

**CNC**

**8060**

**8065**

Operating manual (TC).

(Ref: 1709)



**FAGOR AUTOMATION**

---

## TRANSLATION OF THE ORIGINAL MANUAL

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

---

## MACHINE SAFETY

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

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

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

---

## HARDWARE EXPANSIONS

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

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

---

## COMPUTER VIRUSES

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

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

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

---

## DUAL-USE PRODUCTS

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



FAGOR AUTOMATION

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

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

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

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

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

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

# INDEX

About the product - CNC 8060 .....	7
About the product - CNC 8065 .....	11
Declaration of CE conformity and warranty conditions .....	15
Version history - CNC 8060 .....	17
Version history - CNC 8065 .....	19
Safety conditions .....	21
Returning conditions .....	25
CNC maintenance .....	27

**CHAPTER 1 GENERAL CONCEPTS**

---

1.1 Accessing the conversational mode .....	29
---	----

**CHAPTER 2 OPERATING IN JOG MODE**

---

2.1 Introduction .....	32
2.1.1 Standard screen of the conversational mode.....	32
2.1.2 Auxiliary screen of the conversational mode.....	33
2.1.3 Cycle editing.....	35
2.1.4 Cycle simulation .....	36
2.1.5 Cycle execution.....	37
2.2 Operations with the axes. ....	38
2.2.1 Home search.....	38
2.2.2 Jog .....	39
2.2.3 Jogging the axes with handwheels .....	41
2.2.4 Moving an axis to a particular position (coordinate).....	43
2.2.5 Coordinate preset.....	43
2.3 Spindle control .....	44
2.4 Tool selection and tool change .....	45
2.5 Setting the feedrate and spindle speed. ....	45
2.6 Setting and activating the zero offsets and the fixture offsets.....	46
2.7 Tool calibration.....	47
2.7.1 Manual calibration. Calibration without a probe .....	49
2.7.2 Semi-automatic calibration. Calibration with a probe .....	52
2.7.3 Automatic calibration with a probe and a canned cycle ("plane" geometric configuration)	55
2.7.4 Automatic calibration with a probe and a canned cycle ("trihedron" geometric configuration) .....	57

**CHAPTER 3 WORKING WITH OPERATIONS OR CYCLES**

---

3.1 Canned cycles available in the editor. ....	61
3.2 Configuring the cycle editor. ....	62
3.3 Teach-in mode. ....	63
3.4 Selecting data, profiles and icons .....	64

**CHAPTER 4 CANNED CYCLES OF THE EDITOR. TURNING AND FACING.**

---

4.1 Simple turning cycle.....	66
4.1.1 Basic operation. ....	69
4.2 Turning cycle with vertex rounding. ....	71
4.2.1 Basic operation. ....	75
4.3 Simple facing cycle. ....	77
4.3.1 Basic operation. ....	80
4.4 Facing cycle with vertex rounding.....	82
4.4.1 Basic operation. ....	86
4.5 Vertex chamfering cycle.....	88
4.5.1 Basic operation. ....	93
4.6 "Chamfering between points" cycle. ....	95
4.6.1 Basic operation. ....	100
4.7 Vertex chamfering cycle 2.....	102
4.7.1 Basic operation. ....	107
4.8 Vertex rounding cycle. ....	109
4.8.1 Basic operation. ....	114
4.9 "Rounding between points" cycle.....	116



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

4.9.1 Basic operation. .... 121

**CHAPTER 5 CANNED CYCLES OF THE EDITOR. TAPPING.**

5.1 Longitudinal threading cycle. .... 124  
 5.1.1 Basic operation. .... 128  
 5.2 Taper threading cycle. .... 129  
 5.2.1 Basic operation. .... 134  
 5.3 Face threading cycle. .... 135  
 5.3.1 Basic operation. .... 139  
 5.4 Thread repair cycle. .... 140  
 5.4.1 Basic operation. .... 146  
 5.5 Multi-entry threading cycle. .... 147  
 5.5.1 Basic operation. .... 152

**CHAPTER 6 CANNED CYCLES OF THE EDITOR. GROOVING.**

6.1 Calibrate the grooving tool (location code, shape). .... 154  
 6.1.1 Outside longitudinal grooving. .... 154  
 6.1.2 Inside longitudinal grooving. .... 155  
 6.1.3 Face grooving. .... 156  
 6.2 Simple longitudinal grooving cycle. .... 157  
 6.2.1 Basic operation. .... 161  
 6.3 Simple face grooving cycle. .... 163  
 6.3.1 Basic operation. .... 167  
 6.4 Inclined longitudinal grooving cycle. .... 169  
 6.4.1 Basic operation. .... 174  
 6.5 Inclined face grooving cycle. .... 176  
 6.5.1 Basic operation. .... 181  
 6.6 Cut-off cycle. .... 183  
 6.6.1 Basic operation. .... 186

**CHAPTER 7 CANNED CYCLES OF THE EDITOR. PROFILES.**

7.1 Point-to-point turning cycle. .... 188  
 7.1.1 Basic operation. .... 194  
 7.1.2 Programming example. .... 195  
 7.2 Profile turning cycle. .... 196  
 7.2.1 Basic operation. .... 202  
 7.2.2 Programming example. .... 203  
 7.3 ZC/YZ plane profiling cycle. .... 209  
 7.3.1 Basic operation. .... 213  
 7.4 Rectangular pocket cycle in the ZC/YZ plane. .... 214  
 7.4.1 Basic operation. .... 219  
 7.5 Circular pocket cycle in the ZC/YZ plane. .... 220  
 7.5.1 Basic operation. .... 224  
 7.6 Circular pre-empted pocket cycle in the ZC/YZ plane. .... 225  
 7.6.1 Basic operation. .... 229  
 7.7 2D profile pocket cycle in the ZC/YZ plane. .... 230  
 7.7.1 Basic operation. .... 234  
 7.8 XC/XY plane profiling cycle. .... 235  
 7.8.1 Basic operation. .... 239  
 7.9 Rectangular pocket cycle in the XC/XY plane. .... 240  
 7.9.1 Basic operation. .... 245  
 7.10 Circular pocket cycle in the XC/XY plane. .... 246  
 7.10.1 Basic operation. .... 250  
 7.11 Circular pre-empted pocket in the XC/XY plane. .... 251  
 7.11.1 Basic operation. .... 255  
 7.12 2D profile pocket cycle in the XC/XY plane. .... 256  
 7.12.1 Basic operation. .... 260

**CHAPTER 8 CANNED CYCLES OF THE EDITOR. LONGITUDINAL MACHINING.**

8.1 Center punching. .... 262  
 8.1.1 Basic operation. .... 265  
 8.2 Drilling. .... 266  
 8.2.1 Basic operation. .... 269  
 8.3 Tapping. .... 270  
 8.3.1 Basic operation. .... 272  
 8.4 Multiple center-punching. .... 273  
 8.4.1 Basic operation. .... 276  
 8.5 Multiple drilling. .... 277  
 8.5.1 Basic operation. .... 281  
 8.6 Multiple tapping. .... 282



CNC 8060  
 CNC 8065

(REF: 1709)

8.6.1	Basic operation.....	285
8.7	Multiple reaming.....	286
8.7.1	Basic operation.....	289
8.8	Multiple boring.....	290
8.8.1	Basic operation.....	294
8.9	Multiple bore milling.....	295
8.9.1	Basic operation.....	298
8.10	Multiple thread milling.....	299
8.10.1	Basic operation.....	303
8.11	Multiple slot milling.....	304
8.11.1	Basic operation.....	308

**CHAPTER 9 CANNED CYCLES OF THE EDITOR. POSITIONING.**

9.1	Positioning cycle.....	310
9.2	Positioning cycle with M functions.....	312

**CHAPTER 10 CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.**

10.1	Multiple machining in straight line.....	316
10.1.1	Programming example.....	317
10.2	Multiple machining in arc pattern.....	318
10.2.1	Programming example.....	319
10.3	Multiple machining in rectangular pattern.....	320
10.3.1	Programming example.....	321
10.4	Multiple machining in grid pattern.....	322
10.4.1	Programming example.....	323
10.5	Random multiple machining.....	324
10.5.1	Programming example.....	324

**CHAPTER 11 STANDARD THREADS**

11.1	Regular pitch metric thread — M (S.I.).....	326
11.2	Fine pitch metric thread — M (S.I.F.).....	327
11.3	Regular pitch Whitworth thread — BSW (W.).....	328
11.4	Fin pitch Whitworth thread — BSF.....	329
11.5	Regular pitch Unified American thread — UNC (NC, USS).....	330
11.6	Fine pitch Unified American thread — UNF (NF, SAE).....	331
11.7	Whitworth gas thread — BSP.....	332
11.8	Threads for pipes of the oil sector — A.P.I.....	333

**CHAPTER 12 SAVING PROGRAMS**

12.1	List of saved programs.....	335
12.2	Edit a new part-program.....	336
12.3	Delete a new part program.....	336
12.4	Inserting comments in any part-program.....	336
12.5	Inserting a cycle in the current part-program.....	336

**CHAPTER 13 EXECUTION AND SIMULATION**

13.1	Execute a part-program.....	338
13.1.1	Execute a portion of a part-program.....	338
13.1.2	Graphics screen in execution.....	339
13.2	Simulating a part-program.....	340
13.2.1	Simulate a portion of a part-program.....	340
13.2.2	Graphics screen in simulation.....	341
13.2.3	Simulation options.....	342
13.3	Simulating or executing an operation that has been saved.....	343
13.3.1	Simulating a cycle.....	343
13.3.2	Cycle execution.....	344



