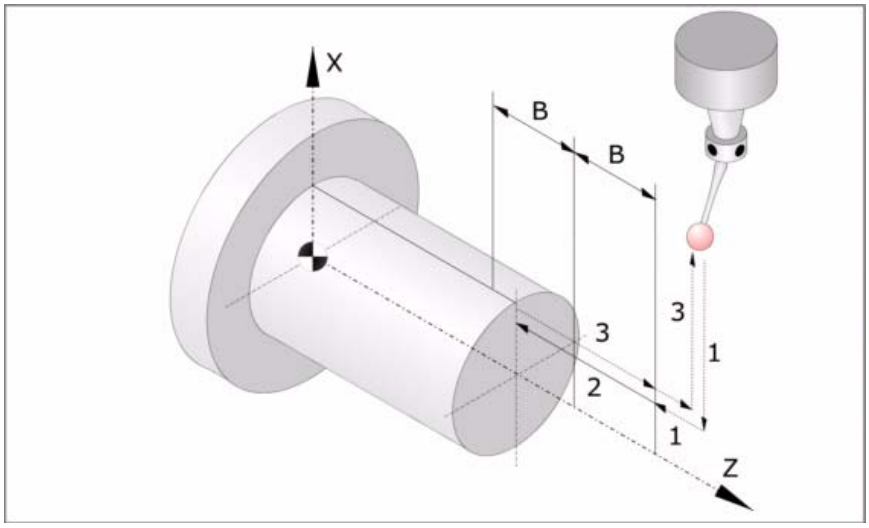
Tool whose wear is to be corrected, which will be the tool used to machine the surface.

·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
```

FAGOR

CNC 8060
CNC 8065

(REF: 1603)

CANNED CYCLES. CYCLE EDITOR.

4

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

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

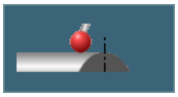
Programming the cycles of the editor.

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

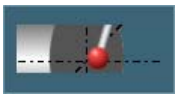
The cycles of the editor are accessed with the following softkeys:

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.

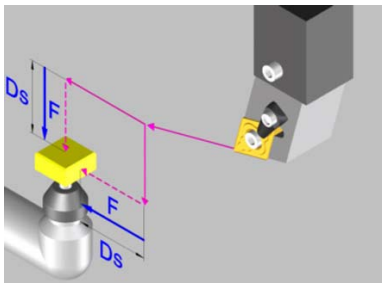
The Teach-in mode is activated from the horizontal softkey menu. When the Teach-in mode is active, the bottom of the screen shows a window with the axes of the channel.

