

**CNC**

**8060**

**8065**

Probing (-M- 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.

---



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

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

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

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

The content of this manual and its validity for the product described here has been verified. Even so, involuntary errors are possible, 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

|                                    |    |
|------------------------------------|----|
| About the product - CNC 8060 ..... | 5  |
| About the product - CNC 8065 ..... | 7  |
| Version history - CNC 8060 .....   | 11 |
| Version history - CNC 8065 .....   | 13 |
| Condiciones de seguridad .....     | 15 |
| Warranty terms .....               | 19 |
| Material returning terms.....      | 21 |
| CNC maintenance .....              | 23 |

**CHAPTER 1 PREVIOUS NOTIONS ABOUT THE PROBE.**

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

**CHAPTER 2 PROBING.**

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

**CHAPTER 3 CANNED CYCLES. ISO CODED PROGRAMMING.**

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



CNC 8060  
CNC 8065

(REF: 1603)

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



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<br>Ethernet                               |          |           |          |          |
| Integrated PLC.<br>PLC execution time.<br>Digital inputs / Digital outputs.<br>Marks / Registers.<br>Timers / Counters.<br>Symbols. | < 1ms/K<br>1024 / 1024<br>8192 / 1024<br>512 / 256<br>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<br>8065<br>8060 | 8070<br>8065<br>8060 | 8070<br>8065<br>--- | 8070<br>8065<br>8060 | 8070<br>8065<br>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 (**)               |

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



CNC 8060  
CNC 8065

(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.

|                                      | 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      |
| Maximum 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 |



CNC 8060  
CNC 8065

(REF: 1603)

# 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<br>Ethernet              |        |
| Integrated PLC.                   |  |        |
| PLC execution time.               | < 1ms/K  |        |
| Digital inputs / Digital outputs. | 1024 / 1024                                    |        |
| Marks / Registers.                | 8192 / 1024                                    |        |
| Timers / Counters.                | 512 / 256                                      |        |
| Symbols.                          | Unlimited                                      |        |
| Block processing time.            | < 1 ms   |        |

| Remote modules.                        | RIOW                 | RIO5                 | RIO70               | RIOR                 | RCS-S                |
|--|----------------------|----------------------|---------------------|----------------------|----------------------|
| Valid for CNC.                         | 8070<br>8065<br>8060 | 8070<br>8065<br>8060 | 8070<br>8065<br>--- | 8070<br>8065<br>8060 | 8070<br>8065<br>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 (**)               |

(\*) 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.<br>INI configuration files.<br>FGUIM visual configuration tool.<br>Visual Basic®, Visual C++®, etc.<br>Internal databases in Microsoft® Access.<br>OPC compatible interface |



CNC 8060  
CNC 8065

(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   |
| Maximum number of axes and spindles.                    | 7        | 10       | 16           | 32       |
| Number of tool magazines.                               | 1        | 1        | 1 to 2       | 1 to 4   |
| Limited to 4 interpolated axes.                         | Option   | Option   | Option       | Option   |
| 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 <sup>3</sup> ).     | ---      | ---      | Option       | Option   |
| Volumetric compensation (more than 10 m <sup>3</sup> ). | ---      | ---      | 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   |



CNC 8060  
CNC 8065

(REF: 1603)



**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 m³).         | ---    | ---      | Option       | Option   |
| Volumetric compensation (more than 10 m³).     | ---    | ---      | Option       | Option   |
| ProGTL3 language.                              | ---    | ---      | ---          | ---      |
| Part-program translator.                       | ---    | ---      | ---          | ---      |
| Conversational user cycles.                    | Option | Option   | Option       | Option   |
| Kinematics calibration.                        | ---    | ---      | ---          | ---      |



**CNC 8060  
CNC 8065**

(REF: 1603)



# VERSION HISTORY - CNC 8060

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

## Ref. 1402

|                        |
|------------------------|
| <b>Software V01.00</b> |
| First version.         |

## Ref. 1512

|  |  |
|--|--|
| <b>Software V01.40</b>   |  |
| The software option "probing cycles" is now called "Probe" and it also affects the probing G functions (G100/G101/G102/G103/G104).                             | <ul style="list-style-type: none"> <li>Function: G100, G101, G102, G103 y G104.</li> </ul>   |
| First and last routines in tool calibration cycles. <ul style="list-style-type: none"> <li>Sub_Probe_Tool_Begin.fst</li> <li>Sub_Probe_Tool_End.fst</li> </ul> | <ul style="list-style-type: none"> <li>ISO probing cycles.</li> <li>Editor cycles</li> </ul> |
| First and last routines in part measuring cycles. <ul style="list-style-type: none"> <li>Sub_Probe_Piece_Begin.fst</li> <li>Sub_Probe_Piece_End.fst</li> </ul> | <ul style="list-style-type: none"> <li>ISO probing cycles.</li> <li>Editor cycles</li> </ul> |

## Ref. 1603

|   |   |
|---|---|
| <b>Software V01.45</b>  |   |
| ISO coded cycles. The #PROBE 9 cycle (circular boss measuring cycle) executes the probing movement between points, passing through the center of the part. This operation lets avoid collisions against clamps or fixtures.   | <ul style="list-style-type: none"> <li>#PROBE 9. Circular boss measuring.</li> </ul>  |
| ISO coded cycles. The #PROBE 10 cycle (rectangular part centering) has been changed as follows. <ul style="list-style-type: none"> <li>The cycle allows the part to be centered on a single axis.</li> <li>The cycle executes the probe movement between points, passing through the center of the part. This operation lets avoid collisions against clamps or fixtures.</li> </ul>  | <ul style="list-style-type: none"> <li>#PROBE 10. Rectangular part centering.</li> </ul>  |
| ISO coded cycles. The #PROBE 11 cycle (circular part centering) has been changed as follows. <ul style="list-style-type: none"> <li>The cycle allows the part to be centered on a single axis.</li> <li>The cycle executes the probe movement between points, passing through the center of the part. This operation lets avoid collisions against clamps or fixtures.</li> </ul>   | <ul style="list-style-type: none"> <li>#PROBE 11. Circular part centering.</li> </ul>   |
| Cycles of the editor. Programming of M functions to be executed before and after the cycle.   | <ul style="list-style-type: none"> <li>Measurement probe calibration</li> <li>Tabletop probe calibration.</li> <li>Surfacing measuring.</li> <li>Outside corner measuring.</li> <li>Inside corner measuring.</li> <li>Measuring the angle with the abscissa axis.</li> <li>Outside corner and angle measuring.</li> <li>Hole measuring.</li> <li>Circular boss measuring.</li> <li>Rectangular or circular part centering.</li> </ul> |
| Cycles of the editor. Save the result of the measurement in a zero offset G159.   | <ul style="list-style-type: none"> <li>Surfacing measuring.</li> <li>Outside corner measuring.</li> <li>Inside corner measuring.</li> <li>Outside corner and angle measuring.</li> <li>Hole measuring.</li> <li>Circular boss measuring.</li> <li>Rectangular or circular part centering.</li> </ul>  |
| Cycles of the editor. The part centering cycle has been changed as follows. <ul style="list-style-type: none"> <li>The cycle allows centering rectangular and circular pockets.</li> <li>The cycle allows centering the part/pocket on a single axis.</li> <li>When the cycle centers the part/pocket on both axes, the cycle lets select whether to touch once or twice on the same side.</li> <li>The cycle executes the probe movement between points, passing through the center of the part. This operation lets avoid collisions against clamps or fixtures.</li> </ul> | <ul style="list-style-type: none"> <li>Rectangular or circular part centering.</li> </ul>   |
| Cycles of the editor. The boss measuring cycle executes the probe movement between points, passing through the center of the part. This operation lets avoid collisions against clamps or fixtures.   | <ul style="list-style-type: none"> <li>Circular boss measuring.</li> </ul>  |



CNC 8060  
CNC 8065

(REF: 1603)

**Software V01.45**

Cycles of the editor. In the surface measurement cycle, parameters Xm, Ym and Zm were required, but are now optional. When not programming the theoretical coordinate of the probing point on some axis, the cycle shows an icon to define the moving direction of that axis.

- Surfacing measuring.

Safe probing cycles.



CNC 8060  
CNC 8065

(REF: 1603)

# VERSION HISTORY - CNC 8065

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

## Ref. 1103

|                |
|----------------|
| First version. |
|----------------|

## Ref. 1309

| Software V04.27  |  |
|--|--|
| #PROBE 1. Tool calibration (dimensions and wear).<br>Parameter ·N· admits negative values.   | <ul style="list-style-type: none"> <li>Statement: #PROBE 1.</li> </ul>   |
| #PROBE 2. Probe calibration.<br>The cycle returns the value of PROBEDELAY of all the axes in arithmetic parameter P299.                                      | <ul style="list-style-type: none"> <li>Statement: #PROBE 2.</li> </ul>   |
| #PROBE 12. Tabletop probe calibration.<br>The cycle returns the value of PROBEDELAY of all the axes and the spindle in arithmetic parameter P294.            | <ul style="list-style-type: none"> <li>Statement: #PROBE 12.</li> </ul>  |
| Cycles of the editor. Probe calibration.<br>The cycle returns the value of PROBEDELAY of all the axes in arithmetic parameter P299.                          | <ul style="list-style-type: none"> <li>Probe calibration.</li> </ul>   |
| Cycles of the editor. Tabletop probe calibration.<br>The cycle returns the value of PROBEDELAY of all the axes and the spindle in arithmetic parameter P294. | <ul style="list-style-type: none"> <li>Tabletop probe calibration.</li> </ul>                                      |
| 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.                                | <ul style="list-style-type: none"> <li>Variables: (V.)A.MEAS.xn<br/>(V.)A.MEASOF.xn<br/>(V.)A.MEASOK.xn</li> </ul> |
| The variable keeps its value after a reset.  | <ul style="list-style-type: none"> <li>Variable: (V.)A.MEASOK.xn</li> </ul>  |
| The CNC lets make a measurement (G100/G103/G104) on any axis of the channel even when function G101 is active.   | <ul style="list-style-type: none"> <li>Function: G101.</li> </ul>  |
| Measuring an axis does not change the G101 of the rest of the axes.  | <ul style="list-style-type: none"> <li>Function: G101.</li> </ul>  |
| 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).                  | <ul style="list-style-type: none"> <li>Function: G101.</li> </ul>  |
| The CNC lets program any axis of the channel in a G102 block even if it does not have a measuring offset included (G101).                                    | <ul style="list-style-type: none"> <li>Function: G102.</li> </ul>  |

## 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).                             | <ul style="list-style-type: none"> <li>Function: G100, G101, G102, G103 y G104.</li> </ul>   |
| First and last routines in tool calibration cycles. <ul style="list-style-type: none"> <li>Sub_Probe_Tool_Begin.fst</li> <li>Sub_Probe_Tool_End.fst</li> </ul> | <ul style="list-style-type: none"> <li>ISO probing cycles.</li> <li>Editor cycles</li> </ul> |
| First and last routines in part measuring cycles. <ul style="list-style-type: none"> <li>Sub_Probe_Piece_Begin.fst</li> <li>Sub_Probe_Piece_End.fst</li> </ul> | <ul style="list-style-type: none"> <li>ISO probing cycles.</li> <li>Editor cycles</li> </ul> |

## Ref. 1603

| Software V05.45  |  |
|--|--|
| ISO coded cycles. The #PROBE 9 cycle (circular boss measuring cycle) executes the probing movement between points, passing through the center of the part. This operation lets avoid collisions against clamps or fixtures.  | <ul style="list-style-type: none"> <li>#PROBE 9. Circular boss measuring.</li> </ul>     |
| ISO coded cycles. The #PROBE 10 cycle (rectangular part centering) has been changed as follows. <ul style="list-style-type: none"> <li>The cycle allows the part to be centered on a single axis.</li> <li>The cycle executes the probe movement between points, passing through the center of the part. This operation lets avoid collisions against clamps or fixtures.</li> </ul> | <ul style="list-style-type: none"> <li>#PROBE 10. Rectangular part centering.</li> </ul> |



CNC 8060  
CNC 8065

(REF: 1603)

| Software V05.45   |   |
|---|---|
| ISO coded cycles. The #PROBE 11 cycle (circular part centering) has been changed as follows. <ul style="list-style-type: none"> <li>The cycle allows the part to be centered on a single axis.</li> <li>The cycle executes the probe movement between points, passing through the center of the part. This operation lets avoid collisions against clamps or fixtures.</li> </ul>   | <ul style="list-style-type: none"> <li>#PROBE 11. Circular part centering.</li> </ul>   |
| Cycles of the editor. Programming of M functions to be executed before and after the cycle.   | <ul style="list-style-type: none"> <li>Measurement probe calibration</li> <li>Tabletop probe calibration.</li> <li>Surfacing measuring.</li> <li>Outside corner measuring.</li> <li>Inside corner measuring.</li> <li>Measuring the angle with the abscissa axis.</li> <li>Outside corner and angle measuring.</li> <li>Hole measuring.</li> <li>Circular boss measuring.</li> <li>Rectangular or circular part centering.</li> </ul> |
| Cycles of the editor. Save the result of the measurement in a zero offset G159.   | <ul style="list-style-type: none"> <li>Surfacing measuring.</li> <li>Outside corner measuring.</li> <li>Inside corner measuring.</li> <li>Outside corner and angle measuring.</li> <li>Hole measuring.</li> <li>Circular boss measuring.</li> <li>Rectangular or circular part centering.</li> </ul>  |
| Cycles of the editor. The part centering cycle has been changed as follows. <ul style="list-style-type: none"> <li>The cycle allows centering rectangular and circular pockets.</li> <li>The cycle allows centering the part/pocket on a single axis.</li> <li>When the cycle centers the part/pocket on both axes, the cycle lets select whether to touch once or twice on the same side.</li> <li>The cycle executes the probe movement between points, passing through the center of the part. This operation lets avoid collisions against clamps or fixtures.</li> </ul> | <ul style="list-style-type: none"> <li>Rectangular or circular part centering.</li> </ul>   |
| Cycles of the editor. The boss measuring cycle executes the probe movement between points, passing through the center of the part. This operation lets avoid collisions against clamps or fixtures.   | <ul style="list-style-type: none"> <li>Circular boss measuring.</li> </ul>  |
| Cycles of the editor. In the surface measurement cycle, parameters Xm, Ym and Zm were required, but are now optional. When not programming the theoretical coordinate of the probing point on some axis, the cycle shows an icon to define the moving direction of that axis.   | <ul style="list-style-type: none"> <li>Surfacing measuring.</li> </ul>  |
| Safe probing cycles.  |   |



CNC 8060  
CNC 8065

(REF: 1603)

# CONDICIONES DE SEGURIDAD

Leer las siguientes medidas de seguridad con objeto de evitar lesiones a personas y prevenir daños a este producto y a los productos conectados a él. Fagor Automation no se responsabiliza de cualquier daño físico o material derivado del incumplimiento de estas normas básicas de seguridad.



*Antes de la puesta en marcha, comprobar que la máquina donde se incorpora el CNC cumple lo especificado en la Directiva 2006/42/EC.*

## PRECAUCIONES ANTES DE LIMPIAR EL APARATO

- No manipular el interior del aparato.** Sólo personal autorizado de Fagor Automation puede manipular el interior del aparato.
- No manipular los conectores con el aparato conectado a la red eléctrica.** Antes de manipular los conectores (entradas/salidas, captación, etc) cerciorarse que el aparato no está alimentado.

## PRECAUCIONES DURANTE LAS REPARACIONES

En caso de mal funcionamiento o fallo del aparato, desconectarlo y llamar al servicio de asistencia técnica.

- No manipular el interior del aparato.** Sólo personal autorizado de Fagor Automation puede manipular el interior del aparato.
- No manipular los conectores con el aparato conectado a la red eléctrica.** Antes de manipular los conectores (entradas/salidas, captación, etc) cerciorarse que el aparato no está alimentado.

## PRECAUCIONES ANTE DAÑOS A PERSONAS

- Interconexión de módulos.** Utilizar los cables de unión proporcionados con el aparato.
- Utilizar cables apropiados.** Para evitar riesgos, utilizar sólo cables y fibra Sercos recomendada para este aparato.  
Para prevenir riesgos de choque eléctrico en la unidad central, utilizar el conector apropiado (el suministrado por Fagor); usar cable de alimentación de tres conductores (uno de ellos de tierra).
- Evitar sobrecargas eléctricas.** Para evitar descargas eléctricas y riesgos de incendio, no aplicar tensión eléctrica fuera del rango indicado.
- Conexión a tierra.** Con objeto de evitar descargas eléctricas, conectar las bornas de tierra de todos los módulos al punto central de tierras. Asimismo, antes de efectuar la conexión de las entradas y salidas de este producto asegurarse que la conexión a tierras está efectuada.  
Con objeto de evitar descargas eléctricas comprobar, antes de encender el aparato, que se ha efectuado la conexión de tierras.
- No trabajar en ambientes húmedos.** Para evitar descargas eléctricas, trabajar siempre en ambientes con humedad relativa dentro del rango 10%-90% sin condensación.
- No trabajar en ambientes explosivos.** Con objeto de evitar riesgos, lesiones o daños, no trabajar en ambientes explosivos.



CNC 8060  
CNC 8065

(REF: 1603)

## PRECAUCIONES ANTE DAÑOS AL PRODUCTO

|  |   |
|--|---|
| <b>Ambiente de trabajo.</b>                              | Este aparato está preparado para su uso en ambientes industriales cumpliendo las directivas y normas en vigor en la Comunidad Económica Europea.<br>Fagor Automation no se responsabiliza de los daños que pudiera sufrir o provocar el CNC si se monta en otro tipo de condiciones (ambientes residenciales, domésticos, etc).   |
| <b>Instalar el aparato en el lugar apropiado.</b>        | Se recomienda que, siempre que sea posible, la instalación del control numérico se realice alejada de líquidos refrigerantes, productos químicos, golpes, etc que pudieran dañarlo.<br>El aparato cumple las directivas europeas de compatibilidad electromagnética. No obstante, es aconsejable mantenerlo apartado de fuentes de perturbación electromagnética, como pueden ser:<br>Cargas potentes conectadas a la misma red que el equipo.<br>Transmisores portátiles cercanos (radioteléfonos, emisores de radio aficionados).<br>Transmisores de radio/TV cercanos.<br>Máquinas de soldadura por arco cercanas.<br>Líneas de alta tensión próximas. |
| <b>Envoltentes.</b>                                      | El fabricante es responsable de garantizar que la envoltente en que se ha montado el equipo cumple todas las directivas al uso en la Comunidad Económica Europea.   |
| <b>Evitar interferencias provenientes de la máquina.</b> | La máquina debe tener desacoplados todos los elementos que generan interferencias (bobinas de los relés, contactores, motores, etc).  |
| <b>Utilizar la fuente de alimentación apropiada.</b>     | Para la alimentación del teclado, panel de mando y módulos remotos, utilizar una fuente de alimentación exterior estabilizada de 24 V DC.   |
| <b>Conexión a tierra de la fuente de alimentación.</b>   | El punto de cero voltios de la fuente de alimentación externa deberá conectarse al punto principal de tierra de la máquina.   |
| <b>Conexión de las entradas y salidas analógicas.</b>    | Realizar la conexión mediante cables apantallados, conectando todas las mallas al terminal correspondiente.   |
| <b>Condiciones medioambientales.</b>                     | Mantener el CNC dentro del rango de temperaturas recomendado, tanto en régimen de funcionamiento como de no-funcionamiento. Ver el capítulo correspondiente en el manual de hardware.   |
| <b>Habitáculo de la unidad central.</b>                  | Para mantener las condiciones ambientales adecuadas en el habitáculo de la unidad central, éste debe cumplir los requisitos indicados por Fagor. Ver el capítulo correspondiente en el manual de hardware.  |
| <b>Dispositivo de seccionamiento de la alimentación.</b> | El dispositivo de seccionamiento de la alimentación ha de situarse en un lugar fácilmente accesible y a una distancia del suelo comprendida entre 0,7 y 1,7 metros (2,3 y 5,6 pies).  |

## SÍMBOLOS DE SEGURIDAD

### Símbolos que pueden aparecer en el manual.



*Símbolo de peligro o prohibición.*

*Este símbolo indica acciones u operaciones que pueden provocar daños a personas o aparatos.*



*Símbolo de advertencia o precaución.*

*Este símbolo indica situaciones que pueden causar ciertas operaciones y las acciones que se deben llevar a cabo para evitarlas.*



*Símbolo de obligación.*

*Este símbolo indica acciones y operaciones que hay que realizar obligatoriamente.*



*Símbolo de información.*

*Este símbolo indica notas, avisos y consejos.*





*Símbolo de documentación adicional.*

*Este símbolo indica que hay otro documento con información más específica o detallada.*

**Símbolos que puede llevar el producto.**



*Símbolo de tierra.*

*Este símbolo indica que dicho punto puede estar bajo tensión eléctrica.*



*Componentes ESD.*

*Este símbolo identifica las tarjetas con componentes ESD (componentes sensibles a cargas electrostáticas).*



# 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.



CNC 8060  
CNC 8065

(REF: 1603)

## 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 months length.

## SERVICE CONTRACTS

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



CNC 8060  
CNC 8065

(REF: 1603)

## 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 8060  
CNC 8065

(REF: 1603)



# 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 75° 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.



CNC 8060  
CNC 8065

(REF: 1603)





# 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 |            |           |            |



CNC 8060  
CNC 8065

(REF: 1603)

## 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.*

# 1.

PREVIOUS NOTIONS ABOUT THE PROBE:  
Activate the probe.