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

BLANK PAGE

# ABOUT THE PRODUCT - CNC 8060

## BASIC CHARACTERISTICS.

Basic characteristics.	8060 M FL	8060 M Power	8060 T FL	8060 T Power	8060 L
Number of axes.	3 to 4	3 to 6	3 to 4	3 to 6	3 to 6
Number of spindles.	1	1 to 2	1 to 2	1 to 3	1
Maximum number of axes and spindles.	5	7	5	7	7
Interpolated axes.	4	4	4	4	4
Number of tool magazines.	1	1	1	1 to 2	1
Number of execution channels.	1	1	1	1 to 2	1
Number of handwheels.	1 to 3				
Type of servo system.	Analog / Sercos Digital				
Communications.	RS485 / RS422 / RS232 Ethernet				
Integrated PLC. PLC execution time. Digital inputs / Digital outputs. Marks / Registers. Timers / Counters. Symbols.	< 1ms/K 1024 / 1024 8192 / 1024 512 / 256 Unlimited				
Block processing time.	< 2.0 ms	< 1.5 ms	< 2.0 ms	< 1.5 ms	< 1 ms

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

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



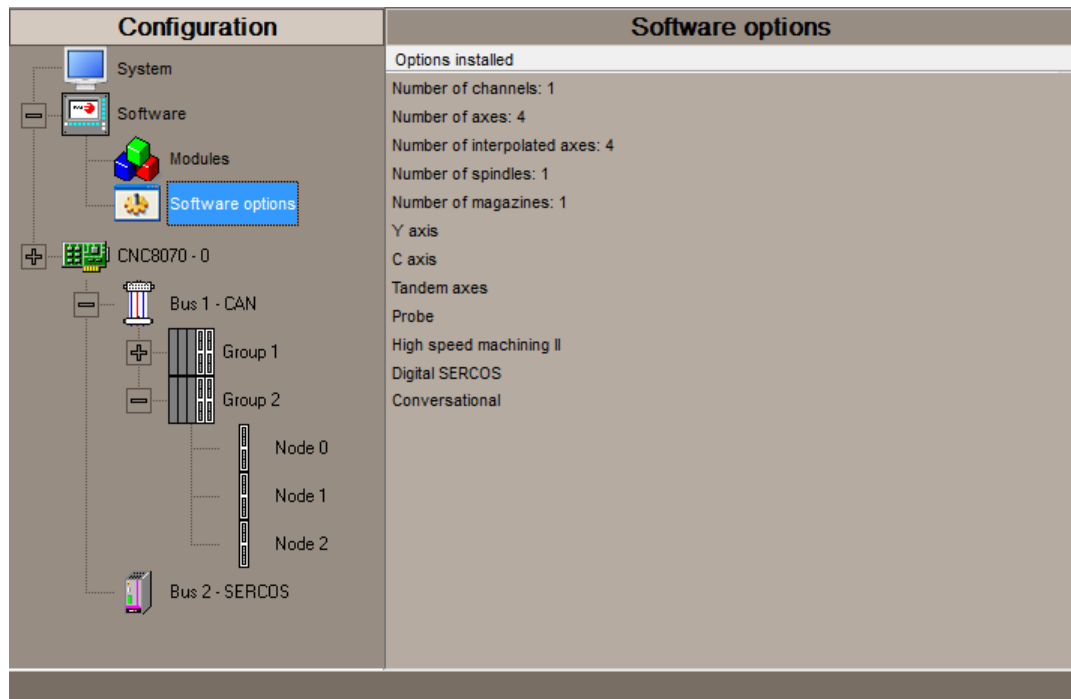
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## SOFTWARE OPTIONS.

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



Consult the ordering handbook for information on the software options available for your model.

### SOFT 8060 ADDIT AXES

#### Additional shaft.

Add axes to the default configuration.

### SOFT 8060 ADDIT SPINDLES

#### Additional spindle.

Add spindles to the default configuration.

### SOFT 8060 ADDIT TOOL MAGAZ

#### Additional tool magazine.

Add tool magazines to the default configuration.

### SOFT 8060 ADDIT CHANNELS

#### Additional channel.

Add channels to the default configuration.

### SOFT DIGITAL SERCOS

#### Sercos digital bus.

Sercos digital bus.

### SOFT EDIT/SIMUL

#### EDISIMU mode (editing and simulation).

It allows for the editing, modification and simulation of a part-program.

### SOFT TOOL RADIUS COMP

#### Compensación de radio.

Tool compensation allows programming the contour to be machined based on part dimensions of the and without taking into account the dimensions of the tool that will be used later on. This avoids having to calculate and define the tool path based on the tool radius.

### SOFT PROFILE EDITOR

#### Profile editor.

Allows for the part profiles to be edited graphically and to import dxf files.

### SOFT 60 F3D GRAPHICS

#### F3D graphics.

High definition solid 3D graphics for the execution and simulation of part-programs and canned cycles of the editor.

During machining, the F3D graphics display the tool removing the material from the part in real time, allowing for the condition of the part to be seen at all times. F3D graphics can display up to 4 views of the part, where each can be rotated, zoomed in or zoomed out. Measurements can also be made on the part and even sections on the piece from any angle.

### SOFT 60 IIP CONVERSATIONAL

#### Interactive Icon-based Pages (conversational mode).

IIP or conversational mode is specifically designed for people without any prior programming knowledge or for those that are not familiarized with Fagor CNC's.

Working in conversational mode is easier than in ISO mode, as it ensures proper data entry and minimizes the number of operations to be defined. There is no need to work with part-programs.

### SOFT 60 RTCP

#### Dynamic RTCP (Rotating Tool Center Point).

The dynamic RTCP option is required for interpolation machining with 4, 5 or 6 axis.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)



## SOFT 60 C AXIS

### C axis.

It activates the kinematics for working with the C axis and the associated canned cycles. The CNC can control several C axes. The parameters of each axis indicate if it will function as a C axis or not, where it will not be necessary to activate another axis for the machine parameters.

## SOFT 60 Y AXIS

### Y axis for lathe.

It activates the kinematics for working with the Y axis and the associated canned cycles.

## SOFT 60 TANDEM AXES

### Tandem axes.

A tandem axis consists in two motors mechanically coupled (slaved) and making up a single transmission system (axis or spindle). A tandem axis helps provide the necessary torque to move an axis when a single motor is not capable of supplying enough torque to do it.

When activating this feature, it should be kept in mind that for each tandem axis of the machine, another axis must be added to the entire configuration. For example, on a large 3-axis lathe (X Z and tailstock), if the tailstock is a tandem axis, the final purchase order for the machine must indicate 4 axes.

## SOFT 60 SYNCHRONISM

### Synchronization of axes and spindles.

The axes and ballscrews may be synchronized in two ways: in terms of speed or position. The CNC configuration takes into consideration the synchronization of 2 axes or 2 spindles. Once synchronized, only the master displays and programs the element.

## SOFT 60 HSSA I MACHINING SYSTEM

### High Speed Surface Accuracy.

This is the new version of algorithms for high speed machining (HSC). This new HSSA algorithm allows for high speed machining optimization, where higher cutting speeds, smoother contours, a better surface finishing and greater precision are achieved.

## SOFT 60 HSSA II MACHINING SYSTEM

### HSSA-II machining system.

This is the new version of algorithms for high speed machining (HSC). This new HSSA algorithm allows for high speed machining optimization, where higher cutting speeds, smoother contours, a better surface finishing and greater precision are achieved.

## SOFT 60 PROBE

### Probing canned cycles.

The CNC may have two probes; usually a tabletop probe to calibrate tools and a measuring probe to measure the part.

This option activates the functions G100, G103 and G104 (for probe movements) and probe canned cycles (which help to measure part surfaces and to calibrate tools).

For the laser model, it only activates the non-cycle function G100.

## SOFT 60 CONV USER CYCLES

### Conversational user cycles.

Incorporation of user cycles in conversational mode.

## SOFT 60 PROGTL3

### ProGTL3 programming language

Another language apart from ISO for the programming of profiles using a geometric language without the need to use external CAD systems. This language allows for program functions to define lines and circles which define the points of intersection on a profile, in addition to macros for the creation of solids defined by a flat profile and one or more section profiles.

## SOFT 60 PPTRANS

### Part-program translator.

The program translator may be used to convert programs written in other languages into Fagor ISO codes.

## SOFT THIRD PARTY CANOPEN

### Third-party CANopen.

Enables the use of non-Fagor CANopen modules.

## SOFT MAB SYSTEM.

### MAB drives.

Sercos connection with MAB drives.

## SOFT 60 PWM CONTROL

### Pulse-Width Modulation.

This function is only available for Sercos bus controlled systems. It is mostly oriented toward laser machines for the cutting of very thick sheets, where the CNC generates a series of PWM pulses to control the power of the laser when drilling the starting point.

This feature is essential for cutting very thick sheets and it requires two quick digital outputs located on the central unit. With this new feature, the OEM does not need to install or program any external device, which reduces machine costs and installation times. The end user also benefits, since the "Cutting with PWM" feature is much easier to use and program.

## SOFT 60 GAP CONTROL

### Gap control.

This is mostly oriented toward laser machines. Gap control makes it possible to maintain a set distance between the laser nozzle and the surface of the sheet. This distance is calculated by a sensor connected to the CNC, so that the CNC offsets the sensor variations on the distance programmed with additional movements in the axis programmed for the gap.