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.

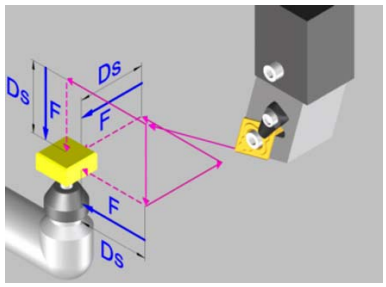
To assign a value to a data, select it with the cursor (focus on it) and press the [RECALL] key. The data is taken from the channel where the editing-simulation mode is active.

- 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.



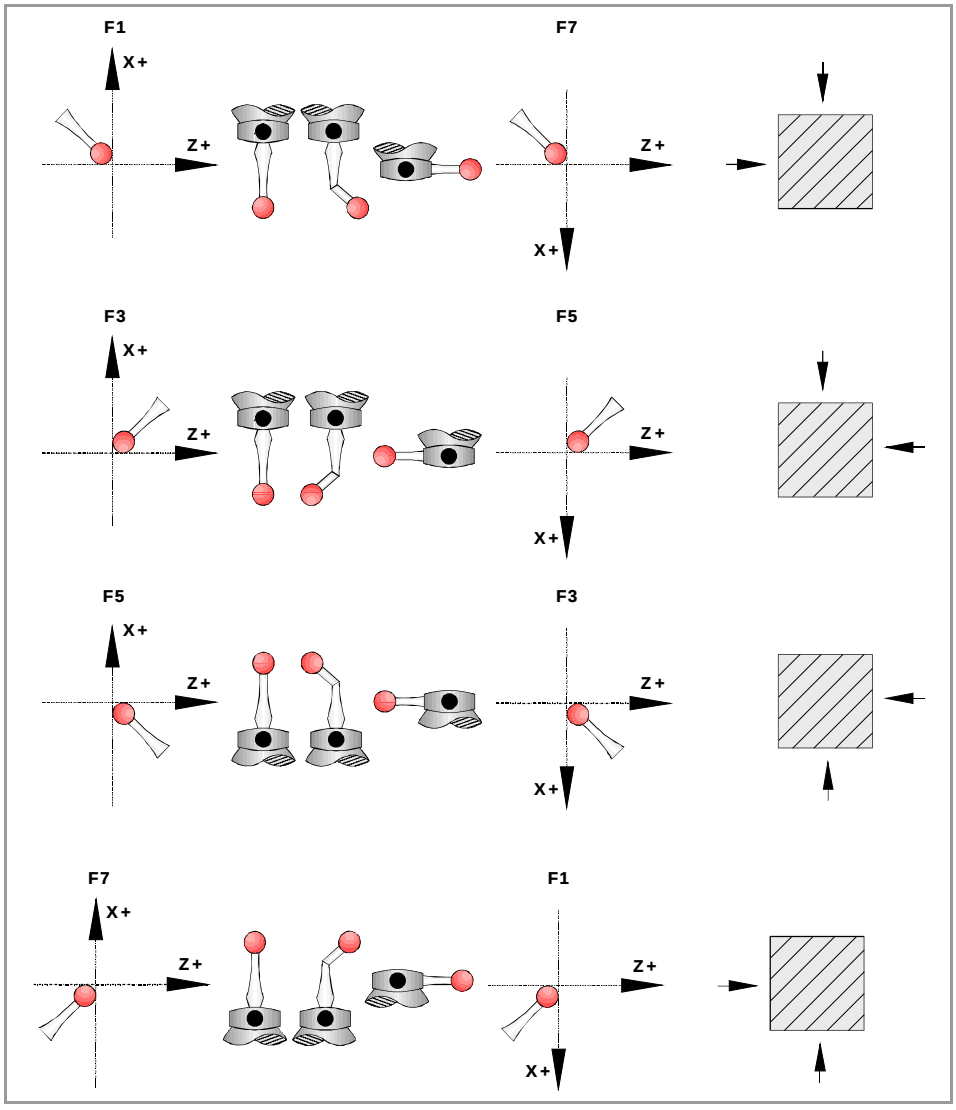
Geometrical configuration of "trihedron" type axes.

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.

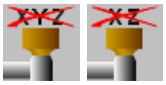
Distance with respect to the point to touch, to which the tool approaches in G00 before making the probing movement. When calling the cycle, the tool must be located, with respect to the point to be measured, at a greater distance than this value

·F· Probing feedrate.

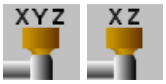
This parameter sets the probing feedrate. The rest of the movements will be carried out in G00.