### 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.

(REF: 1603)



CNC 8060  
CNC 8065

**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.****PREVIOUS NOTIONS ABOUT THE PROBE:**

Activate the probe.

**FAGOR** **CNC 8060  
CNC 8065**

(REF: 1603)

## 1.2 Behavior of the feedrate in probing movements.

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

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

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

G93      Machining time in seconds.

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

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

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

### Maximum probing feedrate.

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

# 1.

**PREVIOUS NOTIONS ABOUT THE PROBE:**  
Behavior of the feedrate in probing movements.



CNC 8060  
CNC 8065

(REF: 1603)

### 1.3 First and last subroutines of the probing cycles.



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

Fagor provides the subroutine *Sub\_Probe\_Tool\_Begin.fst* associated with probe input 1 and the subroutine *Sub\_Probe\_Piece\_Begin.fst* associated with probe input 2.

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

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

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

#### Subroutines and M functions defined in the cycle.

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

When starting the execution, the cycle executes first the subroutines *Sub\_Probe\_Tool\_Begin.fst* or *Sub\_Probe\_Piece\_Begin.fst* (accordingly) and then the M-before functions with their associated subroutines.

At the end of the execution, the cycle executes firsts the M-after functions with their corresponding associated subroutines and then the subroutines *Sub\_Probe\_Tool\_End.fst* or *Sub\_Probe\_Piece\_End.fst* (accordingly).



**PREVIOUS NOTIONS ABOUT THE PROBE:**  
First and last subroutines of the probing cycles.



CNC 8060  
CNC 8065

(REF: 1603)

### 1.3.1 Subroutines supplied by Fagor.

**Subroutine Sub\_Probe\_Tool\_Begin.fst supplied by Fagor (may be modified by the user).**

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

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

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

; Check probe in safe mode feature is activated.
$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 activated by default.

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

# 1.

**PREVIOUS NOTIONS ABOUT THE PROBE:**  
First and last subroutines of the probing cycles.



CNC 8060  
CNC 8065

(REF: 1603)

**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"]
  MO
$ENDIF

; Check probe in safe mode feature is activated.
$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 activated by default.

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

**PREVIOUS NOTIONS ABOUT THE PROBE:**  
First and last subroutines of the probing cycles.



CNC 8060  
CNC 8065

(REF: 1603)

## 1.4 Safe probing cycles.

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

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

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

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

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

### PLC marks.

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

#### PROBE1ENA PROBE2ENA

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

These marks should be tested in the subroutines Sub\_Probe\_Tool\_Begin.fst and Sub\_Probe\_Piece\_Begin.fst so the subroutine waits until the mark is active.

#### PROBE1MONIT PROBE2MONIT

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

These marks should be tested in the subroutines Sub\_Probe\_Tool\_Begin.fst and Sub\_Probe\_Piece\_Begin.fst to warn, if they are deactivated, that the probe is in non-safe mode.

1.

PREVIOUS NOTIONS ABOUT THE PROBE:  
Safe probing cycles.



CNC 8060  
CNC 8065

(REF: 1603)



## 2.1 G100/G103. Probing.

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

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

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

### Probing programming.

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

#### Programming format.

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

G100 X..C <F>

G103 X..C <F>

X..C                      Coordinates of the probing point.

F                              Optional. Feedrate.  
If not programmed, the CNC uses the active feedrate.

```
G100 X45.23 Y20.25 Z23.45
```

```
G100 Z50 F100
```

```
G103 X2.6 Z3 F20
```

```
G103 Z1 F20
```

#### Probing feedrate.

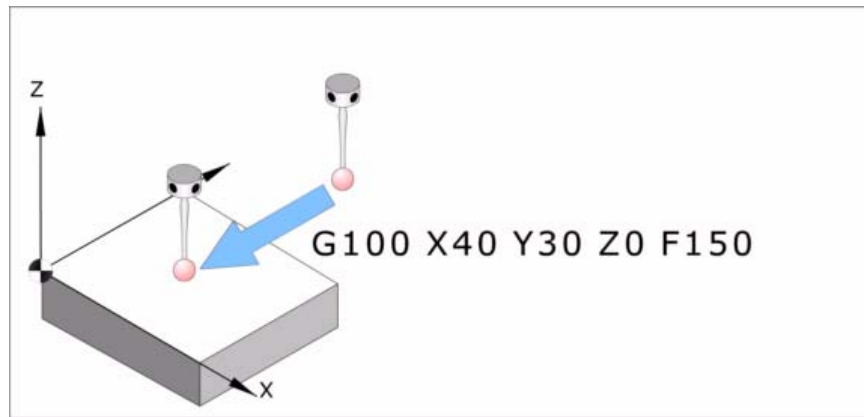
The CNC uses the same feedrate for probing and for machining. The feedrate "F" set for the probe will be the feedrate active at the CNC when done probing.

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

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

2.

PROBING.  
G100/G103. Probing.



**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.

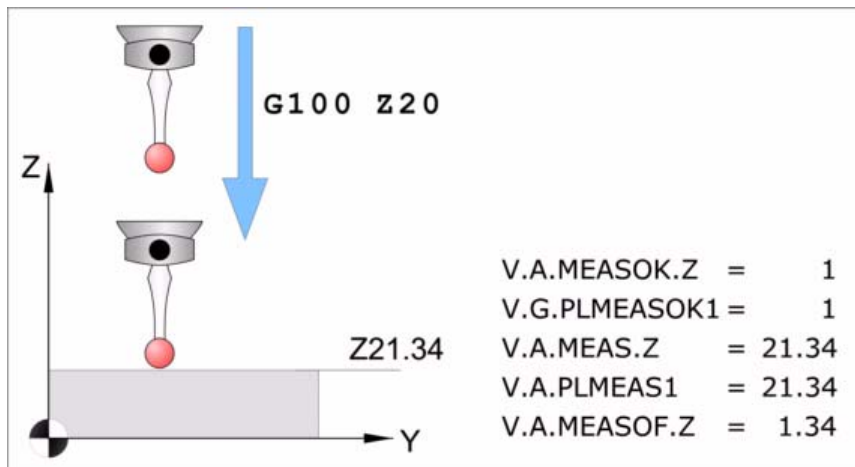
| Mnemoni.  | Variable.  |
|---|--|
| V.G.MEASOK                                      | The probe has made contact (G100) or stopped making contact (G103). <ul style="list-style-type: none"> <li>The variable takes the value of .1. if the probe has made contact (G100) or has stopped making contact (G103).</li> <li>The variables takes the value of .0. if the probe reaches the programmed coordinate.</li> </ul> |
| V.A.MEASOK.xn                                   | Probing done on any axis of the channel. <ul style="list-style-type: none"> <li>The variables of the axes involved in the probing operation take the value of .1. when the probing movement ends.</li> <li>The variables of the rest of the axes take the value of .0.</li> </ul> The variable keeps its value after a reset.      |
| V.G.PLMEASOK1<br>V.G.PLMEASOK2<br>V.G.PLMEASOK3 | Probing on the plane axes completed. <ul style="list-style-type: none"> <li>The variables of the axes involved in the probing operation take the value of .1. when the probing operation ends.</li> <li>The variables of the rest of the axes take the value of .0.</li> </ul>   |
| V.A.MEAS.xn                                     | Measured value. Machine coordinates of the tool base. <ul style="list-style-type: none"> <li>The variables of the axes involved in the probing operation take the measured value.</li> <li>The variables of the rest of the axes take the real position value of the axis.</li> </ul>  |
| V.A.ATIPMEAS.xn                                 | Measured value. Part coordinates of the tool tip. <ul style="list-style-type: none"> <li>The variables of the axes involved in the probing operation take the measured value.</li> <li>The variables of the rest of the axes take the real position value of the axis.</li> </ul>  |



CNC 8060  
CNC 8065

(REF: 1603)

| Mnemoni.                                  | Variable.  |
|---|--|
| V.G.PLMEAS1<br>V.G.PLMEAS2<br>V.G.PLMEAS3 | Value measured on the axes of the plane. Part coordinates of the tool tip. <ul style="list-style-type: none"> <li>The variables of the axes involved in the probing operation take the measured value.</li> <li>The variables of the rest of the axes take the real position value of the axis.</li> </ul> |
| V.A.MEASOF.xn                             | Measuring error. <ul style="list-style-type: none"> <li>The variables of the axes involved in the probing operation take the measuring error (difference between the programmed coordinate and the one measured).</li> <li>The variables of the rest of the axes take the value of ·0·.</li> </ul>         |



## 2.

PROBING.

G100/G103. Probing.

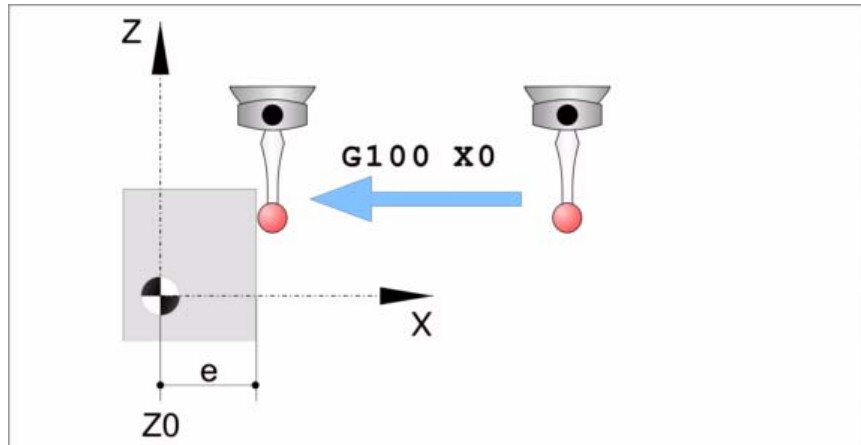


CNC 8060  
CNC 8065

(REF: 1603)

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

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



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

G101 Include the measuring error in the theoretical coordinate.

G102 Exclude the measuring error in the theoretical coordinate.

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

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

### G101 Include the measuring error in the theoretical coordinate.

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

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

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

#### Programming format.

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

G101 X..C

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

G101 X1 Y1 Z1  
G101 X2

# 2.

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

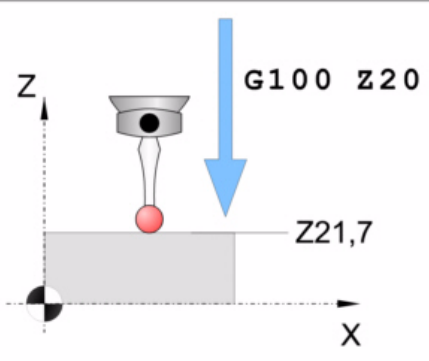


CNC 8060  
CNC 8065

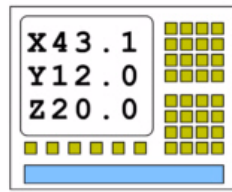
(REF: 1603)

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. |

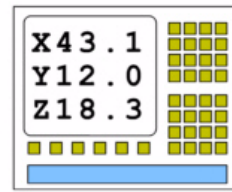


**G101 Z1**



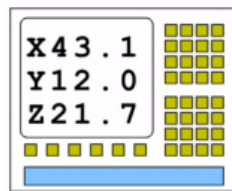
V.A.MEAS.Z = 21.7  
V.A.MEASOF.Z = 0  
V.A.MEASIN.Z = 1.7

**G101 Z2**



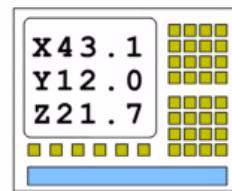
V.A.MEAS.Z = 21.7  
V.A.MEASOF.Z = 0  
V.A.MEASIN.Z = 3.4

**G102**



V.A.MEASOF.Z = 0  
V.A.MEASIN.Z = 0

**G102**



V.A.MEASOF.Z = 0  
V.A.MEASIN.Z = 0

2.

**PROBING.** Include/exclude the measuring error in the theoretical coordinate.



CNC 8060  
CNC 8065

(REF: 1603)

## G102 Exclude the measuring error in the theoretical coordinate.

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

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

### Programming format.

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

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

G102 <X..C>

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

G102 X Y  
G102

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

### Updating the variables after executing function G102.

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

# 2.

## PROBING.

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



CNC 8060  
CNC 8065

(REF: 1603)

## 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.<br>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.

**PROBING.**

G104. Probe movement up to the programmed position.

**FAGOR** 

CNC 8060  
CNC 8065

(REF: 1603)

## 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.<br>"0" = No "1" = Yes   |
| (V.)[n].G.PLMEASOK1   | R   | —   | —   | Probing done on the first axis of the plane (abscissa).<br>"0" = No "1" = Yes                        |
| (V.)[n].G.PLMEASOK2   | R   | —   | —   | Probing done on the second axis of the plane (ordinate).<br>"0" = No "1" = Yes                       |
| (V.)[n].G.PLMEASOK3   | R   | —   | —   | Probing done on the axis perpendicular to the plane.<br>"0" = No "1" = Yes                           |
| (V.)[n].A.MEAS.Xn     | R   | R   | R   | Value measured on the Xn axis.<br>Machine coordinates of the tool base.                              |
| (V.)[n].A.ATIPMEAS.Xn | R   | —   | —   | Value measured on the Xn axis.<br>Part coordinates of the tool tip.                                  |
| (V.)[n].G.PLMEAS1     | R   | —   | —   | Value measured on the first axis of the plane (abscissa).<br>Part coordinates of the tool tip.       |
| (V.)[n].G.PLMEAS2     | R   | —   | —   | Value measured on the second axis of the plane (ordinate).<br>Part coordinates of the tool tip.      |
| (V.)[n].G.PLMEAS3     | R   | —   | —   | Value measured on the axis perpendicular to the plane.<br>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.  |

# 2.

## PROBING.

Properties of measurement related variables.



CNC 8060  
CNC 8065

(REF: 1603)



# CANNED CYCLES. ISO CODED PROGRAMMING.

# 3

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

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

## Programming ISO coded cycles.

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

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

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

## Cycle data programming.

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

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

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

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

## Limitations for executing the cycles.

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

## Canned cycles and the work planes.

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



CNC 8060  
CNC 8065

(REF: 1603)

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

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

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

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

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

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

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

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

#### **On a mill model CNC (milling software installed).**

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

#LATHECY ON - To activate the turning canned cycles.

#LATHECY OFF - To deactivate the turning canned cycles.

#### **On a lathe model CNC (lathe software installed).**

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

#MILLCY ON - To activate the milling canned cycles.

#MILLCY OFF - To deactivate the milling canned cycles.

3.

CANNED CYCLES. ISO CODED PROGRAMMING.



CNC 8060  
CNC 8065

(REF: 1603)

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

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

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

The calibration is done using a tabletop probe.

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

#### Requirements prior to the calibration.

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

#### Tabletop probe.

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

#### Data returned by the cycle after a calibration.

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

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

P298 Error detected on the tool radius.

P299 Error detected on the tool length.

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

#### Data returned by the cycle after a wear measurement.

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

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

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

# 3.

**CANNED CYCLES. ISO CODED PROGRAMMING.**  
#PROBE 1. Tool calibration (dimensions and wear).

**FAGOR** 

**CNC 8060**  
**CNC 8065**

(REF: 1603)

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

P298 Radius wear.

P299 Length wear.

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

### 3.

**CANNED CYCLES. ISO CODED PROGRAMMING.**  
#PROBE 1. Tool calibration (dimensions and wear).



CNC 8060  
CNC 8065

(REF: 1603)

### 3.1.1 Programming the cycle.

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

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

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

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

#### Brief format for calibrating or measuring tool length wear.

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

- To calibrate the tool length.
 

```
#PROBE 1 B <I0> <J0> F <X U Y V Z W>
#PROBE 1 B <I1> <J0> F <S> <N> <D> <E> <X U Y V Z W>
```
- To measure tool length wear.
 

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

#### Brief format for calibrating or measuring tool radius wear.

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

- To calibrate tool radius.
 

```
#PROBE 1 B <I2> <J0> F <K> <S> <N> <X U Y V Z W>
```
- To measure the tool radius wear.
 

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

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

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

- To calibrate the tool radius and length.
 

```
#PROBE 1 B <I3> <J0> F <K> <S> <N> <D> <E> <X U Y V Z W>
```
- Measure or calibrate the tool radius wear and length wear.
 

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

# 3.

**CANNED CYCLES. ISO CODED PROGRAMMING.**  
#PROBE 1. Tool calibration (dimensions and wear).

**FAGOR** 

**CNC 8060**  
**CNC 8065**

(REF: 1603)

## 3.

CANNED CYCLES. ISO CODED PROGRAMMING.  
#PROBE 1. Tool calibration (dimensions and wear).

## Parameter description.

### ·B· Safety distance.

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

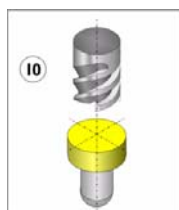
Distance with respect to the probing point that the tool approaches in G00 before touching the probe. The tool must be placed at a distance greater than this value when the cycle is called.

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

Optional parameter, by default 0.

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

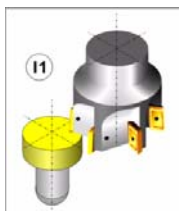
|     |                             |
|-----|-----------------------------|
| I=0 | Length, along its shaft.    |
| I=1 | Length, at its tip.         |
| I=2 | The radius.                 |
| I=3 | The tool radius and length. |



#### Calibration or measuring on the tool shaft.

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

This calibration is done with the spindle stopped.



#### Calibration or measuring at the tool tip.

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

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

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

Optional parameter, by default 0.

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

|     |                    |
|-----|--------------------|
| J=0 | Tool calibration.  |
| J=1 | Measure tool wear. |

### ·F· Probing feedrate.

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

### ·K· Probe side to be used.

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

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

|     |                             |
|-----|-----------------------------|
| K=0 | Calibration on the X+ side. |
| K=1 | Calibration on the X- side. |
| K=2 | Calibration on the Y+ side. |
| K=3 | Calibration on the Y- side. |

**.S- Tool turning speed and direction.**

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

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

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

**.N- Number of edges to measure.**

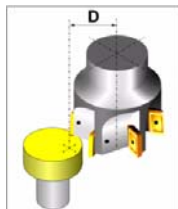
Optional parameter, by default 0 (only one measurement). This parameter is only valid when parameter .S. has been defined with a value greater than 0 (zero).

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

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

**.D- Distance from the probing point to the tool shaft.**

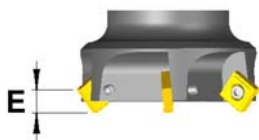
Optional parameter; by default the tool radius. This parameter is only valid when parameter .I. has been defined as I1 or I3.



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

**.E- Distance from the probing point to the tool base.**

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



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

**.L- Maximum length wear allowed.**

Optional parameter, by default 0.

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

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

**.M- Maximum radius wear allowed.**

Optional parameter, by default 0.

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

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

**.C- Behavior when exceeding the maximum of wear.**

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

**3.**

**CANNED CYCLES. ISO CODED PROGRAMMING.**  
#PROBE 1. Tool calibration (dimensions and wear).



CNC 8060  
CNC 8065

(REF: 1603)

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

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

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

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

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

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

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

# 3.

**CANNED CYCLES. ISO CODED PROGRAMMING.**  
#PROBE 1. Tool calibration (dimensions and wear).



CNC 8060  
CNC 8065

(REF: 1603)



### 3.1.2 Basic operation.

#### Calibrate or measure tool length wear.

- 1 Approach movement.  
Rapid probe movement (G00) from the cycle calling point to the safety point. The safety point will be located at a distance  $\cdot B \cdot$  over the probe and at a distance  $\cdot D \cdot$  from the center of the tool to the point to be probed.  
This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.
- 2 Movement to locate an edge (only if  $\cdot N \cdot$  has been defined with a value other than 0).  
The CNC starts the spindle at an  $\cdot S \cdot$  speed and moves the tool at the feedrate  $\cdot F \cdot$  along the axis perpendicular to the active plane until receiving the probe signal. Once it knows the position of an edge, the tool withdraws to the safety point and stops the spindle.
- 3 Probing movement.  
Probing movement along the axis perpendicular to the active plane at the feedrate  $\cdot F \cdot$  until the probe signal is received. Withdrawal to the safety point.  
If  $\cdot N \cdot$  has been defined to a value other than 0, this movement is repeated  $\cdot N \cdot$  times.
- 4 Withdrawal movement.  
Movement in the work plane up to the cycle calling point.

#### Calibrate or measure tool radius wear.

- 1 Approach movement.  
Rapid probe movement (G00) from the cycle calling point to the safety point. The safety point will be located at a  $\cdot B \cdot$  distance from the probe.  
This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.
- 2 Movement to locate an edge (only if  $\cdot N \cdot$  has been defined with a value other than 0).  
The CNC starts the spindle at an  $\cdot S \cdot$  speed and moves the tool at the feedrate  $\cdot F \cdot$  along the selected axis until receiving the probe signal. The CNC stops the spindle and carries out some probing moves to locate the edge more precisely. Once it knows the position of an edge, the tool withdraws to the safety point.
- 3 Probing movement.  
Probing movement at the indicated feedrate  $\cdot F \cdot$  until the probe signal is received. Withdrawal to the safety point.  
If  $\cdot N \cdot$  has been defined to a value other than 0, this movement is repeated  $\cdot N \cdot$  times.
- 4 Withdrawal movement.  
This movement is carried out in two stages. It first moves in the work plane and then along the perpendicular axis and then in the work plane.