BLANK PAGE

# ABOUT THE PRODUCT - CNC 8065

## BASIC CHARACTERISTICS.

Basic characteristics.	8065 M		8065 M Power	
	Basic	Pack 1	Basic	Pack 1
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

Basic characteristics.	8065 T		8065 T Power	
	Basic	Pack 1	Basic	Pack 1
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
Maximum 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

Basic characteristics.	8065 M	8065 M Power	8065 T	8065 T Power
Number of handwheels.	1 to 12			
Type of servo system.	Analog / Digital Sercos / Digital Mechatrolink			
Communications.	RS485 / RS422 / RS232 Ethernet			
Integrated PLC.				
PLC execution time.	< 1ms/K			
Digital inputs / Digital outputs.	1024 / 1024			
Marks / Registers.	8192 / 1024			
Timers / Counters.	512 / 256			
Symbols.	Unlimited			
Block processing time.	< 1 ms			



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

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

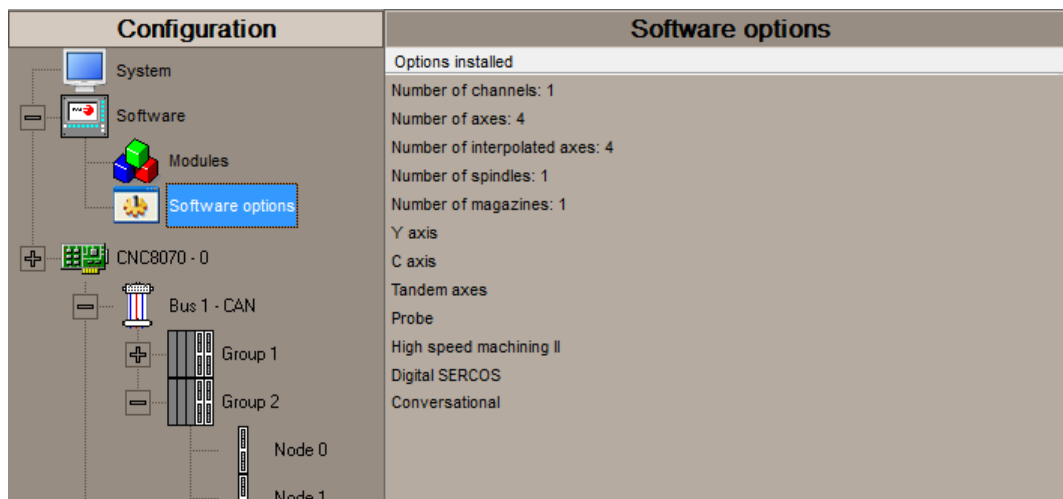
(\*) Differential TTL / Sinusoidal 1 Vpp    (\*\*) TTL / Differential TTL / Sinusoidal 1 Vpp / SSI protocol / FeeDat / EnDat

#### Customizing (on an open system only).

PC-based open system, fully customizable.  
 INI configuration files.  
 Tool for display configuration FGUIM.  
 Visual Basic®, Visual C++®, etc.  
 Internal databases in Microsoft® Access.  
 OPC compatible interface

## SOFTWARE OPTIONS.

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



Consult the ordering handbook for information on the software options available for your model.

### SOFT ADDIT AXES

#### Additional shaft.

Add axes to the default configuration.

### SOFT ADDIT SPINDLES

#### Additional spindle.

Add spindles to the default configuration.

### SOFT ADDIT TOOL MAGAZ

#### Additional tool magazine.

Add tool magazines to the default configuration.

### SOFT ADDIT CHANNELS

#### Additional channel.

Add channels to the default configuration.

### SOFT 4 AXES INTERPOLATION LIMIT

#### Limited to 4 interpolated axes.

It limits the number of axes to 4, where the CNC can also interpolate these at the same time.

### SOFT OPEN SYSTEM

#### Open system.

The CNC is a closed system that offers all the features needed to machine parts. Nevertheless, at times there are some customers who use third-party applications to take measurements, perform statistics or other tasks apart from machining a part.

This feature must be active when installing this type of application, even if they are Office files. Once the application has been installed, it is recommended to close the CNC in order to prevent the operators from installing other kinds of applications that could slow the system down and affect the machining operations.

### SOFT DIGITAL SERCOS

#### Sercos digital bus.

Sercos digital bus.

### SOFT EDIT/SIMUL

#### EDISIMU mode (editing and simulation).

It allows for the editing, modification and simulation of a part-program.

### SOFT DUAL-PURPOSE (M-T)

#### Dual-purpose machine.

A dual-purpose machine allows for both milling and turning cycles. On lathes with a Y axis, it allows making pockets, bosses and even irregular pockets with islands during milling cycles. Turning cycles can be used by milling machines which have a rotary axis that functions as a C axis.

### SOFT IEC 61131 LANGUAGE

#### IEC 61131 language

IEC 61131 is a PLC programming language that is very popular in alternative markets, which is slowly entering into the machine-tool market. With this feature, the PLC may be programmed either in the usual Fagor language or in IEC 61131 format.

This feature requires the MP-PLUS (83700201) processor.

### SOFT TOOL RADIUS COMP

#### Compensación de radio.

Tool compensation allows programming the contour to be machined based on part dimensions of the and without taking into account the dimensions of the tool that will be used later on. This avoids having to calculate and define the tool path based on the tool radius.

### SOFT PROFILE EDITOR

#### Profile editor.

Allows for the part profiles to be edited graphically and to import dxf files.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**SOFT IIP CONVERSATIONAL****Interactive Icon-based Pages (conversational mode).**

IIP or conversational mode is specifically designed for people without any prior programming knowledge or for those that are not familiarized with Fagor CNC's.

Working in conversational mode is easier than in ISO mode, as it ensures proper data entry and minimizes the number of operations to be defined. There is no need to work with part-programs.

**SOFT HD GRAPHICS****HD graphics.**

High definition solid 3D graphics for the execution and simulation of part-programs and canned cycles of the editor.

During machining, the HD graphics display, in real time, the tool removing the material from the part, allowing the condition of the part to be seen at all times. HD graphics can display up to 4 views of the part, where each can be rotated, zoomed in or zoomed out. Measurements can also be made on the part and even sections on the piece from any angle.

In a multi-channel system, this feature requires the MP-PLUS (83700201) processor.

**SOFT RTCP****Dynamic RTCP (Rotating Tool Center Point).**

The dynamic RTCP option is required for interpolation machining with 4, 5 or 6 axis.

This feature requires the MP-PLUS (83700201) processor.

**SOFT C AXIS****C axis.**

It activates the kinematics for working with the C axis and the associated canned cycles. The CNC can control several C axes. The parameters of each axis indicate if it will function as a C axis or not, where it will not be necessary to activate another axis for the machine parameters.

**SOFT Y AXIS****Y axis for lathe.**

It activates the kinematics for working with the Y axis and the associated canned cycles.

**SOFT TANDEM AXES****Tandem axes.**

A tandem axis consists in two motors mechanically coupled (slaved) and making up a single transmission system (axis or spindle). A tandem axis helps provide the necessary torque to move an axis when a single motor is not capable of supplying enough torque to do it.

When activating this feature, it should be kept in mind that for each tandem axis of the machine, another axis must be added to the entire configuration. For example, on a large 3-axis lathe (X Z and tailstock), if the tailstock is a tandem axis, the final purchase order for the machine must indicate 4 axes.

**SOFT SYNCHRONISM****Synchronization of axes and spindles.**

The axes and ballscrews may be synchronized in two ways: in terms of speed or position. The CNC configuration takes into consideration the synchronization of 2 axes or 2 spindles. Once synchronized, only the master displays and programs the element.

**SOFT KINEMATIC CALIBRATION****Kinematics calibration.**

This work mode may be used to calibrate kinematics for the first time and to re-calibrate it every so often to correct any possible deviations originated by the day-to-day machine work.

**SOFT HSSA II MACHINING SYSTEM****HSSA-II machining system.**

This is the new version of algorithms for high speed machining (HSC). This new HSSA algorithm allows for high speed machining optimization, where higher cutting speeds, smoother contours, a better surface finishing and greater precision are achieved.

**SOFT TANGENTIAL CONTROL****Tangential control.**

"Tangential Control" maintains a rotary axis always in the same orientation with respect to the programmed tool path. The machining path is defined on the axes of the active plane and the CNC maintains the orientation of the rotary axis along the entire tool path.

**SOFT PROBE****Probing canned cycles.**

The CNC may have two probes; usually a tabletop probe to calibrate tools and a measuring probe to measure the part.

This option activates the functions G100, G103 and G104 (for probe movements) and probe canned cycles (which help to measure part surfaces and to calibrate tools).

**SOFT CONV USER CYCLES****Conversational user cycles.**

Incorporation of user cycles in conversational mode.

**SOFT 70 PROGTL3****ProGTL3 programming language**

Another language apart from ISO for the programming of profiles using a geometric language without the need to use external CAD systems. This language allows for program functions to define lines and circles which define the points of intersection on a profile, in addition to macros for the creation of solids defined by a flat profile and one or more section profiles.

**SOFT PPTRANS****Part-program translator.**

The program translator may be used to convert programs written in other languages into Fagor ISO codes.

**SOFT THIRD PARTY CANOPEN****Third-party CANopen.**

Enables the use of non-Fagor CANopen modules.

**SOFT FVC UP TO 10m3****SOFT FVC MORE TO 10m3****Medium and large volumetric compensation.**

5-axis machines are generally used during the manufacturing of large parts. The accuracy of the parts is limited by the machine manufacturing tolerances and is effected by temperature variations during machining.

In sectors such as the aerospace industry, machining demands mean that classic compensation tools are becoming suboptimal. Volumetric compensation FVC comes in to complement the machine adjusting tools. When mapping the total work volume of the machine, the CNC knows the exact position of the tool at all times. After applying the required compensation, the resulting part is made with the desired precision and tolerance.

There are 2 choices, which depend on the size of the machine, being up to 10 m<sup>3</sup> and over 10 m<sup>3</sup>.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

# DECLARATION OF CE CONFORMITY AND WARRANTY CONDITIONS

## DECLARATION OF CONFORMITY

The declaration of conformity for the CNC is available in the downloads section of FAGOR'S corporate website. <http://www.fagorautomation.com>. (Type of file: Declaration of conformity).