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.

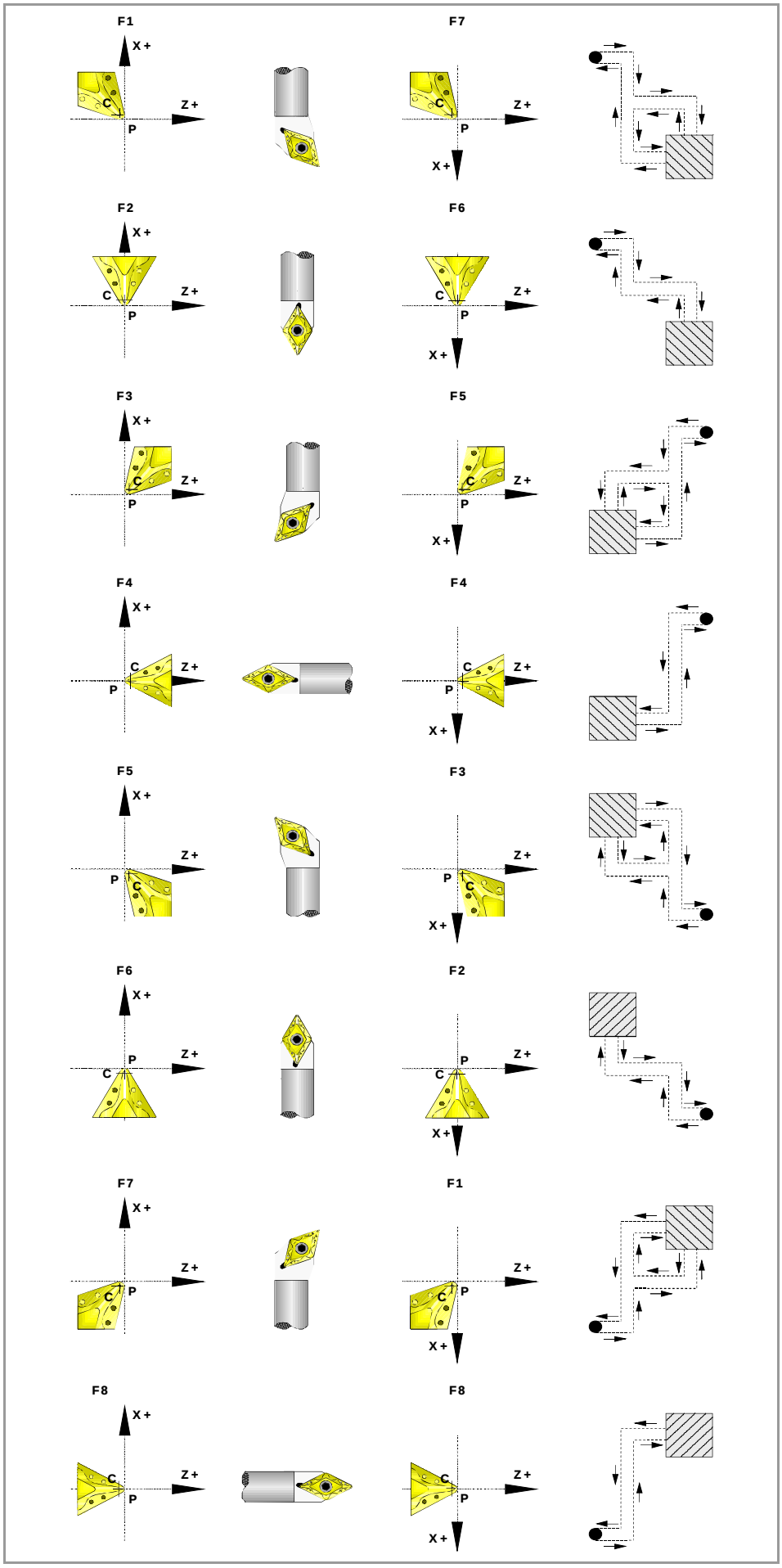
The cycle allows executing up to 4 M functions before the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

·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.



FAGOR

CNC 8060
CNC 8065

(REF: 1603)

1 The cycle selects the programmed tool.

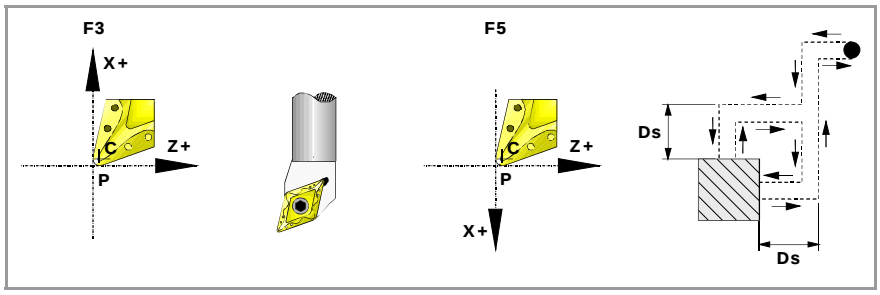
2 The CNC runs the subroutine Sub_Probe_Tool_Begin.fst, defined by the OEM.

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.

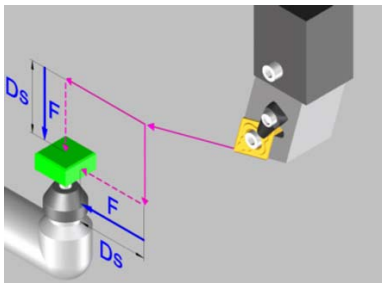
Rapid probe movement (G00) from the approach corner to the cycle calling point.

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

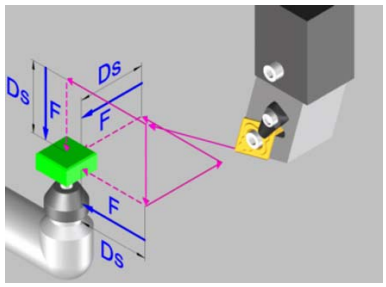
7 The cycle executes the "M-after" functions.

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.