#### Calibrate or measure tool radius or length wear.

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

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

# 3.

**CANNED CYCLES. ISO CODED PROGRAMMING.**  
#PROBE 1. Tool calibration (dimensions and wear).

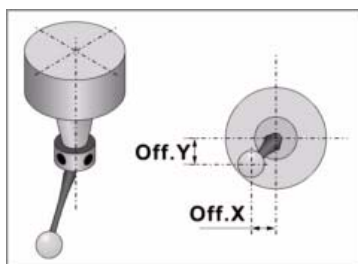
**FAGOR** 

**CNC 8060**  
**CNC 8065**

(REF: 1603)

## 3.2 #PROBE 2. Probe calibration

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



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

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

### Requirements prior to the calibration.

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

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

The following steps must be followed for calibrating the probe:

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

### Data returned by the cycle after the measurement.

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

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

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



CNC 8060  
CNC 8065

(REF: 1603)

### 3.2.1 Programming the cycle.

The programming format for this cycle is.

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

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

**.X Y. Real coordinates of the hole center.**

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

The canned cycles may be programmed in any work plane. See "[Canned cycles and the work planes.](#)" on page 41.

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

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

**.B. Safety distance.**

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

**.J. Real diameter of the hole.**

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

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

**.E. Withdrawal distance after initial probing.**

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

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

**.H. Feedrate for the first probing movement.**

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

**.F. Probing feedrate.**

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

# 3.

CANNED CYCLES. ISO CODED PROGRAMMING.

#PROBE 2. Probe calibration

**FAGOR** 

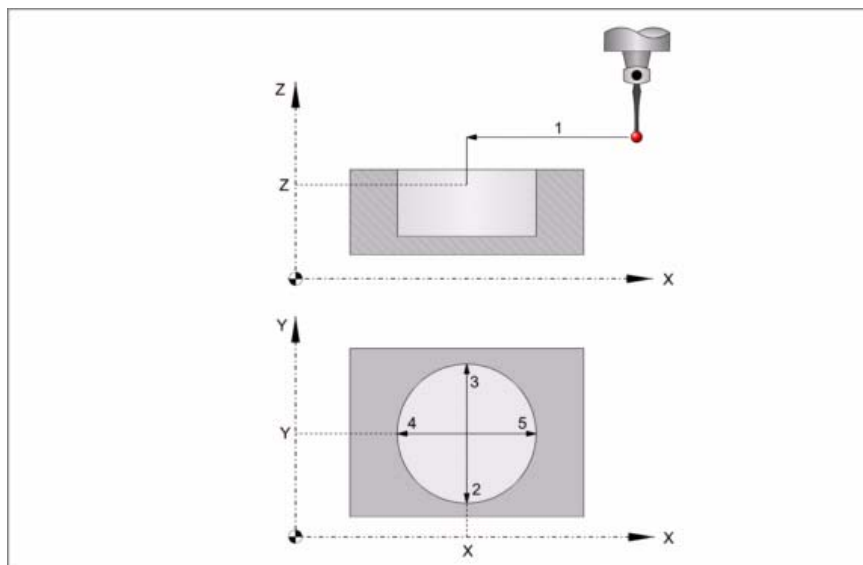
CNC 8060  
CNC 8065

(REF: 1603)

## 3.2.2 Basic operation.

3.

CANNED CYCLES. ISO CODED PROGRAMMING.  
#PROBE 2. Probe calibration



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

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

- 2 Probing of the first point

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

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

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

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

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

- 3 Probing of the second point.

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

- 4 Probing of the third point.

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

- 5 Probing of the fourth point.

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

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

---

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

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

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

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



**FAGOR** 

CNC 8060  
CNC 8065

(REF: 1603)

### 3.3 #PROBE 3. Surfacing measuring.

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

#### Tool wear compensation.

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

#### Data returned by the cycle after the measurement.

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

|      |   |
|------|---|
| P298 | Actual (real) surface coordinate.   |
| P299 | Detected error. Difference between the actual surface coordinate and the programmed theoretical coordinate. |

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

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

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

# 3.

CANNED CYCLES. ISO CODED PROGRAMMING.

#PROBE 3. Surfacing measuring.

**FAGOR** 

CNC 8060  
CNC 8065

(REF: 1603)

### 3.3.1 Programming the cycle.

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

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

- X Y Z Theoretical coordinates of the probing point.
- B Safety distance.
- K Optional. Probing axis.
- F Probing feedrate.
- C Optional. Cycle ending point (calling point or measured point).
- L Optional. Tolerance for the measuring error.
- T Optional. Tool to be corrected.
- D Optional. Tool offset to be corrected.

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

The canned cycles may be programmed in any work plane. See "[Canned cycles and the work planes.](#)" on page 41.

**·B· Safety distance.**

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

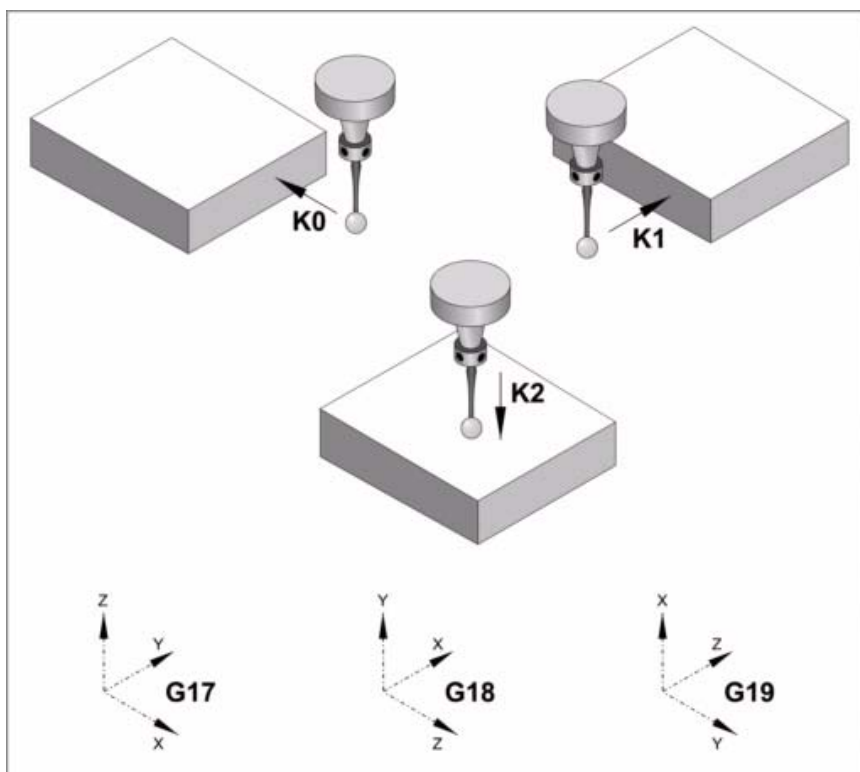
Distance with respect to the point to measure and along the probing axis, to which the probe approaches in G00 before making the probing movement. When calling the cycle, the probe must be located, with respect to the point to be measured, at a greater distance than this value

**·K· Probing axis.**

Optional parameter, by default 0.

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

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



# 3.

CANNED CYCLES. ISO CODED PROGRAMMING.  
#PROBE 3. Surfacing measuring.



CNC 8060  
CNC 8065

(REF: 1603)

**.F. Probing feedrate.**

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

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

Optional parameter, by default 0.

This parameter indicates where the probing cycle ends.

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

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

**.L. Tolerance for the measuring error.**

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

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

**.T. Tool to be corrected.**

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

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

**.D. Tool offset to be corrected.**

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

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

3.

CANNED CYCLES. ISO CODED PROGRAMMING.

#PROBE 3. Surfacing measuring.



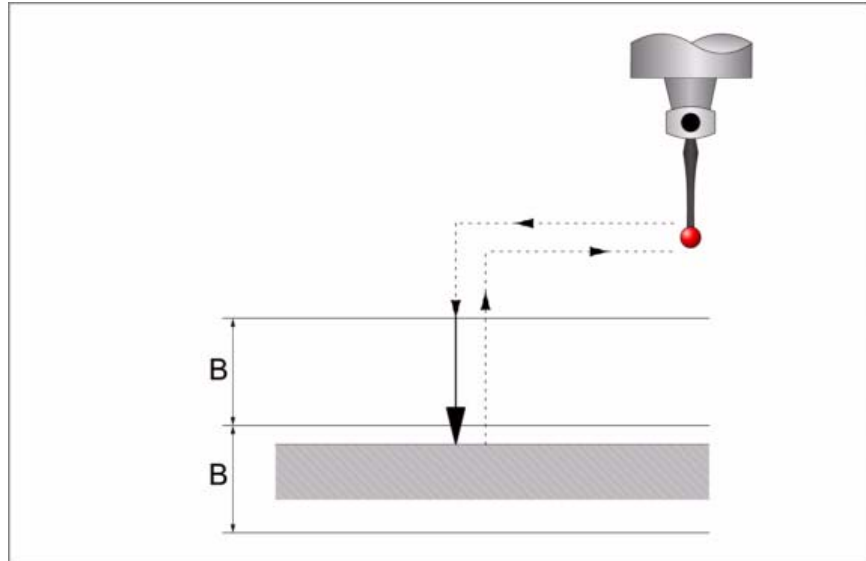
CNC 8060  
CNC 8065

(REF: 1603)

### 3.3.2 Basic operation.

# 3.

**CANNED CYCLES. ISO CODED PROGRAMMING.**  
#PROBE 3. Surfacing measuring.



#### 1 Approach movement.

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

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

#### 2 Probing movement.

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

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

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

#### 3 Withdrawal movement.

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

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



CNC 8060  
CNC 8065

(REF: 1603)



### 3.4 #PROBE 4. Outside corner measuring.

This cycle calculates the position of an outside corner.

#### Data returned by the cycle after the measurement.

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

- P296 Real coordinate of the corner along the abscissa axis.
- P297 Real coordinate of the corner along the ordinate axis.
- P298 Error detected along the abscissa axis. Difference between the actual corner coordinate and the programmed theoretical coordinate.
- P299 Error detected along the ordinate axis. Difference between the actual corner coordinate and the programmed theoretical coordinate.

3.

**CANNED CYCLES. ISO CODED PROGRAMMING.**  
#PROBE 4. Outside corner measuring.



CNC 8060  
CNC 8065

(REF: 1603)

### 3.4.1 Programming the cycle.

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

#PROBE 4 X Y Z B F

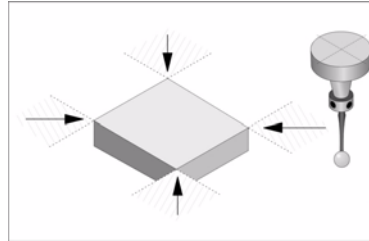
X Y Z Theoretical coordinates corner being measured.

B Safety distance.

F Probing feedrate.

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

The canned cycles may be programmed in any work plane. See "[Canned cycles and the work planes.](#)" on page 41.



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

#### ·B· Safety distance.

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

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

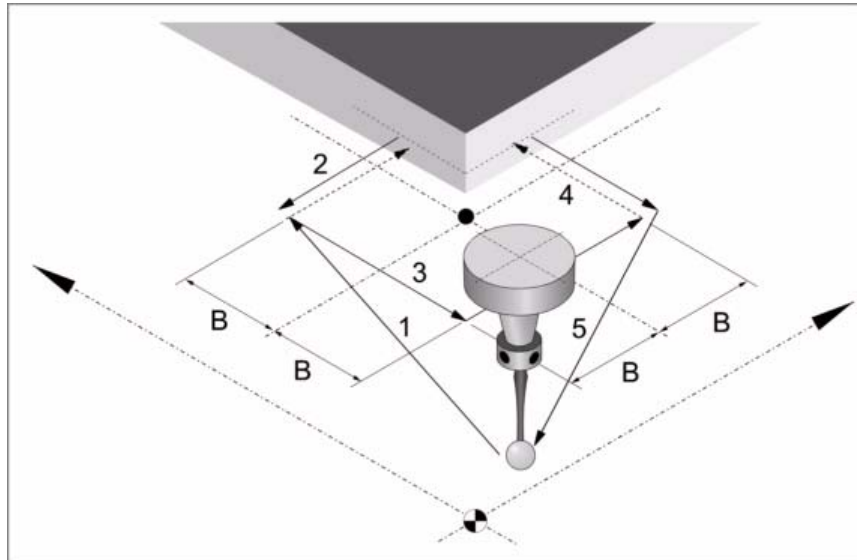
#### ·F· Probing feedrate.

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

# 3.

CANNED CYCLES. ISO CODED PROGRAMMING.  
#PROBE 4. Outside corner measuring.

### 3.4.2 Basic operation.



- 1 Approach movement to the first side to be probed.  
Rapid probe movement (G00) from the cycle calling point to the first approach point located at a  $\cdot B \cdot$  distance from the side to be probed.  
This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.
- 2 Probing on the first side.  
The probing on each side consists of a probing move and a withdrawal move.  
Probing movement. Probe movement along the abscissa axis at feedrate  $\cdot F \cdot$  until the probe signal is received. The maximum probing distance is  $\cdot 2B \cdot$ . If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.  
Withdrawal movement. Rapid probe movement (G00) from the probing point to the first approach point.
- 3 Approach movement to the second side to be probed.  
Rapid probe move (G00) from the first approach point to the second.  
This approach movement is made in two stages. It first moves in the work plane and then along the ordinate axis and then along the abscissa axis.
- 4 Probing on the second side.  
Probing movement. Probe movement along the ordinate axis at feedrate  $\cdot F \cdot$  until the probe signal is received. The maximum probing distance is  $\cdot 2B \cdot$ . If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.  
Withdrawal movement. Rapid probe movement (G00) from the probing point to the second approach point.
- 5 Withdrawal movement.  
Rapid probe movement (G00) from the second approach point to the cycle calling point.  
This withdrawal movement is made in two stages. It first moves in the work plane and then along the perpendicular axis and then in the work plane.

# 3.

**CANNED CYCLES. ISO CODED PROGRAMMING.**

#PROBE 4. Outside corner measuring.

**FAGOR** 

CNC 8060  
CNC 8065

(REF: 1603)

### 3.5 #PROBE 5. Inside corner measuring.

This cycle calculates the position of an inside corner.

#### Data returned by the cycle after the measurement.

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

|      |  |
|------|--|
| P296 | Real coordinate of the corner along the abscissa axis.   |
| P297 | Real coordinate of the corner along the ordinate axis.   |
| P298 | Error detected along the abscissa axis. Difference between the actual corner coordinate and the programmed theoretical coordinate. |
| P299 | Error detected along the ordinate axis. Difference between the actual corner coordinate and the programmed theoretical coordinate. |

# 3.

**CANNED CYCLES. ISO CODED PROGRAMMING.**  
#PROBE 5. Inside corner measuring.



CNC 8060  
CNC 8065

(REF: 1603)

### 3.5.1 Programming the cycle.

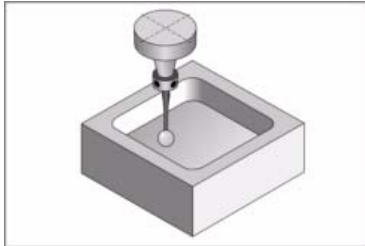
The programming format for this cycle is. Optional parameters are indicated between angle brackets.

#PROBE 5 X Y Z B F

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

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

The canned cycles may be programmed in any work plane. See "*Canned cycles and the work planes.*" on page 41.



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

**·B· Safety distance.**

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

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

**·F· Probing feedrate.**

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

**3.**

**CANNED CYCLES. ISO CODED PROGRAMMING.**

#PROBE 5. Inside corner measuring.



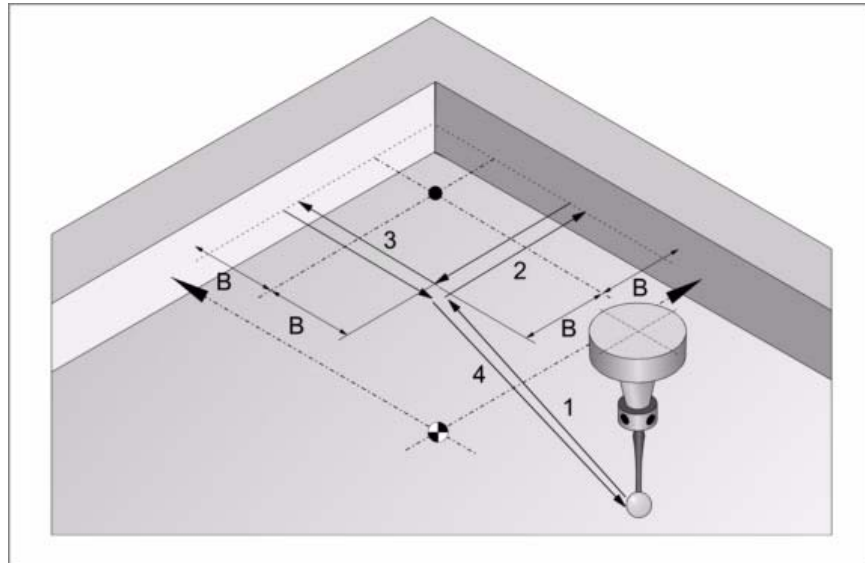
CNC 8060  
CNC 8065

(REF: 1603)

## 3.5.2 Basic operation.

3.

CANNED CYCLES. ISO CODED PROGRAMMING.  
#PROBE 5. Inside corner measuring.



## 1 Approach movement.

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

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

## 2 Probing on the first side.

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

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

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

## 3 Probing on the second side.

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

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

## 4 Withdrawal movement.

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

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



CNC 8060  
CNC 8065

(REF: 1603)

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

This cycle calculates the angle of the part with respect to the abscissa axis. This cycle may be used to measure angles between  $\pm 45^\circ$ .

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

#### Data returned by the cycle after the measurement.

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

P295      Inclination angle between the part and the abscissa axis.

**3.****CANNED CYCLES. ISO CODED PROGRAMMING.**

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

**FAGOR** **CNC 8060  
CNC 8065**

(REF: 1603)

### 3.6.1 Programming the cycle.

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

#PROBE 6 X Y Z B F

X Y Z     Theoretical coordinates corner being measured.

B         Safety distance.

F         Probing feedrate.

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

The canned cycles may be programmed in any work plane. See "[Canned cycles and the work planes.](#)" on page 41.

#### ·B· Safety distance.

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

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

#### ·F· Probing feedrate.

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

# 3.

**CANNED CYCLES. ISO CODED PROGRAMMING.**  
#PROBE 6. Measuring the angle with the abscissa axis.

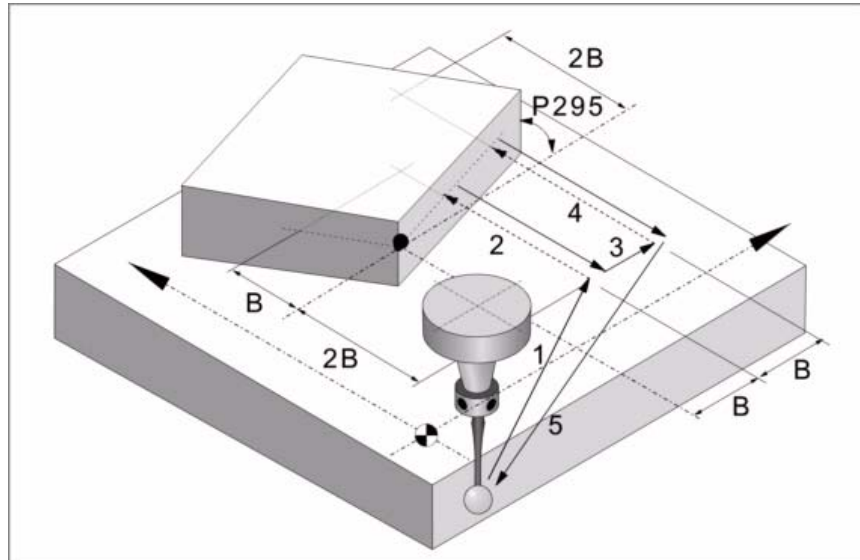


CNC 8060  
CNC 8065

(REF: 1603)



### 3.6.2 Basic operation.



**1** First approach movement.

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

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

**2** First probing move.

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

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

**3** Second approach movement.

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

**4** Second probe.

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

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

**5** Withdrawal movement.

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

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

**3.**

**CANNED CYCLES. ISO CODED PROGRAMMING.**

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



CNC 8060  
CNC 8065

(REF: 1603)

### 3.7 #PROBE 7. Outside corner and angle measuring.

This cycle calculates the position of an outside corner and the angle of the part with respect to the abscissa axis. This cycle may be used to measure angles between  $\pm 45^\circ$ .