## WARRANTY TERMS

The warranty conditions for the CNC are available in the downloads section of FAGOR's corporate website. <http://www.fagorautomation.com>. (Type of file: General sales-warranty conditions).



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

BLANK PAGE



# VERSION HISTORY - CNC 8060

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

## Ref. 1402

<b>Software V01.00</b>
8060. First version.

## Ref. 1412

<b>Software V01.10</b>	
User cycles.	
<b>Software V01.10.03</b>	
Cycle editor.	New configuration option. Invert the starting and end points in face threading and face thread repair.
Cycles of the editor. <ul style="list-style-type: none"> <li>Center punching.</li> <li>Multiple center-punching.</li> <li>Multiple reaming.</li> <li>Multiple boring.</li> <li>Multiple bore milling.</li> <li>Multiple thread milling.</li> <li>Circular pre-empted pocket cycle in the ZC/YZ plane.</li> <li>Circular pre-empted pocket in the XC/XY plane.</li> </ul>	New cycles in the editor.
Cycles of the editor. <ul style="list-style-type: none"> <li>Longitudinal threading.</li> <li>Taper threading.</li> <li>Thread repair (except face threading).</li> <li>Multi-entry (start) threading</li> </ul>	Standard threads A.P.I.
Cycles of the editor. <ul style="list-style-type: none"> <li>Taper threading.</li> <li>Face threading.</li> <li>Thread repair.</li> <li>Multi-entry (start) threading</li> </ul>	The cycles offer the possibility of programming more tool penetration types; penetration by final flank, penetration in zig zag (alternates radial and final flanks) and penetration in zig zag (alternates between starting and final flanks).
Cycles of the editor. <ul style="list-style-type: none"> <li>Longitudinal threading.</li> <li>Taper threading.</li> <li>Multi-entry (start) threading</li> </ul>	The cycles offers the possibility of programming variable-pitch threads.
Cycles of the editor. <ul style="list-style-type: none"> <li>Taper threading.</li> <li>Thread repair.</li> </ul>	The cycle offers the following novelties. <ul style="list-style-type: none"> <li>The cycle offers the possibility of executing a partial thread repair.</li> <li>The cycle offers the possibility of making threads with the start and/or end in different directions to that of their geometry. This option may be used to make threads that do not start or end in "edges".</li> </ul>
Cycles of the editor. <ul style="list-style-type: none"> <li>Thread repair.</li> </ul>	The cycle offers the following novelties. <ul style="list-style-type: none"> <li>The cycle offers the possibility the number of entries (starts) in thread repair.</li> <li>The cycle offers the possibility of programming a face thread repair.</li> <li>Step of the programmed depth (h) of the thread maintaining the geometry.</li> </ul>
Cycles of the editor. <ul style="list-style-type: none"> <li>Longitudinal threading.</li> <li>Taper threading.</li> <li>Face threading.</li> <li>Thread repair.</li> <li>Multi-entry (start) threading</li> </ul>	The cycle offers the following novelties. <ul style="list-style-type: none"> <li>The cycle offers the possibility of making blind threads.</li> <li>The cycle offers the possibility of defining the thread pitch in number of threads per inch.</li> </ul>
Cycles of the editor. <ul style="list-style-type: none"> <li>ZC/YZ plane profiling cycle.</li> </ul>	The cycles offer the possibility of defining a profile in the YZ plane.
Cycles of the editor. <ul style="list-style-type: none"> <li>XC/XY plane profiling cycle.</li> </ul>	The cycles offer the possibility of defining a profile in the XY plane.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

Software V01.10.03	
<p>Cycles of the editor.</p> <ul style="list-style-type: none"> <li>• Rectangular pocket in the ZC/YZ plane.</li> <li>• Circular pocket in the ZC/YZ plane.</li> <li>• Circular pre-empted pocket in the ZC/YZ plane.</li> <li>• 2D profile pocket in the ZC/YZ plane.</li> <li>• Rectangular pocket in the XC/XY plane.</li> <li>• Circular pocket in the XC/XY plane.</li> <li>• Circular pre-empted pocket in the XC/XY plane.</li> <li>• 2D profile pocket in the XC/XY plane.</li> </ul>	<p>The cycles offer the possibility of adding a multiple angular positioning.</p>
<p>Cycles of the editor.</p> <ul style="list-style-type: none"> <li>• Point-to-point turning.</li> <li>• Profile turning.</li> </ul>	<p>The cycles offer the possibility of machining a profile by segments.</p>
<p>Cycles of the editor.</p> <ul style="list-style-type: none"> <li>• Multiple drilling.</li> <li>• Multiple tapping.</li> <li>• Multiple slot milling.</li> </ul>	<p>The 3 parameters that define the angular positioning of the spindle now have new names <math>\alpha</math> is now called <math>\omega</math>, <math>\beta</math> is now called <math>\Delta\omega</math> and N is now called <math>N\omega</math>.</p>
<p>Cycles of the editor.</p> <ul style="list-style-type: none"> <li>• Points in line.</li> <li>• Points in arc.</li> <li>• Points in rectangle.</li> <li>• Points in grid.</li> <li>• Random points.</li> </ul>	<p>The editor lets add multiple machining (linear, arc, rectangular, grid and random) to pockets and to longitudinal machining.</p>

**Ref. 1709**

Software V01.60
Simulation options.
API threads. API thread pitch corrected in the manual.



CNC 8060  
CNC 8065

(REF: 1709)

# VERSION HISTORY - CNC 8065

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

## Ref. 1103

### Software V04.20

First version.

## Ref. 1201

### Software V04.22

Canned cycles. Point-to-point turning cycle. The table to define the points of the profile admits 25 points.

Canned cycles. Point-to-point turning cycle. New icon to delete all the points of the table.

Canned cycles. Multiple tapping cycle. The cycle allows programming the dwell at the bottom.

Canned cycles. Multiple slot milling cycle. The cycle allows defining several penetration passes.

Canned cycles. By default, the cycles assume for Xf the value set for Xi.

- Simple turning cycle.
- Turning cycle with vertex rounding.
- Simple facing cycle.
- Facing cycle with vertex rounding.
- "Chamfering between points" cycle.
- "Rounding between points" cycle.
- Taper threading cycle.
- Face threading cycle.
- Thread repair cycle.
- Multi-entry threading cycle.
- Simple longitudinal grooving cycle.
- Simple face grooving cycle.
- Inclined longitudinal grooving cycle.
- Inclined face grooving cycle.

Canned cycles. When selecting it in a constant surface speed cycle, it allows selecting the gear (range) even if the gear change is automatic.

Canned cycles. The cycles make the approach to the starting point on both axes of the plane at the same time.

Canned cycles. The [DEL] key deletes a profile from the list.

Canned cycles. Pressing [RECALL] on a tool gives access to the tool table.

Keyboard shortcuts [CTRL][C] and [CTRL][V] may be used on the programs list to copy and paste a program.

Selecting a program for editing no longer involves selecting it also for execution. Use the "Execute Program" softkey to select a program for execution.

## Ref. 1709

### Software V05.60

Simulation options.

API threads. API thread pitch corrected in the manual.

BLANK PAGE

# SAFETY CONDITIONS

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



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

## PRECAUTIONS BEFORE CLEANING THE UNIT

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

## PRECAUTIONS DURING REPAIRS

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

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

## PRECAUTIONS AGAINST PERSONAL HARM

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

**FAGOR** 

FAGOR AUTOMATION

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






(REF: 1709)

## PRECAUTIONS AGAINST DAMAGE TO THE PRODUCT

<b>Work environment.</b>	This unit is ready to be used in industrial environments complying with the directives and regulations effective in the European Community. Fagor Automation shall not be held responsible for any damage suffered or caused by the CNC when installed in other environments (residential, homes, etc.).
<b>Install this unit in the proper place.</b>	It is recommended, whenever possible, to install the CNC away from coolants, chemical product, blows, etc. that could damage it. This unit meets the European directives on electromagnetic compatibility. Nevertheless, it is recommended to keep it away from sources of electromagnetic disturbance such as: <ul style="list-style-type: none"> <li>Powerful loads connected to the same mains as the unit.</li> <li>Nearby portable transmitters (radio-telephones, Ham radio transmitters).</li> <li>Nearby radio / TC transmitters.</li> <li>Nearby arc welding machines.</li> <li>Nearby high voltage lines.</li> </ul>
<b>Enclosures.</b>	It is up to the manufacturer to guarantee that the enclosure where the unit has been installed meets all the relevant directives of the European Union.
<b>Avoid disturbances coming from the machine.</b>	The machine must have all the interference generating elements (relay coils, contactors, motors, etc.) uncoupled.
<b>Use the proper power supply.</b>	Use an external regulated 24 Vdc power supply for the keyboard, operator panel and the remote modules.
<b>Connecting the power supply to ground.</b>	The zero Volt point of the external power supply must be connected to the main ground point of the machine.
<b>Analog inputs and outputs connection.</b>	Use shielded cables connecting all their meshes to the corresponding pin.
<b>Ambient conditions.</b>	Maintain the CNC within the recommended temperature range, both when running and not running. See the corresponding chapter in the hardware manual.
<b>Central unit enclosure.</b>	To maintain the right ambient conditions in the enclosure of the central unit, it must meet the requirements indicated by Fagor. See the corresponding chapter in the hardware manual.
<b>Power switch.</b>	This switch must be easy to access and at a distance between 0.7 and 1.7 m (2.3 and 5.6 ft) off the floor.