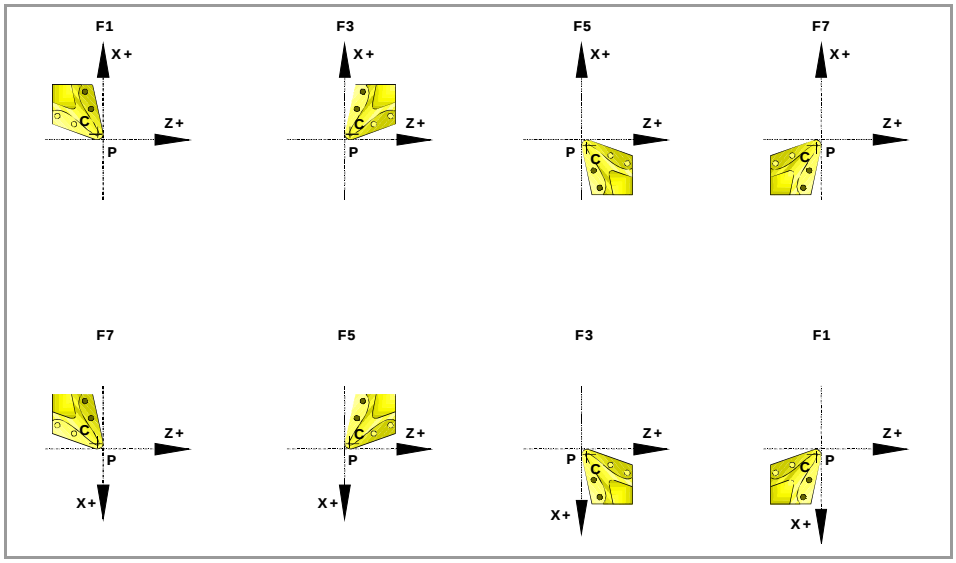
Geometrical configuration of "trihedron" type axes.

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.

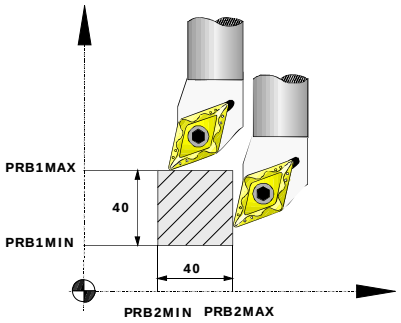
Once the values of these parameters and the probe dimensions are known, the user must calculate the coordinates of the other sides and update the following general machine parameters.

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.

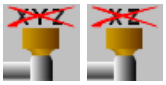
Distance with respect to the point to touch, to which the tool approaches in G00 before making the probing movement. When calling the cycle, the tool must be located, with respect to the point to be measured, at a greater distance than this value

·F· Probing feedrate.

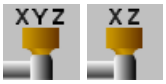