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

#### Data returned by the cycle after the measurement.

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

|      |  |
|------|--|
| P295 | Inclination angle between the part and the abscissa axis.  |
| P296 | Real coordinate of the corner along the abscissa axis.   |
| P297 | Real coordinate of the corner along the ordinate axis.   |
| P298 | Error detected along the abscissa axis. Difference between the actual corner coordinate and the programmed theoretical coordinate. |
| P299 | Error detected along the ordinate axis. Difference between the actual corner coordinate and the programmed theoretical coordinate. |

# 3.

**CANNED CYCLES. ISO CODED PROGRAMMING.**  
#PROBE 7. Outside corner and angle measuring.



CNC 8060  
CNC 8065

(REF: 1603)

### 3.7.1 Programming the cycle.

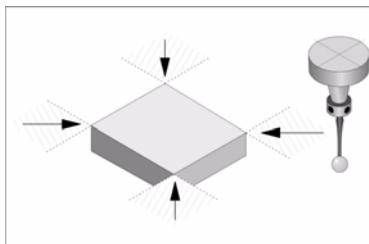
The programming format for this cycle is. Optional parameters are indicated between angle brackets.

#PROBE 7 X Y Z B F

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

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

The canned cycles may be programmed in any work plane. See "*Canned cycles and the work planes.*" on page 41.



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

**.B. Safety distance.**

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

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

**.F. Probing feedrate.**

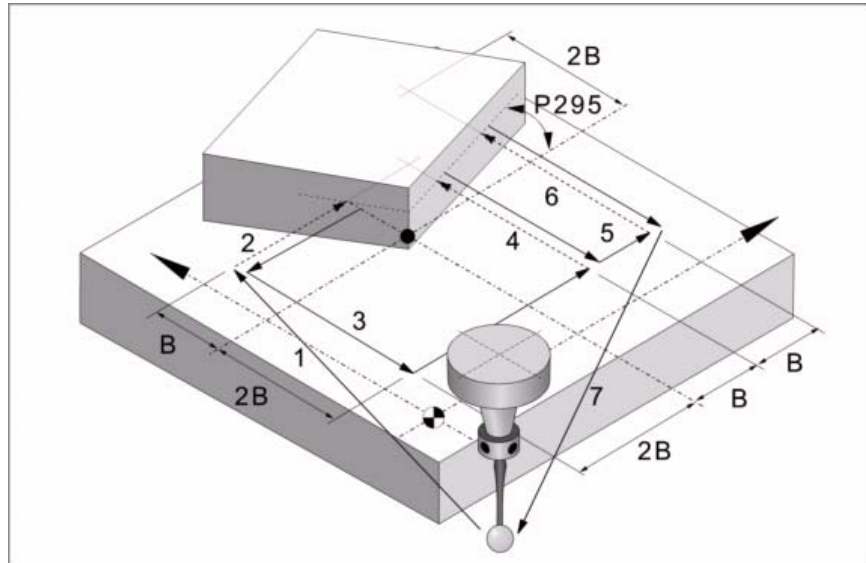
This parameter sets the probing feedrate. The rest of the movements will be carried out in G00.

**3.**

**CANNED CYCLES. ISO CODED PROGRAMMING.**

#PROBE 7. Outside corner and angle measuring.

## 3.7.2 Basic operation.



## 1 First approach movement.

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

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

## 2 First probing move.

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

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

## 3 Second approach movement.

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

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

## 4 Second probe.

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

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

## 5 Third approach movement.

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

## 6 Third probing move.

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

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

## 7 Withdrawal movement.

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

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

3.

CANNED CYCLES. ISO CODED PROGRAMMING.  
#PROBE 7. Outside corner and angle measuring.



CNC 8060  
CNC 8065

(REF: 1603)

### 3.8 #PROBE 8. Hole measuring.

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

#### Data returned by the cycle after the measurement.

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

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

# 3.

**CANNED CYCLES. ISO CODED PROGRAMMING.**

#PROBE 8. Hole measuring.

**FAGOR** 

**CNC 8060  
CNC 8065**

(REF: 1603)

### 3.8.1 Programming the cycle.

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

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

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

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

The canned cycles may be programmed in any work plane. See "[Canned cycles and the work planes.](#)" on page 41.

#### ·B· Safety distance.

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

#### ·J· Theoretical diameter of the hole.

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

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

#### ·E· Withdrawal distance after initial probing.

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

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

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

Optional parameter, by default 0.

This parameter indicates where the probing cycle ends.

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

C=1 The cycle will finish with the probe at the real (calculated) hole center.

#### ·H· Feedrate for the first probing movement.

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

#### ·F· Probing feedrate.

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

# 3.

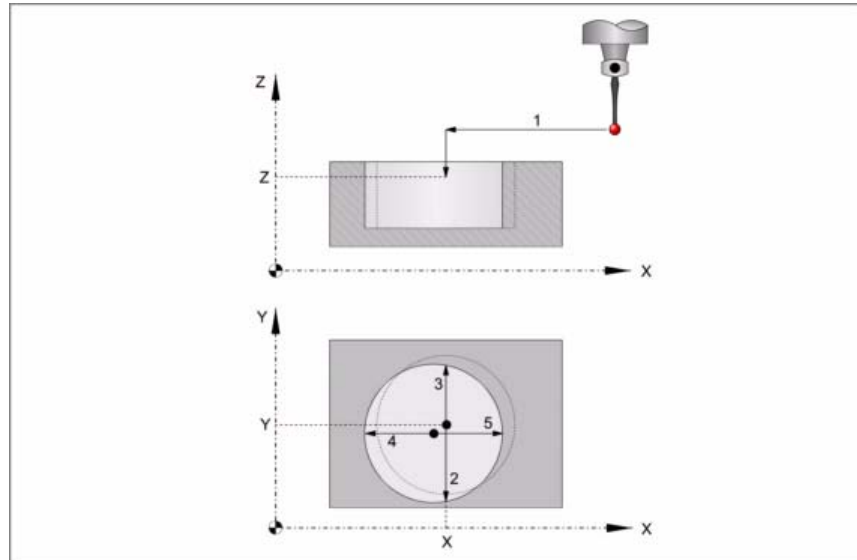
CANNED CYCLES. ISO CODED PROGRAMMING.  
#PROBE 8. Hole measuring.



CNC 8060  
CNC 8065

(REF: 1603)

### 3.8.2 Basic operation.



#### 1 Approach movement.

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

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

#### 2 Probing of the first point

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

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

Withdrawal movement. Probe's rapid movement (G00) the distance indicated by  $\cdot E \cdot$ .

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

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

#### 3 Probing of the second point.

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

#### 4 Probing of the third point.

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

#### 5 Probing of the fourth point.

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

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

#### 6 Withdrawal movement (if it has been defined).

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

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

# 3.

CANNED CYCLES. ISO CODED PROGRAMMING.

#PROBE 8. Hole measuring.

**FAGOR** 

CNC 8060  
CNC 8065

(REF: 1603)

### 3.9 #PROBE 9. Circular boss measuring.

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

#### Data returned by the cycle after the measurement.

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

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

# 3.

**CANNED CYCLES. ISO CODED PROGRAMMING.**  
#PROBE 9. Circular boss measuring.



CNC 8060  
CNC 8065

(REF: 1603)



### 3.9.1 Programming the cycle.

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

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

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

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

The canned cycles may be programmed in any work plane. See "[Canned cycles and the work planes.](#)" on page 41.

#### ·B· Safety distance.

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

#### ·J· Theoretical diameter of the boss.

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

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

#### ·E· Withdrawal distance after initial probing.

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

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

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

Optional parameter, by default 0.

This parameter indicates where the probing cycle ends.

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

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

#### ·H· Feedrate for the first probing movement.

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

#### ·F· Probing feedrate.

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

# 3.

CANNED CYCLES. ISO CODED PROGRAMMING.

#PROBE 9. Circular boss measuring.

**FAGOR** 

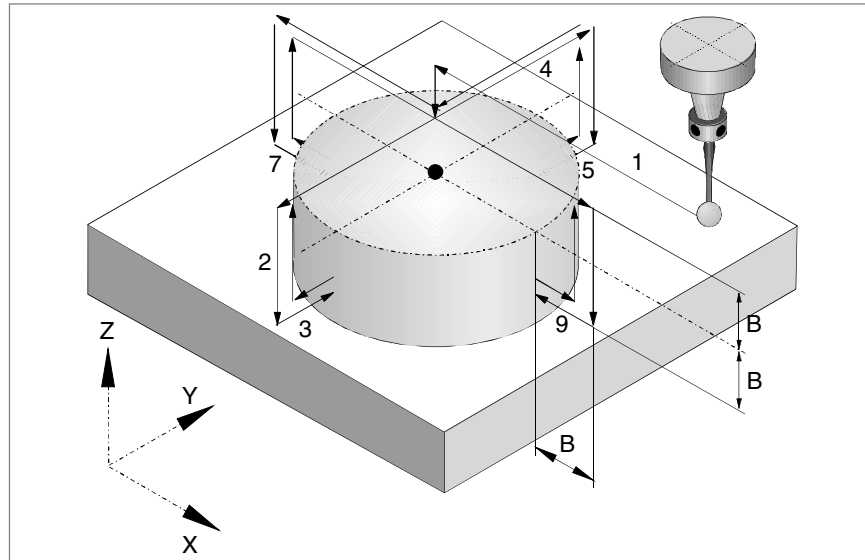
CNC 8060  
CNC 8065

(REF: 1603)

## 3.9.2 Basic operation.

## 3.

CANNED CYCLES. ISO CODED PROGRAMMING.  
#PROBE 9. Circular boss measuring.



**1** Positioning over the boss center.

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

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

**2** First approach movement.

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

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

**3** Probing of the first point

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

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

Withdrawal movement. Probe's rapid movement (G00) the distance indicated by  $\cdot E \cdot$ .

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

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

**4** Second approach movement.

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

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

**5** Probing of the second point.

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

**6** Third approach movement.

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

**7** Probing of the third point.

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

**8** Fourth approach movement.

It is done like the previous one.



CNC 8060  
CNC 8065

(REF: 1603)

9 Probing of the fourth point.

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

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

10 Withdrawal movement (if it has been defined).

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

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

3.

**CANNED CYCLES. ISO CODED PROGRAMMING.**

#PROBE 9. Circular boss measuring.



CNC 8060  
CNC 8065

(REF: 1603)

### 3.10 #PROBE 10. Rectangular part centering.

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

#### Requirements prior to the calibration.

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

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

#### Data returned by the cycle after the measurement.

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

|      |   |
|------|---|
| P296 | Inclination angle between the part and the abscissa axis. |
| P297 | Part surface coordinate.                                  |
| P298 | Part center coordinate along the abscissa axis.           |
| P299 | Part center coordinate along the ordinate axis.           |

# 3.

**CANNED CYCLES. ISO CODED PROGRAMMING.**  
#PROBE 10. Rectangular part centering.



CNC 8060  
CNC 8065

(REF: 1603)

### 3.10.1 Programming the cycle.

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

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

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

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

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

The canned cycles may be programmed in any work plane. See "[Canned cycles and the work planes.](#)" on page 41.

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

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

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

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

#### ·C· Centering type.

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

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

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

Optional parameter, by default 0.

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

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

#### ·L· Surface coordinate measuring.

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

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

# 3.

CANNED CYCLES. ISO CODED PROGRAMMING.

#PROBE 10. Rectangular part centering.

FAGOR 

CNC 8060  
CNC 8065

(REF: 1603)

## 3.

**CANNED CYCLES. ISO CODED PROGRAMMING.**  
 #PROBE 10. Rectangular part centering.

**·B· Safety distance.**

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

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

**·D· Safety distance along Z.**

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

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

**·E· Withdrawal distance after initial probing.**

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

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

**·H· Feedrate for the first probing movement.**

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

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

**·F· Probing feedrate.**

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

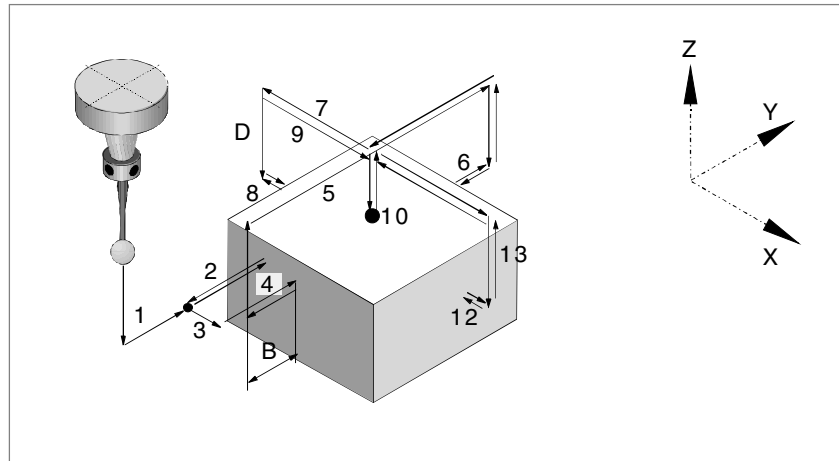
This parameter sets the feedrate for the second probing movement.

**·Q· Feedrate for approach movements.**

Optional parameter; by default, Feedrate G00.

This parameter sets the approaching feedrate.

## 3.10.2 Basic operation.



- 1 Approach movement (if any of parameters  $\cdot XYZ \cdot$  has been defined).  
 Movement of the probe at feedrate  $\cdot Q \cdot$  from the cycle calling point to the first approach point. defined by parameters  $\cdot XYZ \cdot$ .  
 This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.
- 2 Probing of the first point  
 The cycle makes a first probing movement, a withdrawal movement and a second probing move. After the second probing move, the probe withdraws to the approach point.  
 First probing movement. Probe movement along the selected axis at feedrate  $\cdot H \cdot$  until the probe signal is received.  
 Withdrawal movement. Return of the probe in rapid (G00) the distance indicated in  $\cdot E \cdot$ .  
 Second probe. Probing movement at feedrate  $\cdot F \cdot$  until the probe signal is received.  
 Withdrawal movement. Rapid probe movement (G00) from the probing point to the starting point.
- 3 If  $C=0$  has been programmed, repeat the probing somewhere else on the surface. The cycle uses this second probing to calculate the inclination of the part.  
 Approach movement. Movement parallel to the probed side to touch a different point of the same side, located at  $1/4$  the length programmed in  $\cdot J \cdot$ .  
 Probing movement. Probing movement at feedrate  $\cdot F \cdot$  until the probe signal is received.  
 Withdrawal movement. The probe returns to the safety position located at a  $\cdot B \cdot$  distance from the part.
- 4 Approach movement to the second probing point.  
 Movement of the probe from the first approach point to the second located in front of the first one.  
 The perpendicular axis moves the  $\cdot D \cdot$  distance up in rapid (G00). Then the probe moves over the part, at feedrate  $\cdot Q \cdot$ , to the next approach point. The probe moves a  $\cdot D \cdot$  distance back down at feedrate  $\cdot H \cdot$ .  
 For the movement, the cycle takes into consideration the length of the part and its inclination angle. If in this last movement, the probe touches the part, the CNC shows an error at the cycle parameter value.
- 5 Probing of the second point.  
 It is done like the previous one. The probe returns to the safety position located at a  $\cdot B \cdot$  distance from the part.
- 6 If either  $C=0$  or  $C=2$  has been programmed, approach movement to the third probing point.  
 Movement of the probe from the second approach point to the center of the part and then to the third point.
- 7 If either  $C=0$  or  $C=2$  has been programmed, probing of the third point.  
 It is done like the previous one.

3.

CANNED CYCLES. ISO CODED PROGRAMMING.  
#PROBE 10. Rectangular part centering.

**FAGOR** 

CNC 8060  
CNC 8065

(REF: 1603)

## 3.

**CANNED CYCLES. ISO CODED PROGRAMMING.**  
 #PROBE 10. Rectangular part centering.

- 8 Approach movement to the top surface (only if it has been programmed).  
 When selecting to measure the surface (parameter  $\cdot L \cdot$ ), the probe moves the  $\cdot D \cdot$  distance up in rapid (G00) and the probe moves to the center of the part.
- 9 Measuring the top surface (only if it has been programmed).  
 The probe makes a first probing move, a withdrawal movement and a second probing move. After the second probing move, the probe withdraws to the approach point.  
 First probing movement. Probing movement, at feedrate  $\cdot H \cdot$ , until touching the part surface.  
 Withdrawal movement. Return of the probe in rapid (G00) the distance indicated in  $\cdot E \cdot$ .  
 Second probe. Probing movement at feedrate  $\cdot F \cdot$  until the probe signal is received.  
 Withdrawal movement. Rapid probe movement (G00) from the probing point to the safety position.
- 10 If either C=0 or C=2 has been programmed, approach movement to the fourth probing point.  
 Movement of the probe from the third approach point to the fourth.
- 11 If either C=0 or C=2 has been programmed, probing of the fourth point.  
 It is done like the previous one.
- 12 Probe positioning at the calculated center.  
 This movement is carried out in two stages. It first moves along the perpendicular axis in rapid (G00) and then in the plane at feedrate  $\cdot Q \cdot$ .



CNC 8060  
 CNC 8065

(REF: 1603)



### 3.11 #PROBE 11. Circular part centering.

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

#### Requirements prior to the calibration.

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

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

#### Data returned by the cycle after the measurement.

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

- P297 Part surface coordinate.
- P298 Part center coordinate along the abscissa axis.
- P299 Part center coordinate along the ordinate axis.

3.

CANNED CYCLES. ISO CODED PROGRAMMING.

#PROBE 11. Circular part centering.



CNC 8060  
CNC 8065

(REF: 1603)

### 3.11.1 Programming the cycle.

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

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

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

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

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

The canned cycles may be programmed in any work plane. See "[Canned cycles and the work planes.](#)" on page 41.

#### ·J· Part diameter.

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

#### ·C· Centering type.

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

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

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

Optional parameter, by default 0.

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

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

#### ·L· Surface coordinate measuring.

Optional parameter, by default 0.

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

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

# 3.

CANNED CYCLES. ISO CODED PROGRAMMING.  
#PROBE 11. Circular part centering.



CNC 8060  
CNC 8065

(REF: 1603)

**.B. Safety distance.**

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

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

**.D. Safety distance along Z.**

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

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

**.E. Withdrawal distance after initial probing.**

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

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

**.H. Feedrate for the first probing movement.**

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

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

**.F. Probing feedrate.**

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

This parameter sets the feedrate for the second probing movement.

**.Q. Feedrate for approach movements.**

Optional parameter; by default, Feedrate G00.

This parameter sets the approaching feedrate.

3.

CANNED CYCLES. ISO CODED PROGRAMMING.

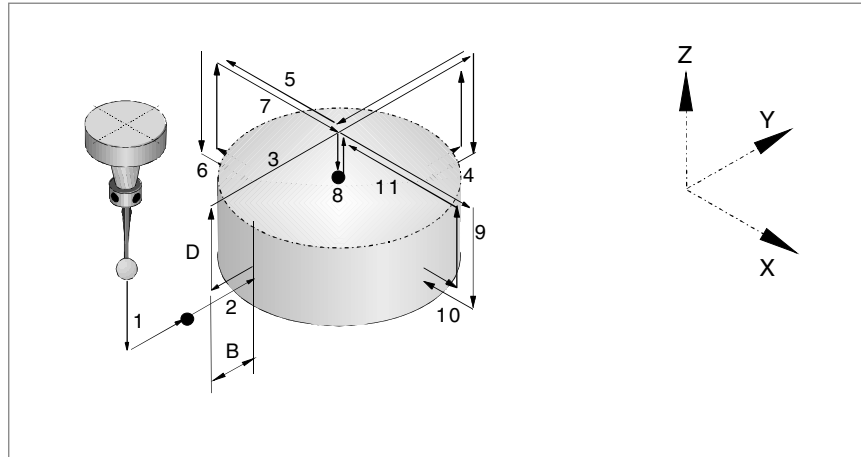
#PROBE 11. Circular part centering.



CNC 8060  
CNC 8065

(REF: 1603)

## 3.11.2 Basic operation.



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

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

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

- 2 Probing of the first point

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

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

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

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

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

- 3 Approach movement to the second probing point.

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

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

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

- 4 Probing of the second point.

It is done like the previous one.

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

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

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

It is done like the previous one.

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

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

3.

CANNED CYCLES. ISO CODED PROGRAMMING.  
#PROBE 11. Circular part centering.

**FAGOR** 

CNC 8060  
CNC 8065

(REF: 1603)

- 8 Measuring the top surface (only if it has been programmed).  
The probe makes a first probing move, a withdrawal movement and a second probing move. After the second probing move, the probe withdraws to the approach point.  
First probing movement. Probing movement, at feedrate ·H·, until touching the part surface.  
Withdrawal movement. Return of the probe in rapid (G00) the distance indicated in ·E·.  
Second probe. Probing movement at feedrate ·F· until the probe signal is received.  
Withdrawal movement. Rapid probe movement (G00) from the probing point to the safety position.
- 9 If C=0 has been programmed, approach movement to the fourth probing point.  
Movement of the probe from the third approach point to the fourth.
- 10 If C=0 has been programmed, probing of the fourth point.  
It is done like the previous one.
- 11 Probe positioning at the calculated center.  
This movement is carried out in two stages. It first moves along the perpendicular axis in rapid (G00) and then in the plane at feedrate ·Q·.

3.

CANNED CYCLES. ISO CODED PROGRAMMING.