## SAFETY SYMBOLS

### Symbols that may appear in the manual.

	<i>Danger or prohibition symbol.</i> This symbol indicates actions or operations that may hurt people or damage products.
	<i>Warning or caution symbol.</i> This symbol indicates situations that certain operations could cause and the suggested actions to prevent them.
	<i>Obligation symbol.</i> This symbol indicates actions and operations that must be carried out.
	<i>Information symbol.</i> This symbol indicates notes, warnings and advises.
	<i>Symbol for additional documentation.</i> This symbol indicates that there is another document with more detailed and specific information.

**Symbols that the product may carry.**



*Ground symbol.*

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



*ESD components.*

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



FAGOR AUTOMATION

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

(REF: 1709)

BLANK PAGE



## RETURNING CONDITIONS

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

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



FAGOR AUTOMATION

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

(REF: 1709)

BLANK PAGE

# 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. Never use air compressed at high pressure to clean the unit because it could cause the accumulation of electrostatic charges that could result in electrostatic shocks.

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

## PRECAUTIONS BEFORE CLEANING THE UNIT

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

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

**FAGOR** 

FAGOR AUTOMATION

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

(REF: 1709)

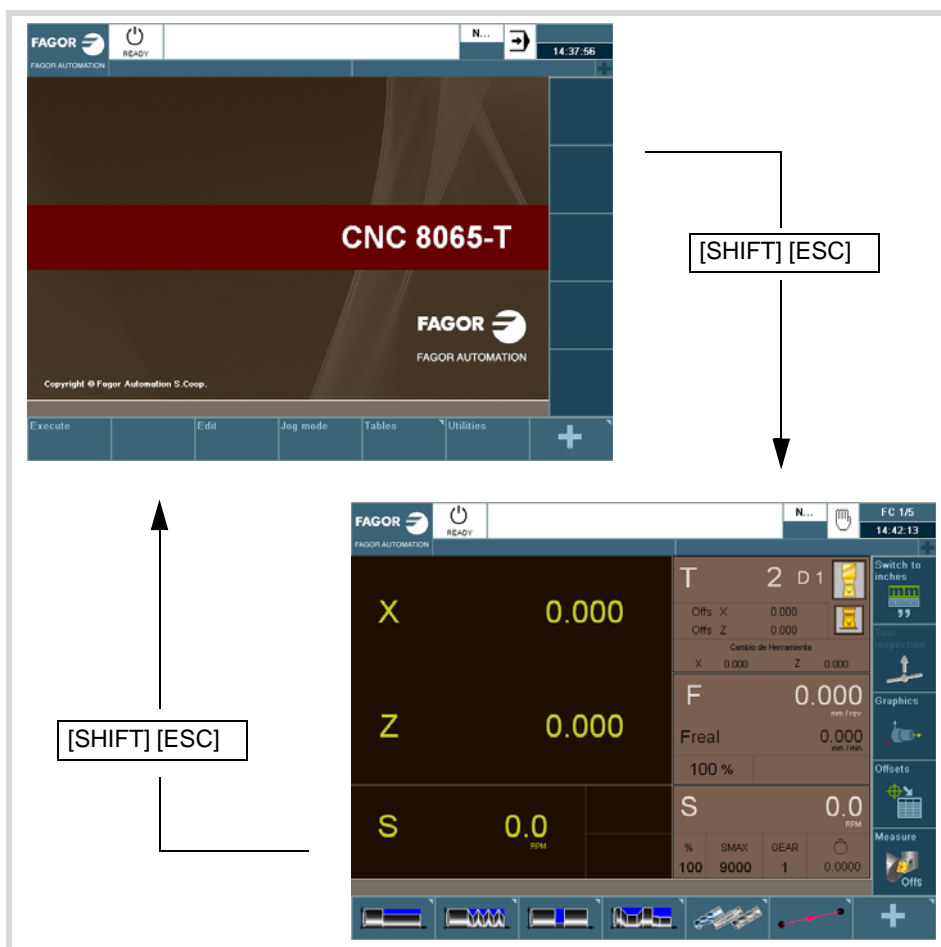
BLANK PAGE

# GENERAL CONCEPTS

# 1

## 1.1 Accessing the conversational mode

Once the CNC has been started up, press the key sequence [SHIFT] [ESC] to switch to conversational mode. Press [SHIFT] + [ESC] again to return to T mode.



The CNC setup must be done in T mode. Likewise, some errors must be eliminated in T mode.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

1.

**GENERAL CONCEPTS**

Accessing the conversational mode



FAGOR AUTOMATION

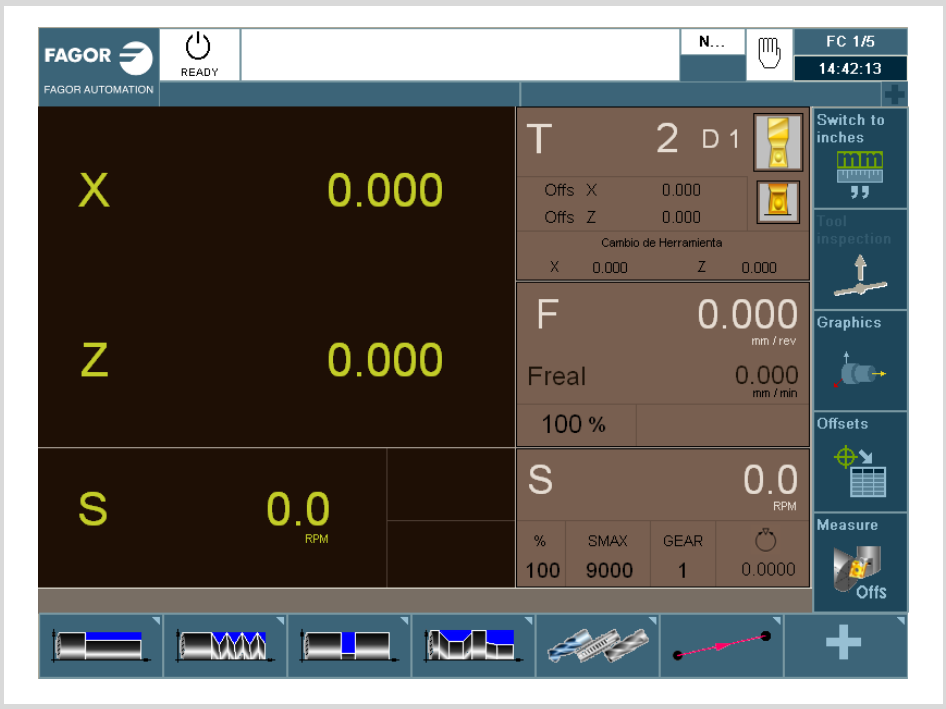
**CNC 8060**  
**CNC 8065**

(REF: 1709)

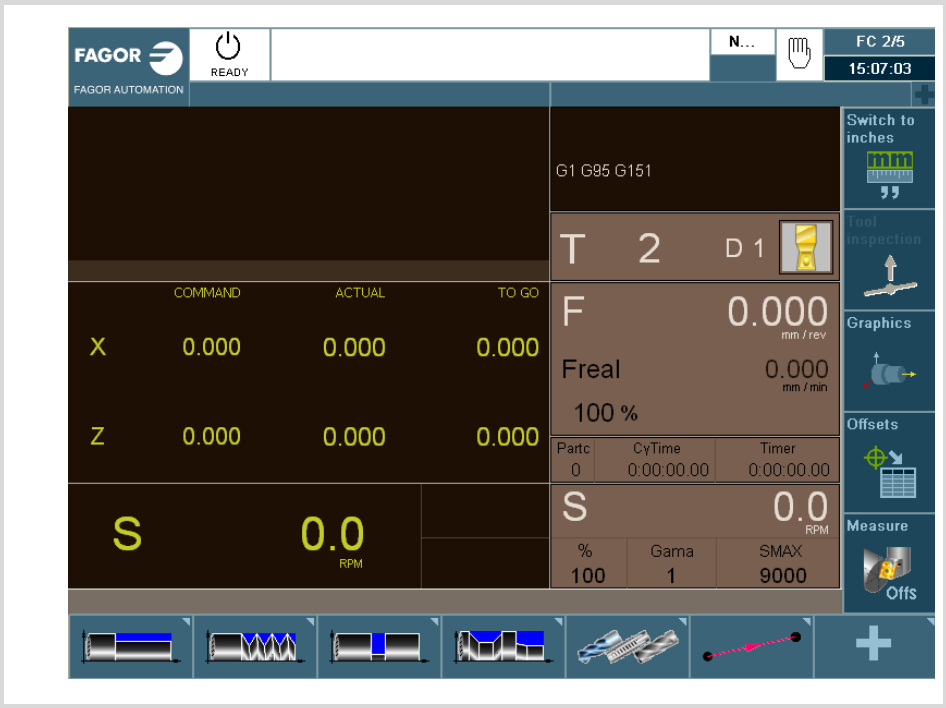
# OPERATING IN JOG MODE

# 2

The standard screen of the TC work mode is:



When pressing the two-colored key, the CNC shows the auxiliary screen of the TC mode:

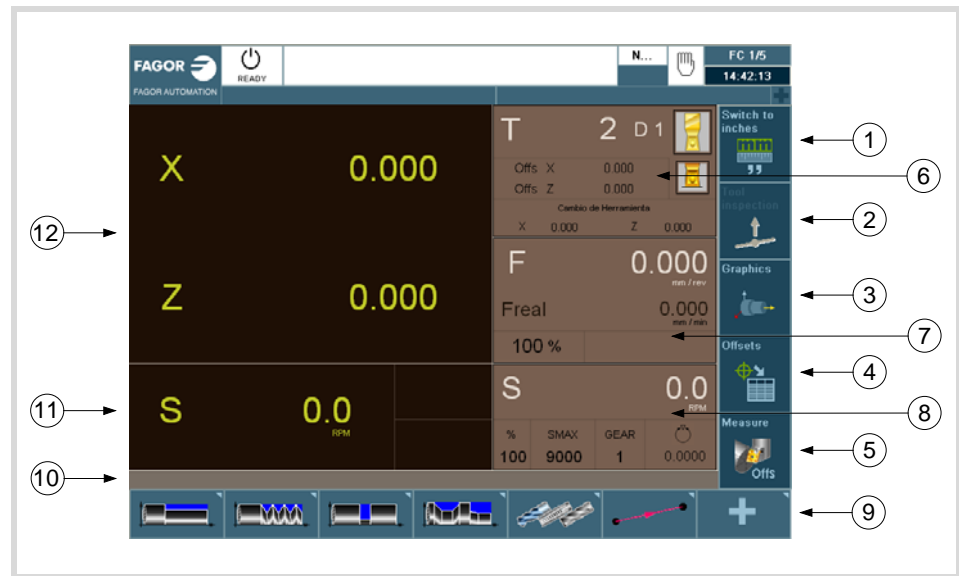


CNC 8060  
CNC 8065

(REF: 1709)

## 2.1 Introduction

### 2.1.1 Standard screen of the conversational mode



- 1 Softkey for selecting units mm/inches.
- 2 Softkey to go into tool inspection.
- 3 Softkey to access the graphics in execution mode.
- 4 Softkey for selecting OFFSETS.
- 5 Softkey for tool calibration.
- 6 Window that shows:
  - The selected tool (T).
  - Graphic representation of the location code (shape).
  - The (D) offset number associated with the selected tool.
  - The offset defined for the tool.
  - The position values (coordinates) of the tool change point referred to machine reference zero. If one of these coordinates is selected, it may be assigned the value of the current position of that axis by pressing [RECALL].
- 7 Window showing the axis feedrate F currently selected, the % of F being applied and the real F value. When selecting an incremental jog or a handwheel, this window will also show the selected % with the corresponding icon and the selected %.
- 8 Window showing spindle related information:
  - The selected theoretical turning speed. S value when working in rpm and CSS value when working at constant surface speed.
  - Spindle status. It is represented with an icon and may be turning clockwise, counterclockwise or stopped.
  - The % of spindle speed being applied.
  - Maximum spindle rpm (Smax).
  - Active spindle range (gear).
- 9 Softkeys for cycle editing.
- 10 Message bar.
- 11 Real spindle rpm.
- 12 Position (coordinates) of the axes. The f symbol indicates that the axis is working in diameter.

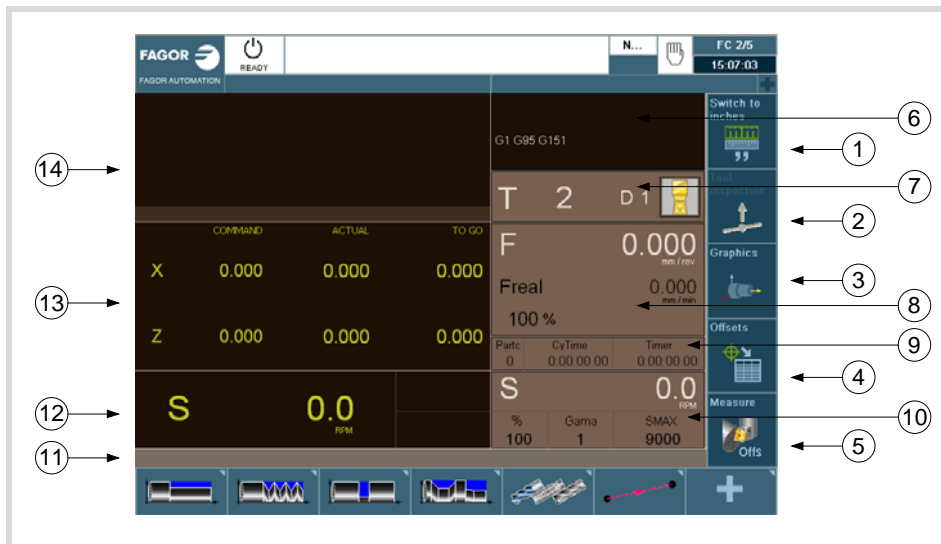
If there are more than one spindle in the active channel, S may be pressed repeatedly to select the spindle whose data is being displayed. If the cell for programmed turning speed is already selected, every time S is pressed, it will show the data of the next spindle.

# 2.

OPERATING IN JOG MODE  
Introduction



## 2.1.2 Auxiliary screen of the conversational mode



- 1 Softkey for selecting units mm/inches.
- 2 Softkey to go into tool inspection.
- 3 Softkey to access the graphics in execution mode.
- 4 Softkey for selecting OFFSETS.
- 5 Softkey for tool calibration.
- 6 Window that shows the status of the G, F, T, D, M functions.
- 7 Window that shows:
  - The selected tool (T).
  - Graphic representation of the location code (shape).
  - The (D) offset number associated with the selected tool.
- 8 Window showing the axis feedrate F currently selected, the % of F being applied and the real F value.
- 9 Window that shows the value of the variables:
  - Partc: It indicates the number of consecutive parts executed with the same part-program. Every time a new program is selected, this variable is reset to "0".
  - CyTime: It indicates the time elapsed while executing the part. It is given in "hours : minutes : seconds : hundredths of a second" format. Every time a part-program execution starts, even when repetitive, this variable is reset to "0".
  - Timer: It indicates the count of the timer enabled by PLC. It is given in "hours : minutes : seconds" format.
- 10 Window with spindle related information:
  - The selected theoretical turning speed. S value when working in rpm and CSS value when working at constant surface speed.
  - The % of spindle speed being applied.
  - Maximum spindle rpm (Smax).
  - Active spindle range (gear).
- 11 Message bar.
- 12 Window with spindle related information:
  - Theoretical speed.
  - Speed in RPM.
  - Speed in m/min.

# 2.

13 Window with axis related information:

- **COMMAD**: It indicates the programmed coordinate or position that the axis must reach.
- **ACTUAL**: It indicates the actual (current) position of the axis.
- **TO GO**: It indicates the distance which is left to run to the programmed coordinate.

14 Window that shows the lines of the program being executed.

2.

OPERATING IN JOG MODE

Introduction



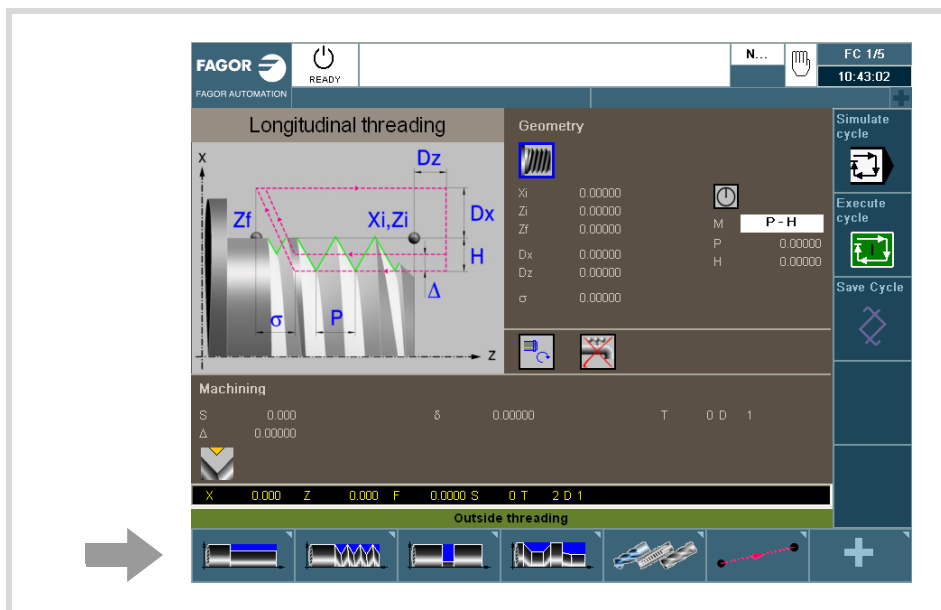
FAGOR AUTOMATION

CNC 8060  
CNC 8065

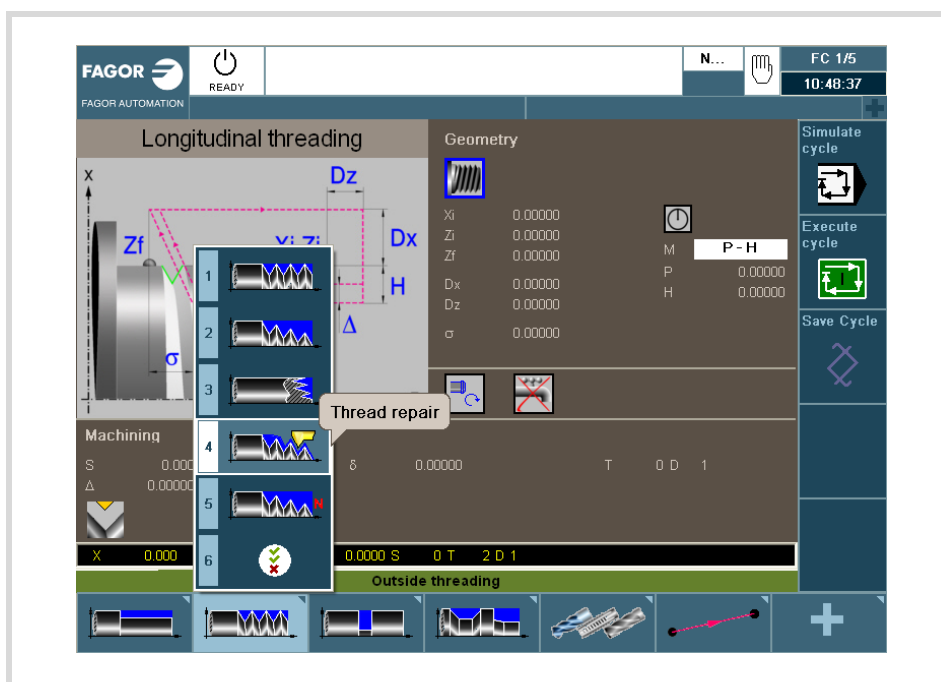
(REF: 1709)

## 2.1.3 Cycle editing

To edit a cycle, press the softkey for the desired cycle.