This parameter sets the probing feedrate. The rest of the movements will be carried out in G00.

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.

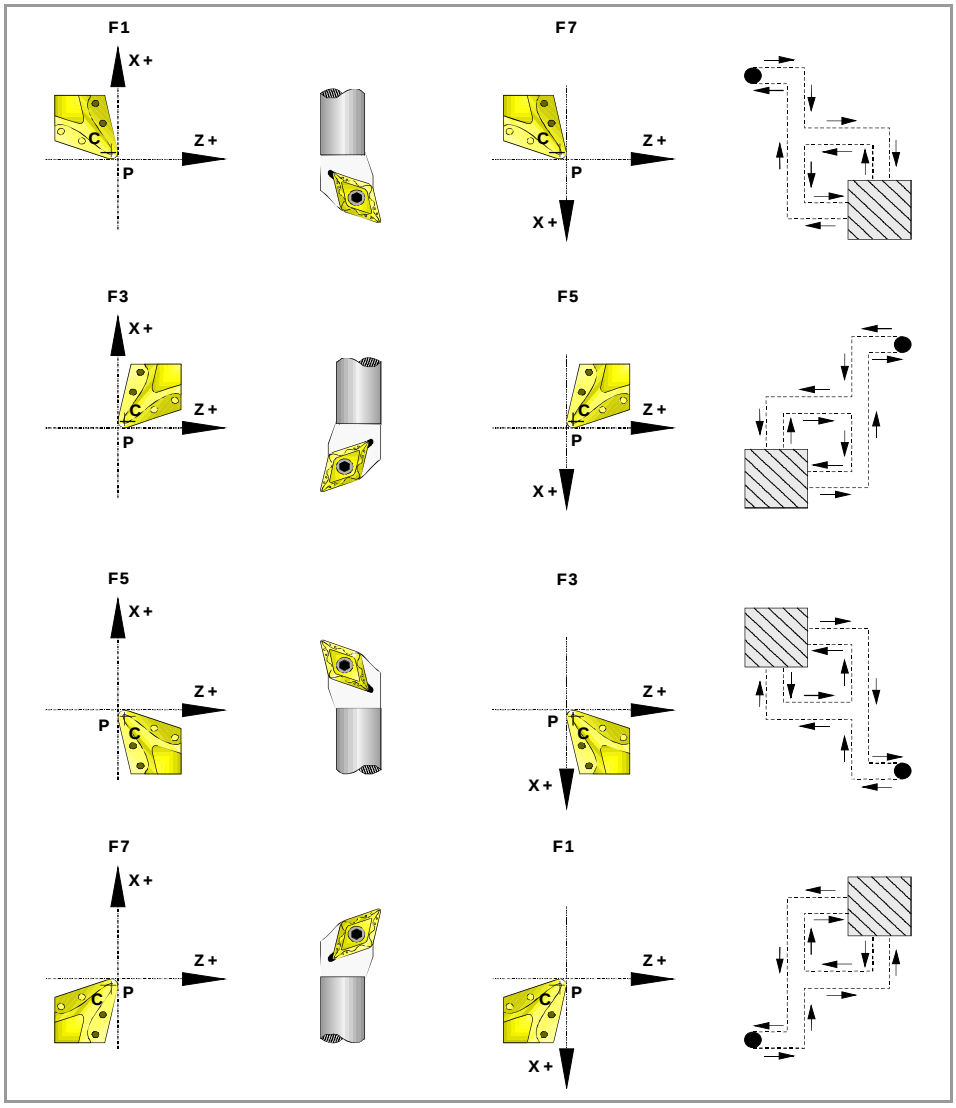
The cycle allows executing up to 4 M functions before the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

·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.

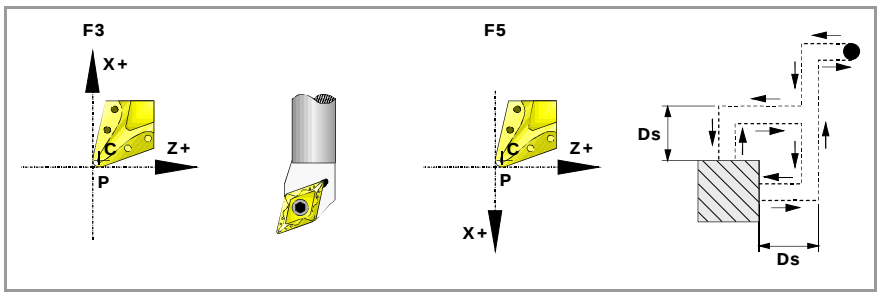
2 The CNC runs the subroutine Sub_Probe_Tool_Begin.fst, defined by the OEM.

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.

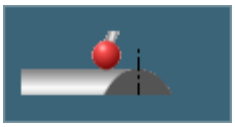
Rapid probe movement (G00) from the approach corner to the cycle calling point.

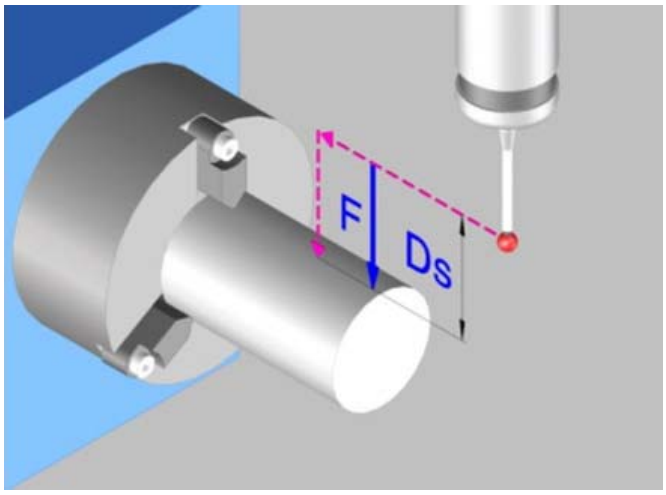
This withdrawal movement is made in two stages. It first moves along the X axis and then along the Z axis.

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.