#PROBE 11. Circular part centering.



CNC 8060  
CNC 8065

(REF: 1603)

### 3.12 #PROBE 12. Tabletop probe calibration.

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

#### Requirements prior to the calibration.

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

#### Data returned by the cycle after the measurement.

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

|      |  |
|------|--|
| P295 | Real coordinate of the probe side closest to the origin, along the abscissa axis.    |
| P296 | Real coordinate of the probe side furthest from the origin, along the abscissa axis. |
| P297 | Real coordinate of the probe side closest to the origin, along the ordinate axis.    |
| P298 | Real coordinate of the probe side furthest from the origin, along the ordinate axis. |
| P299 | Real coordinate of the probe side measured along the longitudinal axis.              |

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

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



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

# 3.

CANNED CYCLES. ISO CODED PROGRAMMING.  
#PROBE 12. Tabletop probe calibration.

### 3.12.1 Programming the cycle.

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

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

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

#### ·B· Safety distance.

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

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

#### ·E· Withdrawal distance after initial probing.

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

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

#### ·H· Feedrate for the first probing movement.

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

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

#### ·F· Probing feedrate.

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

This parameter sets the feedrate for the second probing movement.

#### ·I· Type of calibration.

Optional parameter, by default 0.

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

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

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

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

# 3.

CANNED CYCLES. ISO CODED PROGRAMMING.

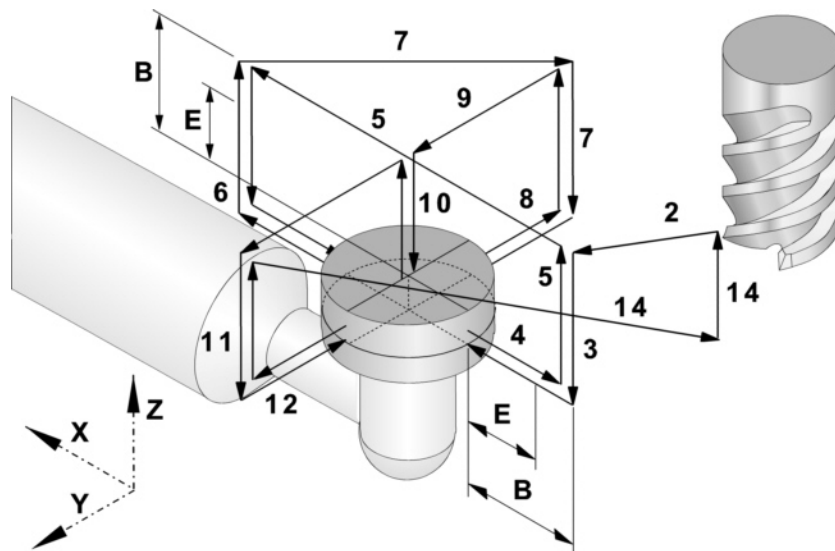
#PROBE 12. Tabletop probe calibration.

**FAGOR** 

CNC 8060  
CNC 8065

(REF: 1603)

### 3.12.2 Basic operation.



#### Initial considerations.

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

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

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

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

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

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

#### Basic operation.

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

This approach movement is made in two stages. If the tool is separated from the Z side of the probe a shorter distance than the one indicated in parameter  $\cdot B \cdot$ , the movement is carried out first along the perpendicular axis and then in the work plane. Otherwise, it first moves in the work plane and then along the axis perpendicular to the plane.

- 3 Approach movement. Tool movement along the perpendicular axis at feedrate  $\cdot H \cdot$  up to the probing Z coordinate (in the middle of the probe). If the tool touches the probe, the cycle will display the corresponding error message.

# 3.

CANNED CYCLES. ISO CODED PROGRAMMING.  
#PROBE 12. Tabletop probe calibration.

**FAGOR** 

CNC 8060  
CNC 8065

(REF: 1603)



- 4 Probing of the first point
 

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

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

Withdrawal movement. Return of the tool in rapid (G00) the distance indicated in  $\cdot E \cdot$ .

Second probe. Tool movement at the indicated feedrate  $\cdot F \cdot$  until the probe signal is received.

Withdrawal movement. Rapid tool movement (G00) from the probing point to the safety position located at a  $\cdot B \cdot$  distance from the probe.
- 5 Approach movement to the second probing point.
 

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

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

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

It is done along the abscissa axis same as the previous one.
- 7 Approach movement to the third probing point.
 

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

It is done along the ordinate axis same as the previous one.
- 9 Approach movement to the fourth approach point (top surface).
 

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

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

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

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

Withdrawal movement. Return of the tool in rapid (G00) the distance indicated in  $\cdot E \cdot$ .

Second probe. Tool movement at the indicated feedrate  $\cdot F \cdot$  until the probe signal is received.

Withdrawal movement. Rapid tool movement (G00) from the probing point to the safety position located at a  $\cdot B \cdot$  distance from the probe.
- 11 Approach movement to the fifth probing point.
 

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

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

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

It is done along the ordinate axis same as the third probing point.
- 13 Withdrawal movement.
 

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

3.

CANNED CYCLES. ISO CODED PROGRAMMING.  
#PROBE 12. Tabletop probe calibration.

FAGOR 

CNC 8060  
CNC 8065

(REF: 1603)

**14** Withdrawal movement.

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

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

**15** If the cycle is going to do a double calibration, the spindle is oriented to 180° and the cycle repeats steps 2 through 14.**3.**

**CANNED CYCLES. ISO CODED PROGRAMMING.**  
#PROBE 12. Tabletop probe calibration.



CNC 8060  
CNC 8065

(REF: 1603)

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

#### Check which parameters have been programmed

(V.)C.CALLP\_a-z

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

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

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

#### Check the value of the programmed parameters.

(V.)C.a-z

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

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

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

# 3.

**CANNED CYCLES. ISO CODED PROGRAMMING.**

Check the data of the canned cycles (variables).

**FAGOR** 

**CNC 8060  
CNC 8065**

(REF: 1603)

# 3.

## **CANNED CYCLES. ISO CODED PROGRAMMING.**

Check the data of the canned cycles (variables).



CNC 8060  
CNC 8065

(REF: 1603)

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

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

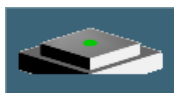
## Programming the cycles of the editor.

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



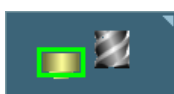
### Measurement.

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



### Part centering.

Rectangular or circular part centering.



### Probe calibration.

Tabletop probe calibration and the measuring probe.



### Tool calibration.

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

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

## Cycle data programming.

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

## Limitations for executing the cycles.

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

## Canned cycles and the work planes.

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

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

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

# 4.

## CANNED CYCLES. CYCLE EDITOR.



CNC 8060  
CNC 8065

(REF: 1603)

## 4.1 How to define the data of the editor.

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

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

### Manual data entry.

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

### Leaving some data undefined.

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

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

### Defining data using arithmetic parameters..

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

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

### Teach-in mode for data entry.



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

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

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

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

# 4.

CANNED CYCLES. CYCLE EDITOR.

How to define the data of the editor.

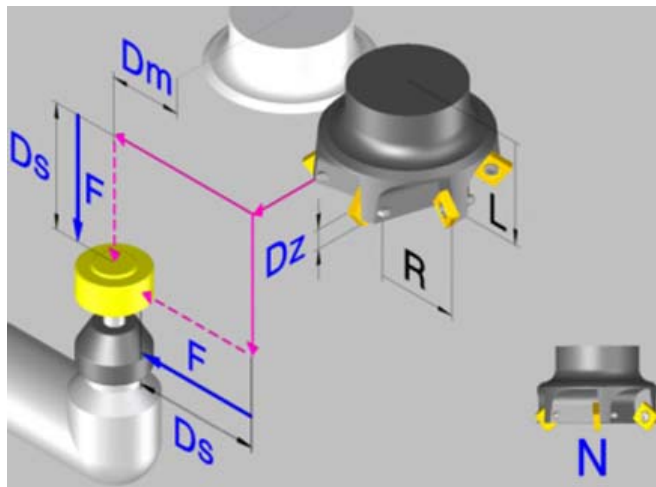
**FAGOR** 

CNC 8060  
CNC 8065

(REF: 1603)

## 4.2 Tool calibration (dimensions and wear)

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



# 4.

CANNED CYCLES. CYCLE EDITOR.  
Tool calibration (dimensions and wear)



Calibrate the length of a tool.



Calibrate the radius of a tool.



Tool length and radius calibration.



Measuring the length wear of a tool.



Measure the radius wear of a tool.



Measuring the length and radius wear of a tool.

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

### Requirements prior to the calibration.

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

#### Tabletop probe.

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

### Data returned by the cycle after a calibration.

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



CNC 8060  
CNC 8065

(REF: 1603)



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

P298 Error detected on the tool radius.

P299 Error detected on the tool length.

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

### Data returned by the cycle after a wear measurement.

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

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

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

P298 Radius wear.

P299 Length wear.

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

# 4.

**CANNED CYCLES. CYCLE EDITOR.**

Tool calibration (dimensions and wear)



CNC 8060  
CNC 8065

(REF: 1603)

## 4.2.1 Programming the cycle.

4.

CANNED CYCLES. CYCLE EDITOR.  
Tool calibration (dimensions and wear)

### Tool to be calibrated.

**·T· Tool to be calibrated.**

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

**·D· Tool offset.**

Offset of the tool to be calibrated.

### Probing movement.

**·icon· Probe side to be used.**

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



Calibration on the X+ side.



Calibration on the X- side.



Calibration on the Y+ side.



Calibration on the Y- side.

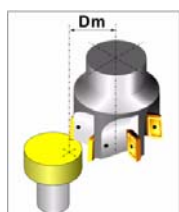
**·Ds· Safety distance.**

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

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

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

Optional parameter; by default the tool radius.

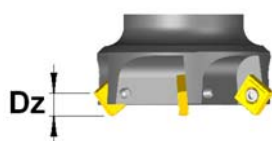


This parameter indicates the distance from the tool shaft to the tool probing point.

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

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

Optional parameter, by default 0.



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

**·F· Probing feedrate.**

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

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



CNC 8060  
CNC 8065

(REF: 1603)

**·S· Tool turning speed and direction.**

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

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

**·N· Number of edges to measure.**

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

This parameter indicates the number of cutting edges to be measured.

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

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

**Wear measurement.**

**·Lw· Maximum length wear allowed.**

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

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

**·Rw· Maximum radius wear allowed.**

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

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

**·icon· Behavior when exceeding the maximum of wear.**

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

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



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



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

**Probe coordinates.**

**·icon· Redefine the tabletop probe position.**



The probe position is assumed from the machine parameters.



The probe position is defined in the cycle.

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

**4.**

**CANNED CYCLES. CYCLE EDITOR.**  
Tool calibration (dimensions and wear)



CNC 8060  
CNC 8065

(REF: 1603)

**.PRB1MIN - PRB3MAX. Tabletop probe position.**

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

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

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

**4.****CANNED CYCLES. CYCLE EDITOR.**

Tool calibration (dimensions and wear)

**CNC 8060  
CNC 8065**

(REF: 1603)

## 4.2.2 Basic operation.

### Calibrate or measure tool length wear.

- 1 Approach movement.  
Rapid probe movement (G00) from the cycle calling point to the safety point. The safety point will be located at a distance  $\cdot D_s \cdot$  over the probe and at a distance  $\cdot D_m \cdot$  from the center of the tool to the point to be probed.  
This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.
- 2 Movement to locate an edge (only if  $\cdot N \cdot$  has been defined with a value other than 0).  
The CNC starts the spindle at an  $\cdot S \cdot$  speed and moves the tool at the feedrate  $\cdot F \cdot$  along the axis perpendicular to the active plane until receiving the probe signal. Once it knows the position of an edge, the tool withdraws to the safety point and stops the spindle.
- 3 Probing movement.  
Probing movement along the axis perpendicular to the active plane at the feedrate  $\cdot F \cdot$  until the probe signal is received. Withdrawal to the safety point.  
If  $\cdot N \cdot$  has been defined to a value other than 0, this movement is repeated  $\cdot N \cdot$  times.
- 4 Withdrawal movement.  
Movement in the work plane up to the cycle calling point.

### Calibrate or measure tool radius wear.

- 1 Approach movement.  
Rapid probe movement (G00) from the cycle calling point to the safety point. The safety point will be located at a  $\cdot D_s \cdot$  distance from the probe.  
This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.
- 2 Movement to locate an edge (only if  $\cdot N \cdot$  has been defined with a value other than 0).  
The CNC starts the spindle at an  $\cdot S \cdot$  speed and moves the tool at the feedrate  $\cdot F \cdot$  along the selected axis until receiving the probe signal. The CNC stops the spindle and carries out some probing moves to locate the edge more precisely. Once it knows the position of an edge, the tool withdraws to the safety point.
- 3 Probing movement.  
Probing movement at the indicated feedrate  $\cdot F \cdot$  until the probe signal is received. Withdrawal to the safety point.  
If  $\cdot N \cdot$  has been defined to a value other than 0, this movement is repeated  $\cdot N \cdot$  times.
- 4 Withdrawal movement.  
This movement is carried out in two stages. It first moves in the work plane and then along the perpendicular axis and then in the work plane.

### Calibrate or measure tool radius or length wear.

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

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

# 4.

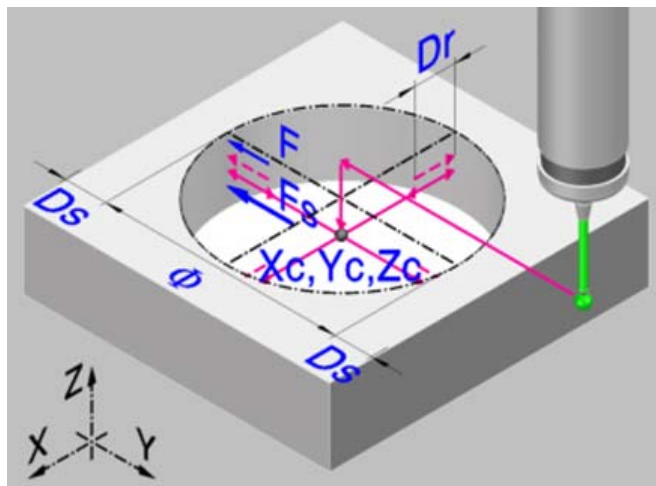
**CANNED CYCLES. CYCLE EDITOR.**  
Tool calibration (dimensions and wear)

**FAGOR** 

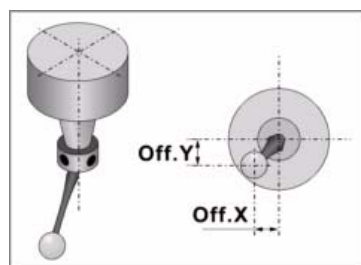
CNC 8060  
CNC 8065

(REF: 1603)

### 4.3 Measurement probe calibration.



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



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

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

#### Requirements prior to the calibration.

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

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

The following steps must be followed for calibrating the probe:

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

#### Data returned by the cycle after the measurement.

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

# 4.

**CANNED CYCLES. CYCLE EDITOR.**  
Measurement probe calibration.



CNC 8060  
CNC 8065

(REF: 1603)

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

P299 Best value of parameter PROBEDELAY for all the axes.

4.

**CANNED CYCLES. CYCLE EDITOR.**

Measurement probe calibration.



CNC 8060  
CNC 8065

(REF: 1603)

### 4.3.1 Programming the cycle.

# 4.

CANNED CYCLES. CYCLE EDITOR.  
Measurement probe calibration.

#### Probe data.

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

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

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

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

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

#### Probing movement.

**·Xc Yc· Real coordinates of the hole center.**

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

The canned cycles may be programmed in any work plane. See "[Canned cycles and the work planes.](#)" on page 93.

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

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

**·Ø· Real diameter of the hole.**

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

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

**·Ds· Safety distance.**

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

**·Dr· Withdrawal distance after initial probing.**

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

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

**·Fs· Feedrate for the first probing movement.**

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

**·F· Probing feedrate.**

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

#### Programming of M functions.

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

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



CNC 8060  
CNC 8065

(REF: 1603)



**·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.**

**CANNED CYCLES. CYCLE EDITOR.**

Measurement probe calibration.



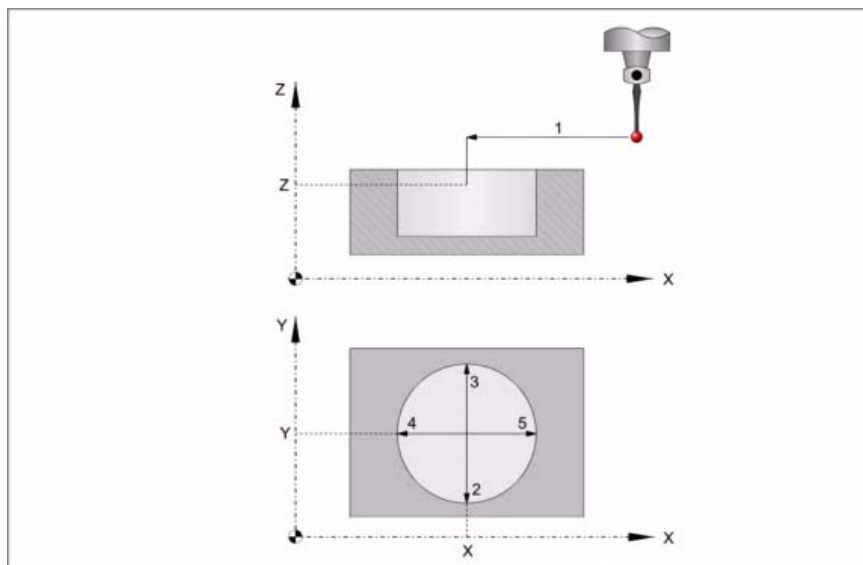
CNC 8060  
CNC 8065

(REF: 1603)

## 4.3.2 Basic operation.

## 4.

**CANNED CYCLES. CYCLE EDITOR.**  
Measurement probe calibration.



- 1 If the approach coordinates  $X_c$   $Y_c$   $Z_c$  have been programmed, approach movement. Probe's rapid movement (G00) from the cycle calling point to the center of the hole.

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

- 2 Probing of the first point

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

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

Withdrawal movement. Probe's rapid movement (G00) the distance indicated by  $\cdot D_r$ .

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

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

- 3 Probing of the second point.

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

- 4 Probing of the third point.

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

- 5 Probing of the fourth point.

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

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

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

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

- 7 Rotation of the probe to its previous position.

- 8 Withdrawal movement. Rapid probe movement (G00) from the center of the hole to the cycle calling point.

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

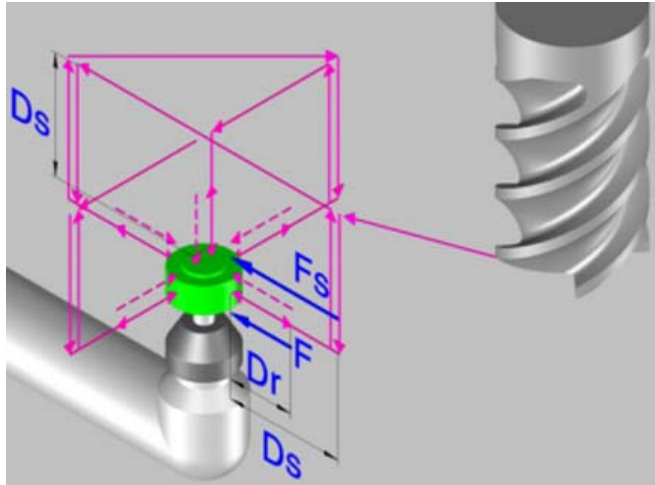
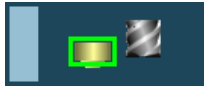


**FAGOR**

CNC 8060  
CNC 8065

(REF: 1603)

## 4.4 Tabletop probe calibration.



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

### Requirements prior to the calibration.

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

### Data returned by the cycle after the measurement.

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

- P295 Real coordinate of the probe side closest to the origin, along the abscissa axis.
- P296 Real coordinate of the probe side furthest from the origin, along the abscissa axis.
- P297 Real coordinate of the probe side closest to the origin, along the ordinate axis.
- P298 Real coordinate of the probe side furthest from the origin, along the ordinate axis.
- P299 Real coordinate of the probe side measured along the longitudinal axis.

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

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



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

# 4.

**CANNED CYCLES. CYCLE EDITOR.**  
Tabletop probe calibration.

**FAGOR** 

**CNC 8060  
CNC 8065**

(REF: 1603)

## 4.4.1 Programming the cycle.

# 4.

CANNED CYCLES. CYCLE EDITOR.  
Tabletop probe calibration.

### Tool data.

**·Tp· Tool number.**

Tool number. Optional parameter, if not defined or set to 0, use the tool that is in the spindle when executing the cycle.

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

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

### Probing movement.

**·Ds· Safety distance.**

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

**·Dr· Withdrawal distance after initial probing.**

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

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

**·Fs· Feedrate for the first probing movement.**

Optional parameter; by default the lowest value of machine parameter PROBEFEED of the axes that make up the trihedron.

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

**·F· Probing feedrate.**

Optional parameter; by default 10% of the lowest value of machine parameter PROBEFEED of the axes that make up the trihedron.

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

**·icon· Probing type.**



Simple calibration. The probe calibrates each side of the probe.