To select another cycle of the same family as the one selected, press the softkey again to drop the menu with the available cycles.



Once the cycle to be edited has been selected, enter the data in the windows corresponding to each parameter of that cycle. To validate each parameter and go on to the next one, [ENTER].

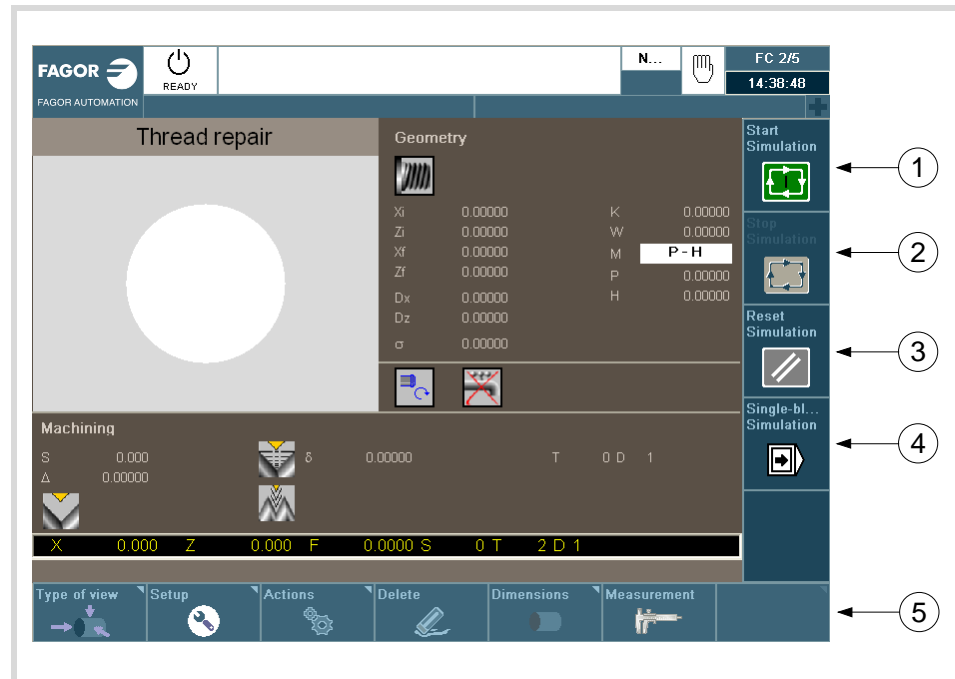
For further information on editing cycles, see the chapter ["3 Working with operations or cycles"](#).

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

For further information on saving cycles, see the chapter ["12 Saving programs"](#).

## 2.1.4 Cycle simulation

To simulate the edited cycle, press the vertical softkey [Simulate cycle].



- 1 Softkey to start cycle simulation.
- 2 Softkey to stop cycle simulation.
- 3 Softkey to reset the simulation.
- 4 Softkey to simulate the cycle block by block.
- 5 The horizontal softkeys may be used to configure how to display the simulated cycle.
  - Type of view.
  - Configuration.
  - Actions.
  - Delete.
  - Dimensions.
  - Measurement.

# 2.

OPERATING IN JOG MODE  
Introduction

**FAGOR**   
FAGOR AUTOMATION

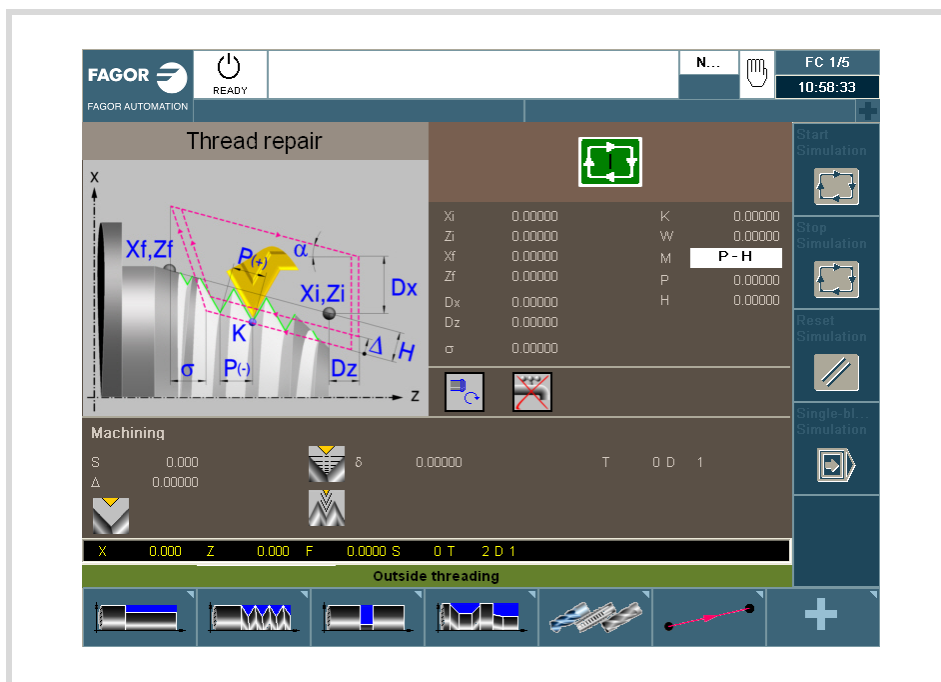
CNC 8060  
CNC 8065

(REF: 1709)

## 2.1.5 Cycle execution

To execute an edited cycle, press the vertical softkey [Execute cycle]. An icon will then appear with the start symbol to warn the user that it is going to execute the cycle.

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



# 2.

OPERATING IN JOG MODE  
Introduction

**FAGOR**  
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 2.2 Operations with the axes.

### 2.2.1 Home search.

Home search is the operation used to synchronize the system. This operation must be carried out when the CNC loses the position of the origin point (e.g. by turning the machine off).

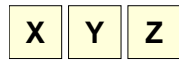
When "searching home", the axes move to the machine reference point and the CNC assumes the coordinate values assigned to that point by the machine manufacturer, referred to machine zero. When using distance-coded reference marks or absolute feedback, the axes will only move the distance necessary to verify their position.

The axes may be homed manually (axis by axis from the operator panel) or automatically (using a subroutine).

#### Manual home search (one axis at a time).



*The axis-by-axis home search cancels the zero offset, the fixture offset and the measuring offset. The CNC assumes the machine reference zero point (home) as the new part zero.*

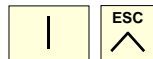


- 1 Select the axis to be homed using the alphanumeric keyboard. The CNC will highlight that axis to indicate that it is selected.

To select the numbered axes (e.g. "X1"), select any axis and then move the selection until positioning on the desired one. The focus moves with the [↑][↓] keys.



- 2 Press the homing key [ZERO]. The CNC will display the "1" symbol in the numeric area indicating that a home search will take place.



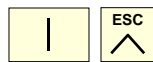
- 3 Press [START] to go ahead with the home search or [ESC] to cancel the operation.

#### Automatic home search (with subroutine).

This homing method is only available if the machine manufacturer has previously defined a homing subroutine.



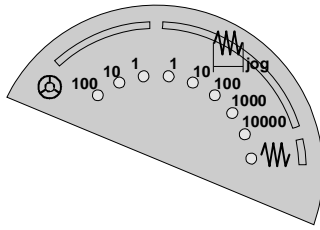
- 1 Press the homing key [ZERO]. The CNC will display a dialog box requesting confirmation to execute the home search.



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

## 2.2.2 Jog

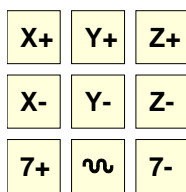
The axes may be jog using the JOG keyboard on the operator panel. The type of jog is selected with the jog selector switch on the operator panel.



### The jog keyboard and the feedrate selector

#### JOG keypad.

There are two types of jog keyboards depending on the behavior of the keys.



The keypad has two keys for each axis. One to jog the axis in the positive direction and another one to move it in the negative direction.

To move a single axis, press the axis key and the one for its jogging direction.

The keypad has a key for each axis and two keys for moving direction, common to all the axes.

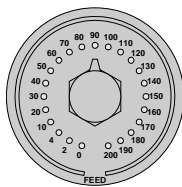
To jog an axis requires activating both the axis key and the moving direction. There are two options, depending on how the jog keyboard has been configured.

- The axis will move while both keys are pressed, the axis key and the direction key.
- When pressing the axis key, the key remains active. The axis will move while the direction key is kept pressed. To de-select the axis, press [ESC] or [STOP].

#### User keys as jog keys

The CNC offers the OEM the possibility to enable the user keys as jog keys. The user keys defined this way behave like the jog keys.

#### Feedrate selector.



The movement is carried out at the feedrate defined by the OEM. The feedrate may be varied between 0% and 200% using the feedrate override switch on the operator panel.

### Movement in continuous jog.

In continuous jog, the axes keep moving while the jog keyboard is acted upon. Continuous jog allows moving several axes at the same time.

- 1 Turn the jog selector switch of the operator panel to the continuous jog position on the dial.
- 2 Jog the desired axis using the JOG panel (keypad). If while moving, a second axis is selected, the new one will move at the same time and under the same conditions.