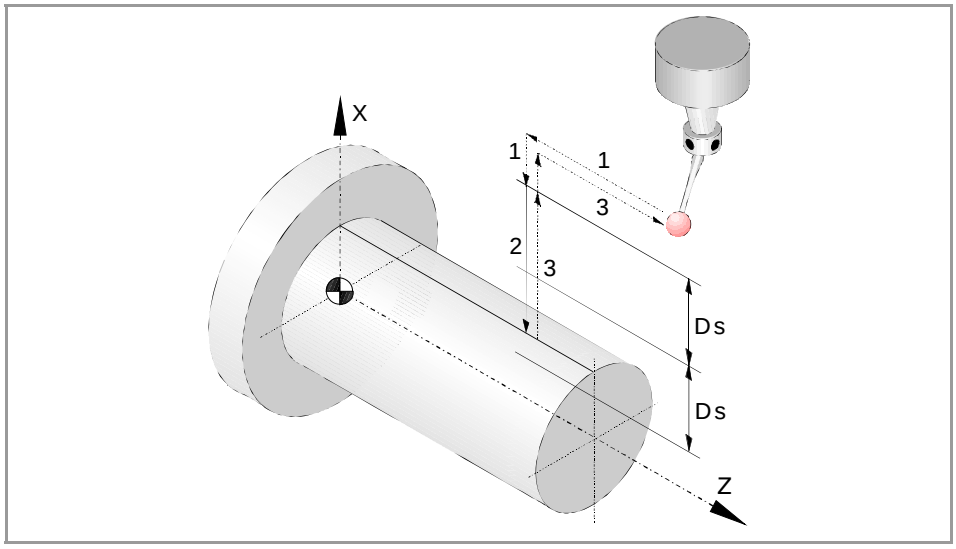
·M after· M functions to be executed after the cycle.

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.

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.

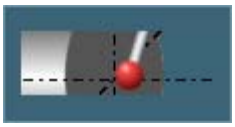
Rapid probe movement (G00) from the probing point to the cycle calling point.

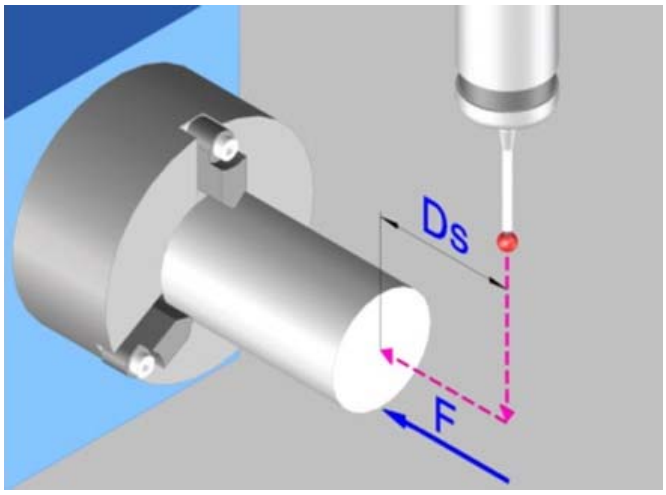
This withdrawal movement is made in two stages. It first moves along the X axis and then along the Z axis.

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.

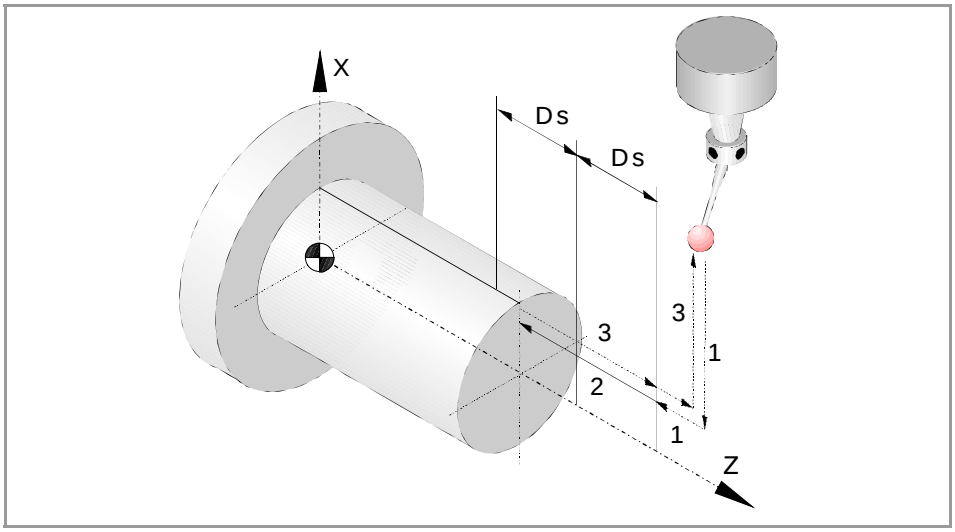
·M after· M functions to be executed after the cycle.

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.

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