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

### Probe coordinates.

**·icon· Redefine the tabletop probe position.**



The probe position is assumed from the machine parameters.



The probe position is defined in the cycle.

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

**·PRB1MIN - PRB3MAX· Tabletop probe position.**

They are optional parameters that usually need not be defined. On certain machines, due to lack of repeatability in the mechanical positioning of the probe, the probe must be

calibrated again before each tool calibration. Instead of re-defining the machine parameters every time the probe is calibrated, those coordinates may be indicated in these parameters.

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

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

## Programming of M functions.

### **.M before. M functions to be executed before the cycle.**

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

### **.M after. M functions to be executed after the cycle.**

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




---

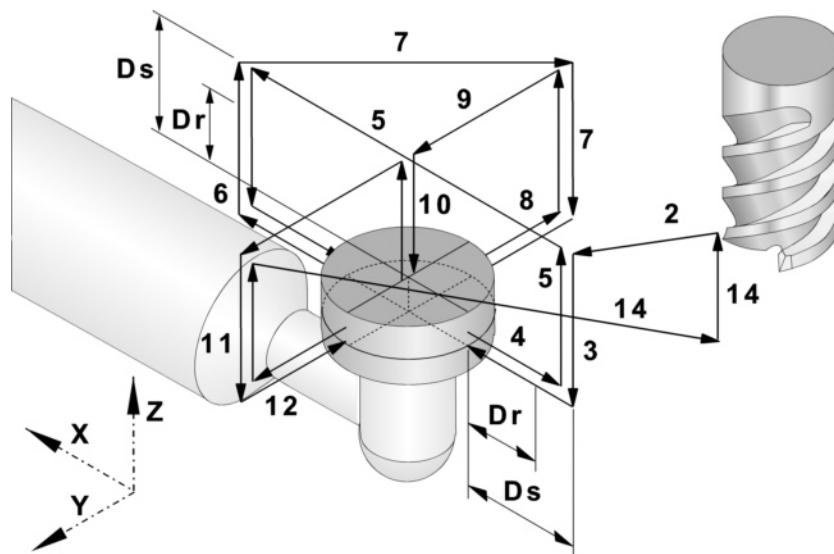
*We recommend using these functions, for example, to manage wireless probes. Wireless probes are not always active, they have to be turned on before using the probing cycles and turned off afterwards. For this type of probes, set an M function to turn the probe on and another one to turn it off. Having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

---

4.

**CANNED CYCLES. CYCLE EDITOR.**  
 Tabletop probe calibration.

## 4.4.2 Basic operation.

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

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

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

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

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

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

**Basic operation.**

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

This approach movement is made in two stages. If the tool is separated from the Z side of the probe a shorter distance than the one indicated in parameter ·Ds·, the movement is carried out first along the perpendicular axis and then in the work plane. Otherwise, it first moves in the work plane and then along the axis perpendicular to the plane.

- 3 Approach movement. Tool movement along the perpendicular axis at feedrate ·Fs· up to the probing Z coordinate (in the middle of the probe). If the tool touches the probe, the cycle will display the corresponding error message.

4.

CANNED CYCLES. CYCLE EDITOR.  
Tabletop probe calibration.

**FAGOR** 

CNC 8060  
CNC 8065

(REF: 1603)

- 4 Probing of the first point
 

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

First probing movement. Tool movement along the abscissa axis at feedrate  $\cdot F_s \cdot$  until the probe signal is received. Moving direction according to initial considerations.

Withdrawal movement. Return of the tool in rapid (G00) the distance indicated in  $\cdot D_r \cdot$ .

Second probe. Tool movement at the indicated feedrate  $\cdot F \cdot$  until the probe signal is received.

Withdrawal movement. Rapid tool movement (G00) from the probing point to the safety position located at a  $\cdot D_s \cdot$  distance from the probe.
- 5 Approach movement to the second probing point.
 

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

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

For the movement, the cycle takes into consideration the theoretical length of the probe and parameter  $\cdot D_s \cdot$ . If in this last movement, the tool touches the probe, the tool goes back up to the approach coordinate and moves the safety distance  $\cdot D_s \cdot$  in the same direction. The cycle repeats this operation until the tool releases the probe.
- 6 Probing of the second point.
 

It is done along the abscissa axis same as the previous one.
- 7 Approach movement to the third probing point.
 

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

It is done along the ordinate axis same as the previous one.
- 9 Approach movement to the fourth approach point (top surface).
 

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

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

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

First probing movement. Tool movement along the perpendicular axis at feedrate  $\cdot F_s \cdot$  until the probe signal is received.

Withdrawal movement. Return of the tool in rapid (G00) the distance indicated in  $\cdot D_r \cdot$ .

Second probe. Tool movement at the indicated feedrate  $\cdot F \cdot$  until the probe signal is received.

Withdrawal movement. Rapid tool movement (G00) from the probing point to the safety position located at a  $\cdot D_s \cdot$  distance from the probe.
- 11 Approach movement to the fifth probing point.
 

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

The tool moves in rapid (G00) to the next approach point. The tool moves back down to the probing Z coordinate at feedrate  $\cdot F_s \cdot$ .

For the movement, the cycle takes into consideration the theoretical length of the probe and parameter  $\cdot D_s \cdot$ . If in this last movement, the tool touches the probe, the tool goes back up to the approach coordinate and moves the safety distance  $\cdot D_s \cdot$  in the same direction. The cycle repeats this operation until the tool releases the probe.
- 12 Probing of the fifth point.
 

It is done along the ordinate axis same as the third probing point.
- 13 Withdrawal movement.
 

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

4.

CANNED CYCLES: CYCLE EDITOR.  
Tabletop probe calibration.

FAGOR 

CNC 8060  
CNC 8065

(REF: 1603)

**14** Withdrawal movement.

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

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

**15** If the cycle is going to do a double calibration, the spindle is oriented to 180° and the cycle repeats steps 2 through 14.**4.****CANNED CYCLES. CYCLE EDITOR.**

Tabletop probe calibration.

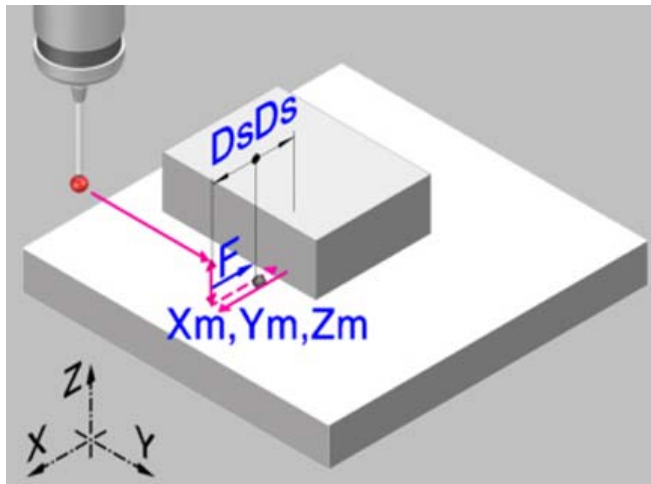


CNC 8060  
CNC 8065

(REF: 1603)



## 4.5 Surfacing measuring.



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

The probing movement may be programmed in the three axes of the active channel.

### Data returned by the cycle after the measurement.

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

- P298 Actual (real) surface coordinate.  
 P299 Detected error. Difference between the actual surface coordinate and the programmed theoretical coordinate.

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

Depending on the axis used for measuring, the cycle will correct either the tool length value or the tool radius value.

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

# 4.

**CANNED CYCLES. CYCLE EDITOR.**  
Surfacing measuring.

**FAGOR** 

CNC 8060  
CNC 8065

(REF: 1603)

## 4.5.1 Programming the cycle.

### Probe data.

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

Number of the tool used to define the probe in the tool table. Optional parameter, if not defined or set to 0, use the probe that is in the spindle when executing the cycle.

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

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

### Probing movement.

**·icon· Probing axis.**

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



Probing with the abscissa axis.



Probing with the ordinate axis.



Probing with the axis perpendicular to the plane.

**·icon· Cycle ending point (calling point or measured point).**

This parameter indicates where the probing cycle ends.



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



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

**·Xm Ym Zm· Theoretical coordinates of the probing point.**

Optional parameter; if any of them is not programmed, the cycle assumes the current position of the axis as theoretical coordinate of the probing point. The axes that are not programmed do not execute the positioning movement before probing.

Xm Theoretical coordinate of the probing point along the abscissa axis.

Ym Theoretical coordinate of the probing point along the ordinate axis.

Zm Theoretical coordinate of the probing point along the axis perpendicular to the plane.

The canned cycles may be programmed in any work plane.

**·icon· Moving direction for axis that has not been programmed.**

When not programming the theoretical coordinate of the probing point on some axis, the cycle shows an icon to define the moving direction of that axis.



The probe moves in the positive direction of the X axis.



The probe moves in the negative direction of the X axis.



The probe moves in the positive direction of the Y axis.



The probe moves in the negative direction of the Y axis.



The probe moves in the positive direction of the Z axis.



The probe moves in the negative direction of the Z axis.

# 4.

CANNED CYCLES. CYCLE EDITOR.  
Surfacing measuring.



CNC 8060  
CNC 8065

(REF: 1603)

**·Ds· Safety distance.**

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

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

**·F· Probing feedrate.**

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

**Wear measurement.**

Tool wear correction is optional. If it is activated, the correction only takes place when the measuring error exceeds the programmed value.



**Wear correction active.**

No tool wear correction is applied if this box is not selected.

**·T· Tool to be corrected.**

Optional parameter; by default, undefined. If T=0 (or not programmed), the CNC interprets that it is the tool used for machining.

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

**·D· Tool offset to be corrected.**

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

**·WT· Tolerance for the measuring error.**

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

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

**Save the result of the measurement.**

**·icon· Save the result of the measurement in a zero offset G159.**

This parameter indicates whether or not the result of the measurement will be saved in a zero offset G159. Regardless of the selected option, the cycle always saves the result of the measurements in the corresponding arithmetic parameters.



Save the result of the measurement in a zero offset G159.



Do not save the result of the measurement in a zero offset G159.

**·OffsetG159· Zero offset number**

Number of the zero offset into which to save the result of a measurement. If it is set to 0, the cycle does not save any data into the zero offset.

**Programming of M functions.**

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

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

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

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



---

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

---

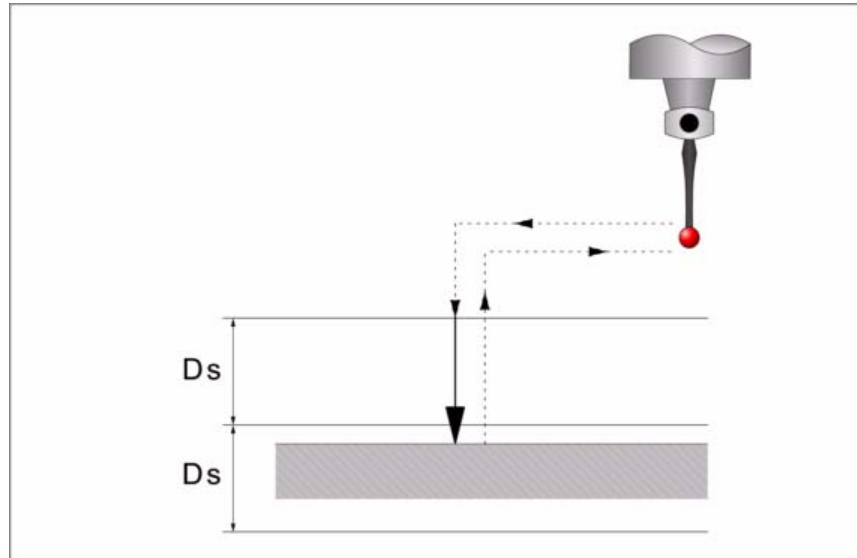
**4.****CANNED CYCLES. CYCLE EDITOR.**

Surfacing measuring.

**CNC 8060  
CNC 8065**

(REF: 1603)

## 4.5.2 Basic operation.



- 1 The cycle executes the "M-before" functions.
- 2 Approach movement (if any of parameters ·XYZ· has been defined).  
Rapid probe movement (G00) from the cycle calling point to the approach point located at a distance ·Ds· from the probing point and along the probing axis.  
This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.
- 3 Probing movement.  
Probing movement along the selected axis at the indicated feedrate ·F· until the probe signal is received. The maximum probing distance is ·2Ds·. If once this distance has been reached, the CNC has not yet received the probe signal, it will issue the relevant error code and stop the movement of the axes.  
Once probing is over, the CNC will assume the actual position of the axes when the probe signal is received as their theoretical position.  
Withdrawal movement. Rapid probe movement (G00) from the probing point to the approach point.
- 4 Withdrawal movement.  
Rapid probe movement (G00) from the probing point to the cycle calling point.  
It first moves along the perpendicular axis up to the coordinate of the cycle calling point along that axis. If programmed, it makes a movement in the main plane to the cycle calling point.
- 5 If programmed, the cycle saves the result of the measurement in the selected G159 zero offset and in the corresponding arithmetic parameters.
- 6 The cycle executes the "M-after" functions.

# 4.

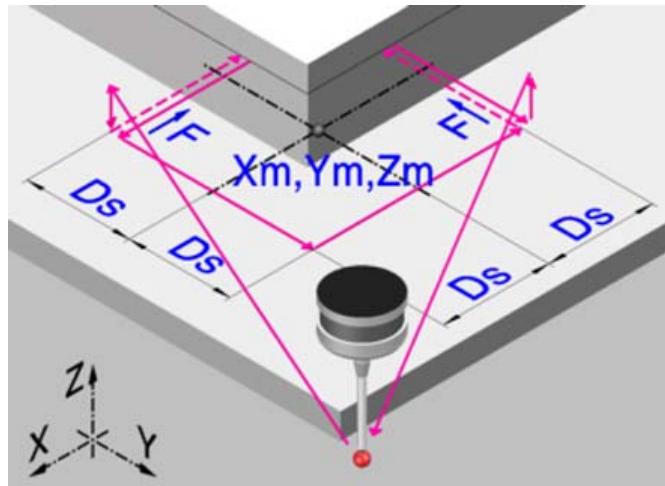
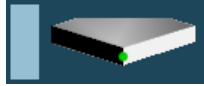
**CANNED CYCLES. CYCLE EDITOR.**  
Surfacing measuring.

**FAGOR** 

**CNC 8060**  
**CNC 8065**

(REF: 1603)

## 4.6 Outside corner measuring.



This cycle calculates the position of an outside corner.

### Data returned by the cycle after the measurement.

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

- P296 Real coordinate of the corner along the abscissa axis.
- P297 Real coordinate of the corner along the ordinate axis.
- P298 Error detected along the abscissa axis. Difference between the real coordinate of the corner and the theoretical programmed coordinate.
- P299 Error detected along the ordinate axis. Difference between the real coordinate of the corner and the theoretical programmed coordinate.

# 4.

**CANNED CYCLES. CYCLE EDITOR.**  
Outside corner measuring.



CNC 8060  
CNC 8065

(REF: 1603)

## 4.6.1 Programming the cycle.

### Probe data.

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

Number of the tool used to define the probe in the tool table. Optional parameter, if not defined or set to 0, use the probe that is in the spindle when executing the cycle.

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

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

### Probing movement.

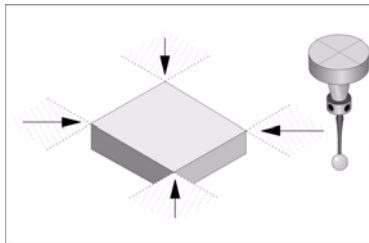
**·Xm Ym Zm. Theoretical coordinates of the corner to be measured.**

Xm Theoretical coordinate of the corner along the abscissa axis.

Ym Theoretical coordinate of the corner along the ordinate axis.

Zm Theoretical coordinate of the corner along the axis perpendicular to the plane.

The canned cycles may be programmed in any work plane. See "[Canned cycles and the work planes.](#)" on page 93.



Depending on the corner of the part it is required to measure, the probe must be placed in the corresponding shaded area (see figure) before calling the cycle.

**·Ds. Safety distance.**

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

Distance with respect to the point to measure, to which the probe approaches in G00 before making the probing movement. The probe must be placed, with respect to the point to be measured, at a distance greater than this value when the cycle is called.

**·F. Probing feedrate.**

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

### Save the result of the measurement.

**·icon. Save the result of the measurement in a zero offset G159.**

This parameter indicates whether or not the result of the measurement will be saved in a zero offset G159. Regardless of the selected option, the cycle always saves the result of the measurements in the corresponding arithmetic parameters.



Save the result of the measurement in a zero offset G159.



Do not save the result of the measurement in a zero offset G159.

**·OffsetG159. Zero offset number**

Number of the zero offset into which to save the result of a measurement. If it is set to 0, the cycle does not save any data into the zero offset.

## Programming of M functions.

### **.M before** M functions to be executed before the cycle.

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

### **.M after** M functions to be executed after the cycle.

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



---

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

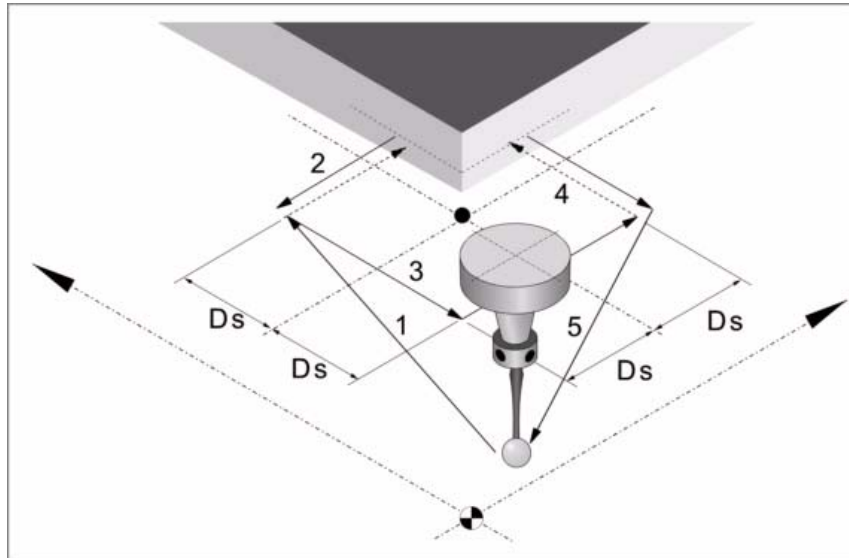
---

**4.****CANNED CYCLES. CYCLE EDITOR.**

Outside corner measuring.



## 4.6.2 Basic operation.



- 1 Approach movement to the first side to be probed.  
Rapid probe movement (G00) from the cycle calling point to the first approach point located at a  $\cdot D_s \cdot$  distance from the side to be probed.  
This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.
- 2 Probing on the first side.  
The probing on each side consists of a probing move and a withdrawal move.  
Probing movement. Probe movement along the abscissa axis at feedrate  $\cdot F \cdot$  until the probe signal is received. The maximum probing distance is  $\cdot 2D_s \cdot$ . If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.  
Withdrawal movement. Rapid probe movement (G00) from the probing point to the first approach point.
- 3 Approach movement to the second side to be probed.  
Rapid probe move (G00) from the first approach point to the second.  
This approach movement is made in two stages. It first moves in the work plane and then along the ordinate axis and then along the abscissa axis.
- 4 Probing on the second side.  
Probing movement. Probe movement along the ordinate axis at feedrate  $\cdot F \cdot$  until the probe signal is received. The maximum probing distance is  $\cdot 2D_s \cdot$ . If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes and displays the relevant error message.  
Withdrawal movement. Rapid probe movement (G00) from the probing point to the second approach point.
- 5 Withdrawal movement.  
Rapid probe movement (G00) from the second approach point to the cycle calling point.  
This withdrawal movement is made in two stages. It first moves in the work plane and then along the perpendicular axis and then in the work plane.

4.

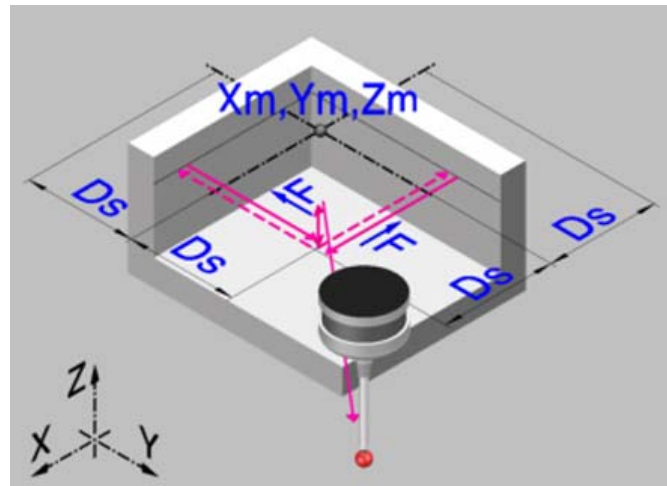
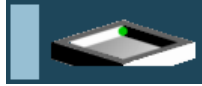
**CANNED CYCLES. CYCLE EDITOR.**  
Outside corner measuring.

**FAGOR** 

CNC 8060  
CNC 8065

(REF: 1603)

## 4.7 Inside corner measuring.



This cycle calculates the position of an inside corner.

### Data returned by the cycle after the measurement.

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

- P296 Real coordinate of the corner along the abscissa axis.
- P297 Real coordinate of the corner along the ordinate axis.
- P298 Error detected along the abscissa axis. Difference between the real coordinate of the corner and the theoretical programmed coordinate.
- P299 Error detected along the ordinate axis. Difference between the real coordinate of the corner and the theoretical programmed coordinate.

# 4.

**CANNED CYCLES. CYCLE EDITOR.**  
Inside corner measuring.



CNC 8060  
CNC 8065

(REF: 1603)

## 4.7.1 Programming the cycle.

### Probe data.

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

Number of the tool used to define the probe in the tool table. Optional parameter, if not defined or set to 0, use the probe that is in the spindle when executing the cycle.

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

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

### Probing movement.

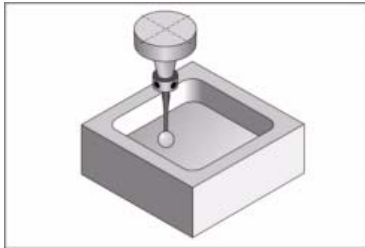
**·Xm Ym Zm· Theoretical coordinates of the corner to be measured.**

Xm Theoretical coordinate of the corner along the abscissa axis.

Ym Theoretical coordinate of the corner along the ordinate axis.

Zm Theoretical coordinate of the corner along the axis perpendicular to the plane.

The canned cycles may be programmed in any work plane. See "[Canned cycles and the work planes.](#)" on page 93.



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

**·Ds· Safety distance.**

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

Distance with respect to the point to measure, to which the probe approaches in G00 before making the probing movement. The probe must be placed, with respect to the point to be measured, at a distance greater than this value when the cycle is called.

**·F· Probing feedrate.**

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

### Save the result of the measurement.

**·icon· Save the result of the measurement in a zero offset G159.**

This parameter indicates whether or not the result of the measurement will be saved in a zero offset G159. Regardless of the selected option, the cycle always saves the result of the measurements in the corresponding arithmetic parameters.



Save the result of the measurement in a zero offset G159.



Do not save the result of the measurement in a zero offset G159.

**·OffsetG159· Zero offset number**

Number of the zero offset into which to save the result of a measurement. If it is set to 0, the cycle does not save any data into the zero offset.

# 4.

CANNED CYCLES. CYCLE EDITOR.  
Inside corner measuring.

**FAGOR** 

CNC 8060  
CNC 8065

(REF: 1603)

## 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.*

---

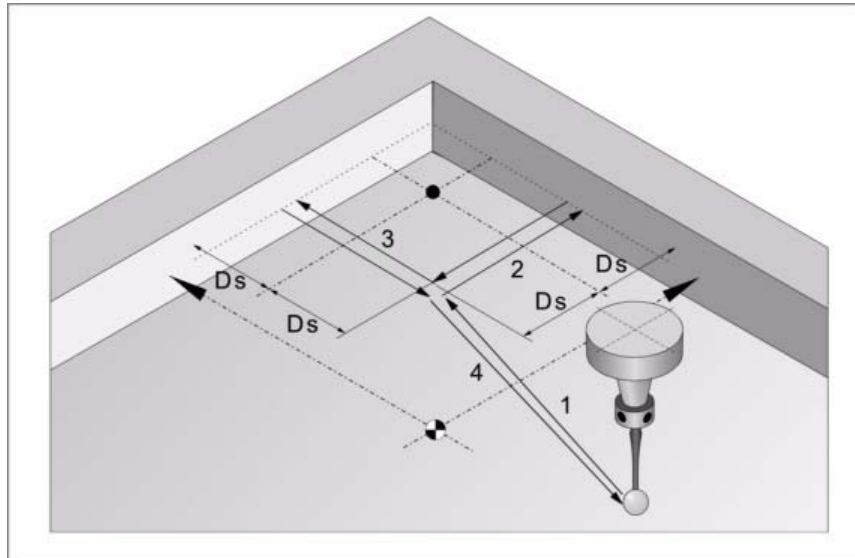
**4.****CANNED CYCLES. CYCLE EDITOR.**

Inside corner measuring.

**CNC 8060  
CNC 8065**

(REF: 1603)

## 4.7.2 Basic operation.



### 1 Approach movement.

Rapid probe movement (G00) from the cycle calling point to the first approach point located at a  $\cdot D_s \cdot$  distance from the two sides to be probed.

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

### 2 Probing on the first side.

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

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

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

### 3 Probing on the second side.

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

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

### 4 Withdrawal movement.

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

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

# 4.

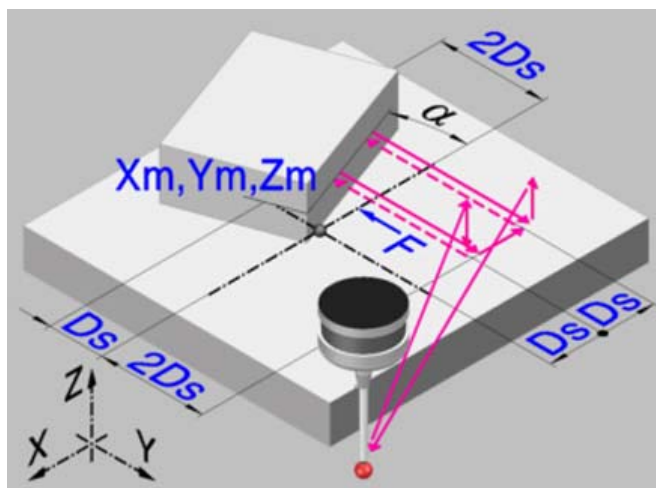
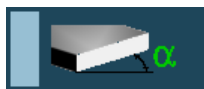
**CANNED CYCLES. CYCLE EDITOR.**  
Inside corner measuring.

**FAGOR** 

CNC 8060  
CNC 8065

(REF: 1603)

## 4.8 Measuring the angle with the abscissa axis.



This cycle calculates the angle of the part with respect to the abscissa axis. This cycle may be used to measure angles between  $\pm 45^\circ$ .

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

### Data returned by the cycle after the measurement.

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

P295      Inclination angle between the part and the abscissa axis.

# 4.

**CANNED CYCLES. CYCLE EDITOR.**

Measuring the angle with the abscissa axis.



CNC 8060  
CNC 8065

(REF: 1603)

## 4.8.1 Programming the cycle.

### Probe data.

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

Number of the tool used to define the probe in the tool table. Optional parameter, if not defined or set to 0, use the probe that is in the spindle when executing the cycle.

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

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

### Probing movement.

#### ·Xm Ym Zm· Theoretical coordinates of the corner to be measured.

Xm Theoretical coordinate of the corner along the abscissa axis.

Ym Theoretical coordinate of the corner along the ordinate axis.

Zm Theoretical coordinate of the corner along the axis perpendicular to the plane.

The canned cycles may be programmed in any work plane. See "[Canned cycles and the work planes.](#)" on page 93.

#### ·Ds· Safety distance.

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

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

#### ·F· Probing feedrate.

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

### Programming of M functions.

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

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

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

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



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

# 4.

**CANNED CYCLES. CYCLE EDITOR.**

Measuring the angle with the abscissa axis.

**FAGOR** 

CNC 8060  
CNC 8065

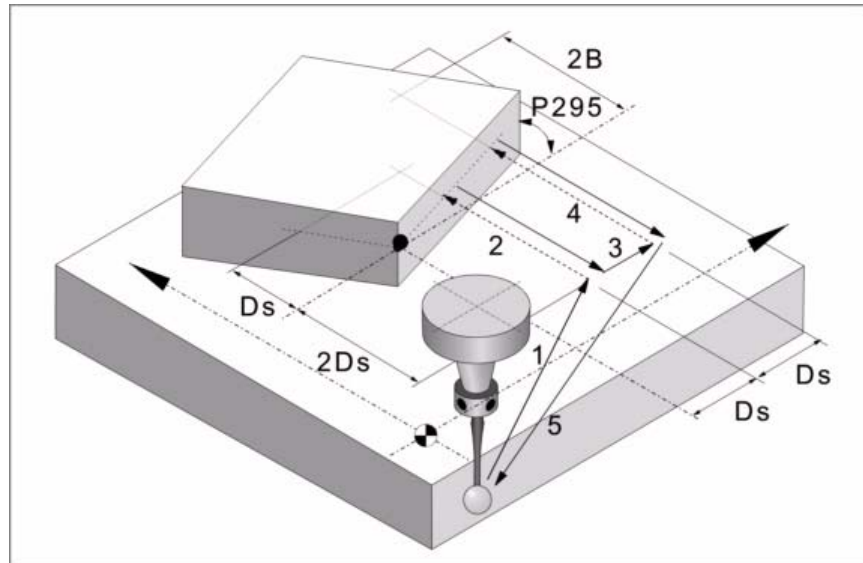
(REF: 1603)

## 4.8.2 Basic operation.

## 4.

## CANNED CYCLES. CYCLE EDITOR.

Measuring the angle with the abscissa axis.



## 1 First approach movement.

Rapid probe movement (G00) from the cycle calling point to the first approach point located at a distance  $\cdot 2D_s \cdot$  from the side to be probed.

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

## 2 First probing move.

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

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

## 3 Second approach movement.

Rapid probe move (G00) from the first approach point to the second. The second approach point is located at a distance  $\cdot D_s \cdot$  from the first one.

## 4 Second probe.

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

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

## 5 Withdrawal movement.

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

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

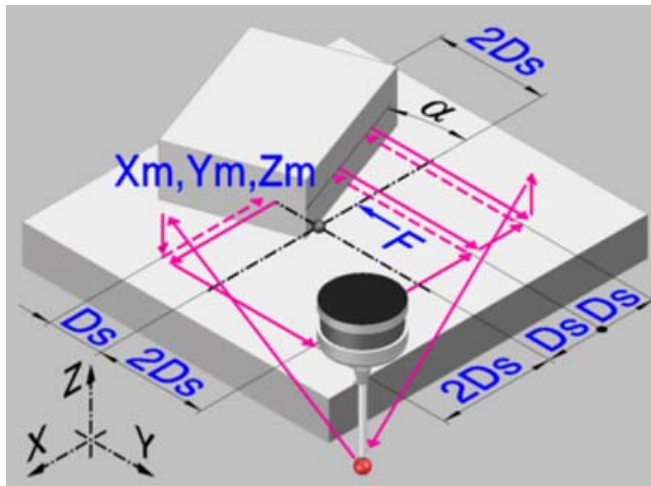
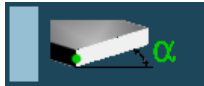


CNC 8060  
CNC 8065

(REF: 1603)



## 4.9 Outside corner and angle measuring.



This cycle calculates the position of an outside corner and the angle of the part with respect to the abscissa axis. This cycle may be used to measure angles between  $\pm 45^\circ$ .

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

### Data returned by the cycle after the measurement.

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

- |      |   |
|------|---|
| P295 | Inclination angle between the part and the abscissa axis.   |
| P296 | Real coordinate of the corner along the abscissa axis.  |
| P297 | Real coordinate of the corner along the ordinate axis.  |
| P298 | Error detected along the abscissa axis. Difference between the real coordinate of the corner and the theoretical programmed coordinate. |
| P299 | Error detected along the ordinate axis. Difference between the real coordinate of the corner and the theoretical programmed coordinate. |

# 4.

**CANNED CYCLES. CYCLE EDITOR.**  
Outside corner and angle measuring.

**FAGOR** 

CNC 8060  
CNC 8065

(REF: 1603)

## 4.9.1 Programming the cycle.

# 4.

CANNED CYCLES. CYCLE EDITOR.  
Outside corner and angle measuring.

### Probe data.

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

Number of the tool used to define the probe in the tool table. Optional parameter, if not defined or set to 0, use the probe that is in the spindle when executing the cycle.

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

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

### Probing movement.

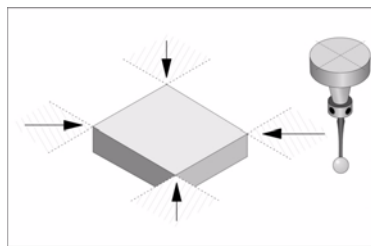
**·Xm Ym Zm· Theoretical coordinates of the corner to be measured.**

Xm Theoretical coordinate of the corner along the abscissa axis.

Ym Theoretical coordinate of the corner along the ordinate axis.

Zm Theoretical coordinate of the corner along the axis perpendicular to the plane.

The canned cycles may be programmed in any work plane. See "[Canned cycles and the work planes.](#)" on page 93.



Depending on the corner of the part it is required to measure, the probe must be placed in the corresponding shaded area (see figure) before calling the cycle.

**·Ds· Safety distance.**

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

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

**·F· Probing feedrate.**

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

### Save the result of the measurement.

**·icon· Save the result of the measurement in a zero offset G159.**

This parameter indicates whether or not the result of the measurement will be saved in a zero offset G159. Regardless of the selected option, the cycle always saves the result of the measurements in the corresponding arithmetic parameters.



Save the result of the measurement in a zero offset G159.



Do not save the result of the measurement in a zero offset G159.

**·OffsetG159· Zero offset number**

Number of the zero offset into which to save the result of a measurement. If it is set to 0, the cycle does not save any data into the zero offset.

## Programming of M functions.

### **.M before- M functions to be executed before the cycle.**

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

### **.M after- M functions to be executed after the cycle.**

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



---

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

---

4.

**CANNED CYCLES. CYCLE EDITOR.**

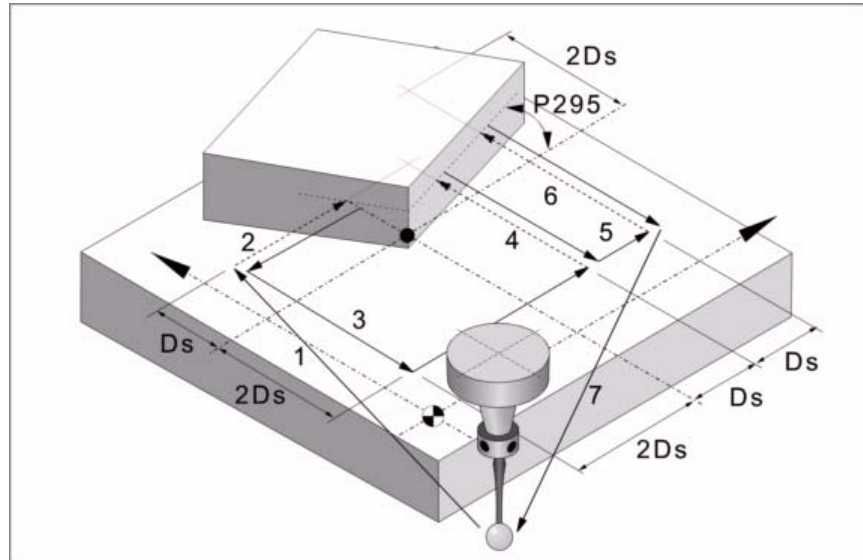
Outside corner and angle measuring.



CNC 8060  
CNC 8065

(REF: 1603)

## 4.9.2 Basic operation.



## 1 First approach movement.

Rapid probe movement (G00) from the cycle calling point to the first approach point located at a  $\cdot 2D_s$  distance from the side to be probed.

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

## 2 First probing move.

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

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

## 3 Second approach movement.

Rapid probe movement (G00) from the first approach point to the second, located at a  $\cdot 2D_s$  distance from the second side to be probed.

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

## 4 Second probe.

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

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

## 5 Third approach movement.

Rapid probe move (G00) from the second approach point to the third. It is located at a  $\cdot D_s$  distance from the previous one.

## 6 Third probing move.

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

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

4.

CANNED CYCLES. CYCLE EDITOR.  
Outside corner and angle measuring.

FAGOR 

CNC 8060  
CNC 8065

(REF: 1603)

7 Withdrawal movement.

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

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

4.

**CANNED CYCLES. CYCLE EDITOR.**

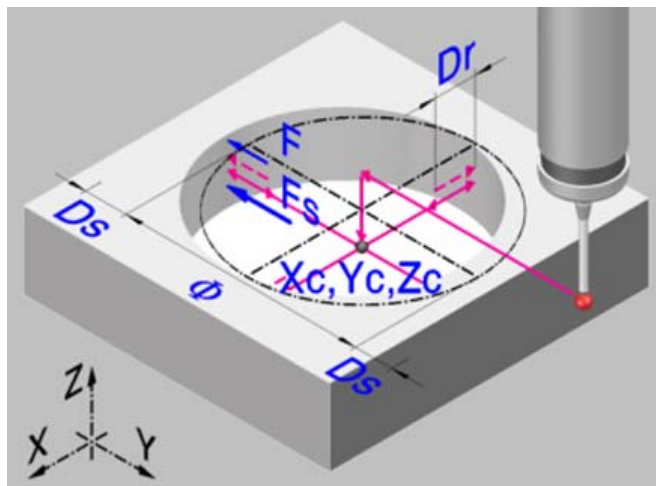
Outside corner and angle measuring.



CNC 8060  
CNC 8065

(REF: 1603)

## 4.10 Hole measuring.



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

### Data returned by the cycle after the measurement.

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

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

4.

CANNED CYCLES. CYCLE EDITOR.  
Hole measuring.



CNC 8060  
CNC 8065

(REF: 1603)

## 4.10.1 Programming the cycle.

### Probe data.

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

Number of the tool used to define the probe in the tool table. Optional parameter, if not defined or set to 0, use the probe that is in the spindle when executing the cycle.

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

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

### Probing movement.

#### **·icon· Cycle ending point (calling point or measured point).**

This parameter indicates where the probing cycle ends.



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



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

#### **·Xm Ym Zm· Theoretical coordinates of the hole center.**

Xm Theoretical hole center coordinate along the abscissa axis.

Ym Theoretical hole center coordinate along the ordinate axis.

Zm Theoretical hole center coordinate along the axis perpendicular to the plane.

The canned cycles may be programmed in any work plane. See "[Canned cycles and the work planes.](#)" on page 93.

#### **·Ø· Theoretical diameter of the hole.**

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

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

#### **·Ds· Safety distance.**

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

#### **·Dr· Withdrawal distance after initial probing.**

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

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

#### **·Fs· Feedrate for the first probing movement.**

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

#### **·F· Probing feedrate.**

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

# 4.

CANNED CYCLES: CYCLE EDITOR.  
Hole measuring.

**FAGOR** 

CNC 8060  
CNC 8065

(REF: 1603)

## Save the result of the measurement.

### ·icon· Save the result of the measurement in a zero offset G159.

This parameter indicates whether or not the result of the measurement will be saved in a zero offset G159. Regardless of the selected option, the cycle always saves the result of the measurements in the corresponding arithmetic parameters.



Save the result of the measurement in a zero offset G159.



Do not save the result of the measurement in a zero offset G159.

### ·OffsetG159· Zero offset number

Number of the zero offset into which to save the result of a measurement. If it is set to 0, the cycle does not save any data into the zero offset.

## Programming of M functions.

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

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

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

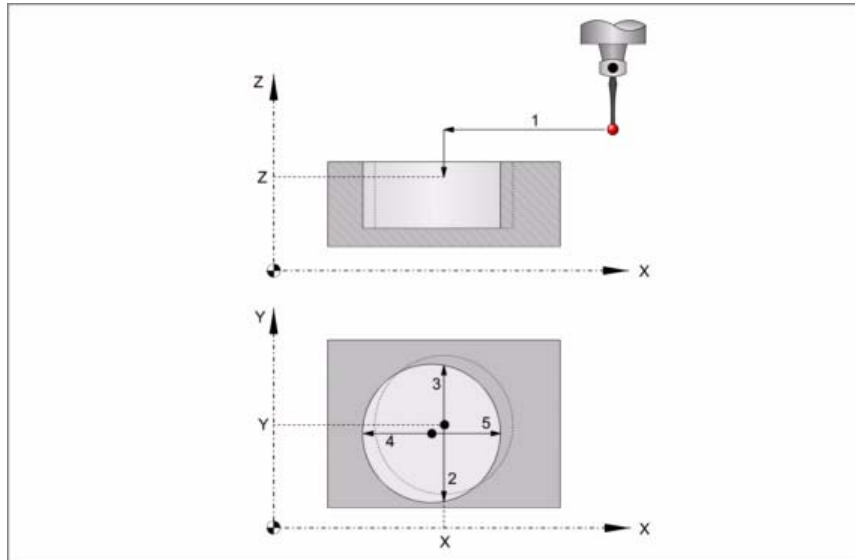
The cycle allows executing up to 4 M functions after the cycle. To execute only some of them, define them first and leave the rest unprogrammed.



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



## 4.10.2 Basic operation.



## 1 Approach movement.

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

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

## 2 Probing of the first point

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

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

Withdrawal movement. Probe's rapid movement (G00) the distance indicated by  $\cdot Dr \cdot$ .

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

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

## 3 Probing of the second point.

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

## 4 Probing of the third point.

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

## 5 Probing of the fourth point.

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

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

## 6 Withdrawal movement (if it has been defined).

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

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

4.

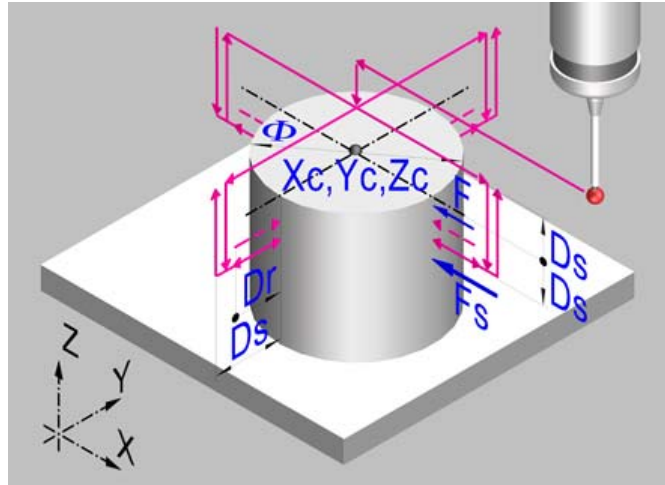
CANNED CYCLES. CYCLE EDITOR.  
Hole measuring.

FAGOR 

CNC 8060  
CNC 8065

(REF: 1603)

## 4.11 Circular boss measuring.



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

### Data returned by the cycle after the measurement.

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

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

# 4.

CANNED CYCLES. CYCLE EDITOR.  
Circular boss measuring.



CNC 8060  
CNC 8065

(REF: 1603)

### 4.11.1 Programming the cycle.

#### Probe data.

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

Number of the tool used to define the probe in the tool table. Optional parameter, if not defined or set to 0, use the probe that is in the spindle when executing the cycle.

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

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

#### Probing movement.

**·icon· Cycle ending point (calling point or measured point).**

This parameter indicates where the probing cycle ends.



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



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

**·Xm Ym Zm· Theoretical coordinates of the boss center.**

Xm Theoretical boss center coordinate along the abscissa axis.

Ym Theoretical boss center coordinate along the ordinate axis.

Zm Theoretical boss center coordinate along the axis perpendicular to the plane.

The canned cycles may be programmed in any work plane. See "[Canned cycles and the work planes.](#)" on page 93.

**·Ø· Theoretical diameter of the boss.**

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

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

**·Ds· Safety distance.**

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

**·Dr· Withdrawal distance after initial probing.**

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

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

**·Fs· Feedrate for the first probing movement.**

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

**·F· Probing feedrate.**

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

# 4.

**CANNED CYCLES. CYCLE EDITOR.**  
Circular boss measuring.

**FAGOR** 

**CNC 8060  
CNC 8065**

(REF: 1603)

## Save the result of the measurement.

### ·icon· Save the result of the measurement in a zero offset G159.

This parameter indicates whether or not the result of the measurement will be saved in a zero offset G159. Regardless of the selected option, the cycle always saves the result of the measurements in the corresponding arithmetic parameters.



Save the result of the measurement in a zero offset G159.



Do not save the result of the measurement in a zero offset G159.

### ·OffsetG159· Zero offset number

Number of the zero offset into which to save the result of a measurement. If it is set to 0, the cycle does not save any data into the zero offset.

## Programming of M functions.

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

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

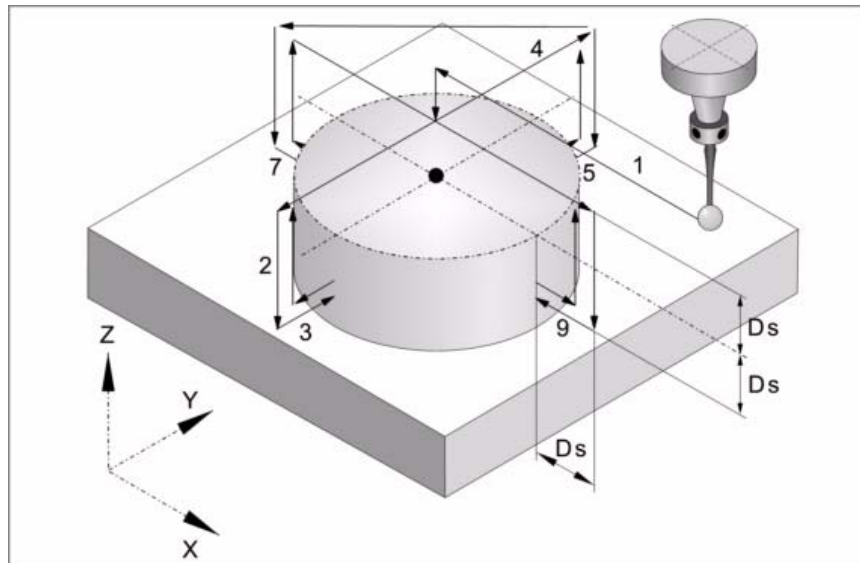
### ·M after· M functions to be executed after the cycle.

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



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

### 4.11.2 Basic operation.



**1** Positioning over the boss center.

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

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

**2** First approach movement.

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

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

**3** Probing of the first point

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

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

Withdrawal movement. Probe's rapid movement (G00) the distance indicated by  $\cdot D_r \cdot$ .

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

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

**4** Second approach movement.

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

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

**5** Probing of the second point.

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

**6** Third approach movement.

It is done like the previous one.

**7** Probing of the third point.

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

**8** Fourth approach movement.

It is done like the previous one.

# 4.

**CANNED CYCLES. CYCLE EDITOR.**  
Circular boss measuring.



CNC 8060  
CNC 8065

(REF: 1603)

**9** Probing of the fourth point.

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

The probe moves in rapid (G00) at a distance  $\cdot D_s \cdot$  over the boss up to the calculated real boss center.

**10** Withdrawal movement (if it has been defined).

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

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

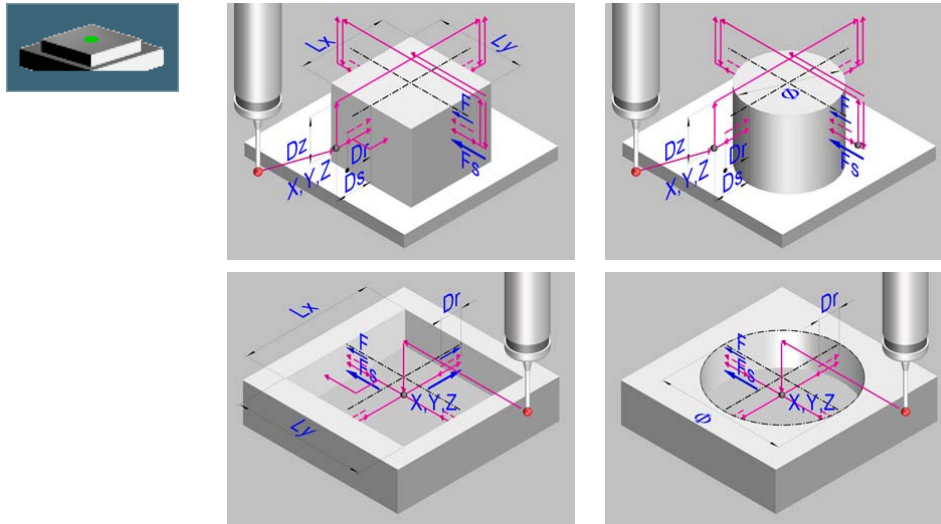
**4.****CANNED CYCLES. CYCLE EDITOR.**

Circular boss measuring.

**CNC 8060  
CNC 8065**

(REF: 1603)

## 4.12 Rectangular or circular part/pocket centering.



This cycle may be used to calculate the center of a rectangular or circular part or pocket of known dimensions as well as, in rectangular parts, the inclination of the part with respect to the abscissa axis.

### Requirements prior to the calibration.

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

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

### Data returned by the cycle after the measurement.

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

- P296 Inclination angle between the part and the abscissa axis (rectangular part).
- P297 Part surface coordinate.
- P298 Part center coordinate along the abscissa axis.
- P299 Part center coordinate along the ordinate axis.

Optionally, in this cycle, it is possible to preset the coordinates to select a new part zero and, in rectangular parts, rotate the coordinate axes to align the axes with the part.

4.

**CANNED CYCLES. CYCLE EDITOR.**  
Rectangular or circular part/pocket centering.

**FAGOR** 

CNC 8060  
CNC 8065

(REF: 1603)

### 4.12.1 Programming the cycle.

# 4.

CANNED CYCLES. CYCLE EDITOR.  
Rectangular or circular part/pocket centering.

#### Probe data.

**.Tp- Number of the tool that identifies the probe.**

Number of the tool used to define the probe in the tool table. Optional parameter, if not defined or set to 0, use the probe that is in the spindle when executing the cycle.

**.Dp- Number of the tool offset that identifies the probe.**

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

#### Probing movement.

**.X Y Z- Probe position to execute the cycle.**

X Probe position along the abscissa axis

Y Probe position along the ordinate axis.

Z Probe position along the axis perpendicular to the plane.

The canned cycles may be programmed in any work plane.

**.icon- Geometry of the part to be centered.**

This parameter indicates the type of part/pocket to be centered.



Rectangular part centering.



Circular part centering.



Rectangular pocket centering.



Circular pocket centering.

**.icon- Axis and direction of the first probing movement.**

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



The probe moves in the positive direction of the X axis.



The probe moves in the negative direction of the X axis.



The probe moves in the positive direction of the Y axis.



The probe moves in the negative direction of the Y axis.

**.icon- Center the part/pocket on one or two axes with one or two probeings on the first side.**

This parameter indicates on how many axes the part is centered and the number of probing movements to carry out on the first side.



Rectangular part. Center the part on one axis.



Rectangular part. Center the part on both axes, with one probing on the first side.



Rectangular part. Center the part on both axes, with two probeings on the first side.



Circular part. Center the part on one axis.



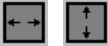


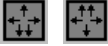


Circular part. Center the part on both axes, with one probing on the first side.



CNC 8060  
CNC 8065



(REF: 1603)



-  Rectangular pocket. Center the pocket on one axis.
-  Rectangular pocket. Center the pocket on both axes, with one probing on the first side.
-  Rectangular pocket. Center the pocket on both axes, with two probings on the first side.
- 
-  Circular pocket. Center the pocket on one axis.
-  Circular pocket. Center the pocket on both axes, with one probing on the first side.




**.icon- Surface coordinate measuring.**

This parameter indicates whether the cycle must also measure the position of the top surface of the part. This icon is only available when the part is centered on two axes.

-  The cycle does not measure the coordinate of the boss surface or of the bottom of the pocket.
-  The cycle measures the coordinate of the boss surface or of the bottom of the pocket.



**.icon- Preset the part zero after ending the cycle.**

This parameter indicates whether the part zero is to be preset or not, if so, the point taken as reference. This point may be preset with any value using parameters .Px Py Pz.

-  Do not preset the part zero.
-  Preset the part zero at the center of the part.
-  Preset the part zero at one of the corners (the cycle shows an icon for each corner).

**.icon- Pattern rotation.**

For rectangular parts, this parameter indicates whether a coordinate rotation is to be applied or not with the measured angle. This icon is only available when the part is centered on two axes, with two probings on the first side.

-  Do not rotate the coordinates (pattern).
-  Rotate the coordinates (pattern).

**.Lx Ly Ø- Part dimensions.**

These parameters establish the dimensions of the rectangular or circular part.

- Lx Rectangular part. Approximate part length along the abscissa axis.
- Ly Rectangular part. Approximate part length along the ordinate axis.
- Ø Circular part. Part diameter.

**.Ds- Safety distance.**

Optional parameter; if not defined, it assumes the distance between the part and the probe position when calling the cycle.

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

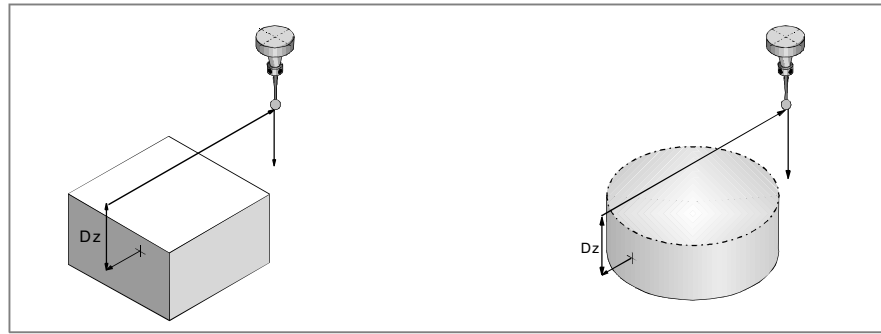
**.Dr- Withdrawal distance after initial probing.**

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

## 4.

## CANNED CYCLES. CYCLE EDITOR.

Rectangular or circular part/pocket centering.

**.Dz- Z safety distance from the probing coordinate (position).**

Distance for the probe to go up from the probing coordinate (position) for its movements over the part. This parameter must have enough value to prevent the probe from colliding when moving over the part.

**.Fs- Feedrate for the first probing movement.**

Optional parameter; by default the lowest value of machine parameter PROBEFEED of the axes that make up the trihedron.

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

**.F- Probing feedrate.**

Optional parameter; by default 10% of the lowest value of machine parameter PROBEFEED of the axes that make up the trihedron.

This parameter sets the feedrate for the second probing movement.

**.icon- Feedrate for approach movements.**

This parameter sets the type of feedrate for the movements to the approach points.



The movements are carried out in rapid.



The movements are carried out at work feedrate.

**Coordinate preset value.****.Px Py Pz- Coordinate preset value on each axis.**

Px Preset value on the abscissa axis.

Py Preset value on the ordinate axis.

Pz Preset value on the axis perpendicular to the plane.

These parameters are only valid when presetting the part zero and any this point may be assigned any value.

## Save the result of the measurement.

### ·icon· Save the result of the measurement in a zero offset G159.

This parameter indicates whether or not the result of the measurement will be saved in a zero offset G159. Regardless of the selected option, the cycle always saves the result of the measurements in the corresponding arithmetic parameters.



Save the result of the measurement in a zero offset G159.



Do not save the result of the measurement in a zero offset G159.

### ·OffsetG159· Zero offset number

Number of the zero offset into which to save the result of a measurement. If it is set to 0, the cycle does not save any data into the zero offset.

## Programming of M functions.

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

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

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

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



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

# 4.

**CANNED CYCLES. CYCLE EDITOR.**

Rectangular or circular part/pocket centering.

**FAGOR** 

CNC 8060  
CNC 8065

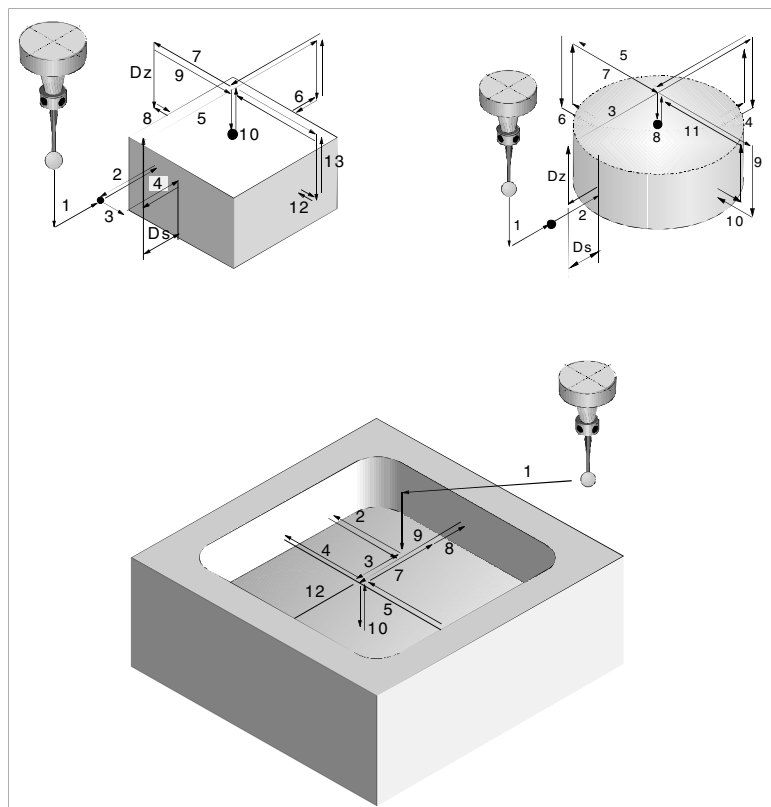
(REF: 1603)

## 4.12.2 Basic operation.

4.

## CANNED CYCLES. CYCLE EDITOR.

Rectangular or circular part/pocket centering.



- 1 The cycle executes the "M-before" functions.
- 2 Approach movement (if any of parameters  $\cdot XYZ \cdot$  has been defined).  
Movement of the probe at the selected feedrate from the cycle calling point to the first approach point defined by parameters  $\cdot XYZ \cdot$ .  
This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.
- 3 Probing of the first point  
The cycle makes a first probing movement, a withdrawal movement and a second probing move. After the second probing move, the probe withdraws to the approach point.  
First probing movement. Probe movement along the selected axis at feedrate  $\cdot Fs \cdot$  until the probe signal is received.  
Withdrawal movement. Return of the probe in rapid (G00) the distance indicated in  $\cdot Dr \cdot$ .  
Second probe. Probing movement at feedrate  $\cdot F \cdot$  until the probe signal is received.  
Withdrawal movement. Rapid probe movement (G00) from the probing point to the starting point.
- 4 If the part is rectangular and probing on two points has been programmed, repeat the probing in a different point of the surface. The cycle uses this second probing to calculate the inclination of the part.  
Approach movement. Movement parallel to the probed side to touch a different point of the same side, located at  $1/4$  the length programmed in  $\cdot Ly \cdot$ .  
Probing movement. Probing movement at feedrate  $\cdot F \cdot$  until the probe signal is received.  
Withdrawal movement. The probe returns to the safety position located at a  $\cdot Ds \cdot$  distance from the part.

CNC 8060  
CNC 8065

(REF: 1603)

- 5 Approach movement to the second probing point.  
 Movement of the probe from the first approach point to the second located in front of the first one.  
 The perpendicular axis moves the  $\cdot Dz \cdot$  distance up in rapid (G00). Then the probe moves over the part, at the selected feedrate, to the next approach point. The probe moves a  $\cdot Dz \cdot$  distance back down at feedrate  $\cdot Fs \cdot$ .  
 For the movement, the cycle takes into consideration the dimensions of the part and, if it is rectangular, its inclination angle. If in this last movement, the probe touches the part, the CNC shows an error at the cycle parameter value.
- 6 Probing of the second point.  
 It is done like the previous one. The probe returns to the safety position located at a  $\cdot Ds \cdot$  distance from the part.
- 7 If part centering on two axes has been programmed, approach movement to the third probing point.  
 Movement of the probe from the second approach point to the center of the part and then to the third point.
- 8 If part centering on two axes has been programmed, probing of the third point.  
 It is done like the previous one.
- 9 Approach movement to the top surface (only if it has been programmed).  
 When selecting to measure the surface, the probe moves the  $\cdot Dz \cdot$  distance up in rapid (G00) and the probe moves to the center of the part.
- 10 Measuring the top surface of the boss or the bottom of the pocket (only if it has been programmed).  
 The probe makes a first probing move, a withdrawal movement and a second probing move. After the second probing move, the probe withdraws to the approach point.  
 First probing movement. Probing movement, at feedrate  $\cdot Fs \cdot$ , until touching the part surface.  
 Withdrawal movement. Return of the probe in rapid (G00) the distance indicated in  $\cdot Dr \cdot$ .  
 Second probe. Probing movement at feedrate  $\cdot F \cdot$  until the probe signal is received.  
 Withdrawal movement. Rapid probe movement (G00) from the probing point to the safety position.
- 11 If part centering on two axes has been programmed, approach movement to the fourth probing point.  
 Movement of the probe from the third approach point to the fourth.
- 12 If part centering on two axes has been programmed, probing of the four point.  
 It is done like the previous one.
- 13 Probe positioning at the calculated center.  
 This movement is carried out in two stages. It first moves along the perpendicular axis in rapid (G00) and then in the plane.
- 14 If programmed, the cycle saves the result of the measurement in the selected G159 zero offset and in the corresponding arithmetic parameters.
- 15 The cycle executes the "M-after" functions.

4.

CANNED CYCLES. CYCLE EDITOR.

Rectangular or circular part/pocket centering.



CNC 8060  
CNC 8065

(REF: 1603)

## 4.13 Simulating a cycle from the editor.

At the canned cycle editor, it is possible to simulate the cycle being edited without having to simulate the whole part-program. During simulation, another canned cycle may be viewed and edited and it is also possible to return to the program editor.



*If the cycle editor is included in the automatic operating mode, it will not be possible to simulate a cycle.*

# 4.

**CANNED CYCLES. CYCLE EDITOR.**  
Simulating a cycle from the editor.

### 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 area 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, first press [CTRL]+[F2] (window change)



CNC 8060  
CNC 8065

(REF: 1603)

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.

4.

**CANNED CYCLES. CYCLE EDITOR.**

Simulating a cycle from the editor.



CNC 8060  
CNC 8065

(REF: 1603)

# 4.

## **CANNED CYCLES. CYCLE EDITOR.**

Simulating a cycle from the editor.



CNC 8060  
CNC 8065

(REF: 1603)

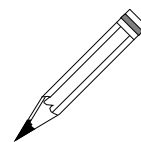




**FAGOR** 

**CNC 8060  
CNC 8065**

(REF: 1603)



CNC 8060  
CNC 8065

(REF: 1603)