If while the axes are moving, the rapid key is pressed, the axes will move at the rapid rate set by the machine manufacturer. This feedrate will be applied while that key is kept pressed and, when released, the axes will recover their previous feedrate. This rapid rate may be varied between 0% and 200% with the feedrate override switch on the operator panel.

## Movement in incremental jog.

In incremental jog, the axis moves a specific distance every time the key is pressed. In incremental jog, the axes may be jogged simultaneously.

- 1 Turn the jog selector switch of the operator panel to one of the incremental jog positions. Each position will move the axis a fixed distance; the typical values are the following.

Position.	Movement for each key push.
1	0.001 mm or 0.0001 inch.
10	0.010 mm or 0.0010 inches.
100	0.100 mm or 0.0100 inches.
1000	1.000 mm or 0.1000 inches.
10000	10.000 mm or 1.0000 inches.

- 2 Jog the desired axis using the JOG panel (keypad). Every time the JOG panel is acted upon, the axis will move the distance indicated on the dial of the jog selector switch. If while moving, a second axis is selected, the new one will move at the same time and under the same conditions.

# 2.

**OPERATING IN JOG MODE**  
Operations with the axes.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)



### 2.2.3 Jogging the axes with handwheels

Electronic handwheels may be used to move the axes. Depending on the type of handwheel, The CNC may have general handwheels to move any axis or individual handwheels that will only move their associated axes.

To move the axes with the handwheels, turn the jog selector switch of the operator panel to one of the handwheel positions. Every position indicates the multiplying factor applied to the handwheel pulses; the typical values are the following.

Position.	Movement per revolution of the handwheel.
1	0.100 mm or 0.0100 inches.
10	1.000 mm or 0.1000 inches.
100	10.000 mm or 1.0000 inches.

Once the desired resolution has been selected and depending on the type of handwheel being used, general or individual, proceed as follows.

#### General handwheel

The CNC may have several general handwheels. The general handwheel is not associated with any axis in particular, it may be used to move any axis of the machine even if it has an individual handwheel associated with it.

- If there are several axes selected in handwheel mode, the general handwheel will move all of them.
- If an axis has been selected which has an individual handwheel selected with it, this axis may be moved with the general handwheel, with the individual one or with both at the same time. When using both handwheels simultaneously, the CNC will add or subtract the pulses provided by both handwheels depending on which direction they are turned.
- If the CNC has several general handwheels, any of them can move the axes selected in handwheel mode. When using several handwheels simultaneously, each axis involved will be applied the sum of the increments of all the handwheels.

These are the steps to follow for moving one or several axes with the general handwheel.

- 1 Select the axis or axes to be jogged. The CNC will highlight the selected axes. When selecting an axis or quitting the handwheel mode using the movement selector, the previous one is automatically deselected.
- 2 Once the axis has been selected, the CNC will move it as the handwheel is turned depending on the setting of the selector switch and on the turning direction of the handwheel.

The feedrate depends on how fast the handwheel is turned.

#### Selecting the axes to be jogged

There are two ways to select the axes.

- 1 On the JOG keyboard, press one of the keys for the axis to be jogged. Selecting an axis de-selects the previous one. To select several axes, press one of the keys of each key at the same time.

An axis needs not belong to the active channel in order to be selected. An axis from one channel may be set in handwheel mode from another channel, if the channel of the axis is also in jog mode.

- 2 When using a handwheel with a push-button, the push-button may be used to select, sequentially, the axes to be jogged. Pushing the button selects the first one of the axes being displayed. If an axis has already been selected, it de-selects it and selects the next one. If it was the last one, it selects the first one again.

Only the axes being displayed in the active channel may be selected, regardless of the channel they belong to. The axes of another channel cannot be selected if they are not being displayed.

An axis is de-selected when quitting the handwheel mode using the movement selector and after a reset. If an axis has been set in handwheel mode from the PLC, it can only be deactivated from the PLC; a reset does not deactivate it.

## 2.

## Selecting an axis from the automatic mode

When having only one channel, if while in automatic mode, you set the switch in handwheel mode and select an axis, when going to jog mode, it maintains the selected axis.

## Individual handwheel

The CNC can have several individual handwheels, where each of them is associated with a particular axis. The CNC moves each axis as its relevant handwheel is turned depending on the setting of the selector switch and on the turning direction of the handwheel.



In handwheel mode, this symbol next to an axis indicates that the axis has an individual handwheel associated with it.

When moving several axes simultaneously using handwheels, all the axes having their own handwheel plus the ones that may be selected with the general handwheel may be involved. When moving several axes at the same time, the feedrate of each axis depends on how fast its associated handwheel is turned.



*It may happen that depending on the turning speed and the selector position, the CNC be demanded a faster feedrate than the maximum allowed. In that case, the CNC will move the axis the indicated distance but at the maximum feedrate allowed.*

## Feed handwheel.

Usually, when machining a part for the first time, the feedrate is controlled by the switch on the operator panel. The "feed handwheel" allows using one of the handwheels of the machine to control that feedrate depending on how fast the handwheel is turned.



*This feature must be managed from the PLC. Usually, this feature is turned on and off using an external push button or key configured for that purpose.*

# 2.

**OPERATING IN JOG MODE**  
Operations with the axes.

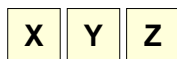


FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 2.2.4 Moving an axis to a particular position (coordinate)



- 1 Select the axis to be moved using the alphanumeric keyboard. The CNC will highlight that axis to indicate that it is selected.

To select the numbered axes (e.g. "X1"), select any axis and then move the selection until positioning on the desired one. The focus moves with the [↑][↓] keys.

- 2 Enter the coordinate of the target point.



- 3 Press [START] to execute the movement or [ESC] to cancel the operation.

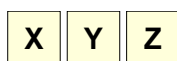
### Feedrate behavior

The moving feedrate depends on whether G00 or G01 is active. This feedrate may be varied between 0% and 200% using the feedrate override switch on the operator panel. The percentage will be applied on to all the movements carried out in G00 and in G01.

- If G00 is active, the movement is carried out at the rapid rate defined by the machine manufacturer.
- If G01 is active, the movement is carried out at the active feedrate. If no feedrate is active, the movement is executed at the feedrate defined by the machine manufacturer.

## 2.2.5 Coordinate preset

The coordinates must be preset one axis at a time. The preset may be canceled by homing the axes one by one or by means of function "G53".



- 1 Use the alphanumeric keyboard to select the axis whose position value (coordinate) is to be preset. The CNC will highlight that axis to indicate that it is selected.

To select the numbered axes (e.g. "X1"), select any axis and then move the selection until positioning on the desired one. The focus moves with the [↑][↓] keys.

- 2 Key in the desired preset value.

- 3 Press [ENTER] to preset the entered value or [ESC] to cancel the operation.

2.

OPERATING IN JOG MODE  
Operations with the axes.

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 2.3 Spindle control





### Displaying the data of several spindles.

The screen only shows the data on one spindle. If there are several spindles in the channel, the data on the next spindle may be displayed by pressing the "S" key. The first push is to program the turning speed, the second one shows the data on the second spindle and so on.

### Spindle control


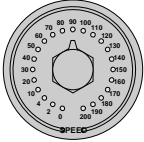
The spindle may be controlled manually using the following keys of the operator panel. The keys always refer to the master spindle of the active channel.

The spindle speed should be set (in the MDI mode) before selecting the turning direction, thus avoiding a sudden start of the spindle when setting an "S" because the turning direction was active.

Key.	Meaning.
	Start the spindle clockwise (same as M03 function) at the active speed. The CNC shows the M03 function in the program history.
	Start the spindle counterclockwise (same as M04 function) at the active speed. The CNC shows the M04 function in the program history.
	Stop the spindle (same as M05 function). The CNC shows the M05 function in the program history.
	Orient the spindle (same as M19 function). The CNC shows the M19 function in the program history.

### Vary the speed override from the operator panel.

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

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

2.

OPERATING IN JOG MODE  
Spindle control



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 2.4 Tool selection and tool change

The tool located in the spindle may be changed in manual mode. Proceed as follows.

- 1 Press [T] at the alphanumeric keyboard. The CNC will highlight the current tool indicating that it is selected.
- 2 Key in the number of the tool to be placed in the spindle.
- 3 Press [START] to execute the tool change or [ESC] to cancel the operation.

## 2.5 Setting the feedrate and spindle speed.

### Setting a new feedrate in the channel.

The feedrate set in jog mode is only applied in that work mode and for the active channel. When setting a new feedrate in the MDI/MDA mode, it will become the new feedrate for the jog and automatic modes.

Follow these steps to set a new feedrate.

- 1 Press [F] at the alphanumeric keyboard. The CNC will highlight the relevant data indicating that it is selected.
- 2 Enter the new feedrate.
- 3 Press [START] to assume the entered value or [ESC] to cancel the operation.

### Setting a new spindle speed.

The spindle speed set in the jog mode is applied to the spindle displayed at the time. If there are several spindles in the channel, the rest of the spindles may be displayed by pressing the [S] key. The spindle speed set in jog mode is maintained when switching to automatic mode and vice versa.

Follow these steps to set a new spindle speed.

- 1 Press [S] at the alphanumeric keyboard until selecting the desired spindle. When pressing this key for the first time, the CNC will highlight the relevant data indicating that it is selected.
- 2 Enter the new spindle speed.
- 3 Press [START] to assume the entered value or [ESC] to cancel the operation.

# 2.

OPERATING IN JOG MODE  
Tool selection and tool change

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 2.6 Setting and activating the zero offsets and the fixture offsets.

In jog mode, it is possible to save the active offset in the zero offset table or in the fixture offset table (zero offset, coordinate presetting, etc.) and to activate a zero offset already defined in the tables.



This softkey shows the zero offsets and the fixture offsets of the system and their value in each axis of the channel. This list is a brief information of the zero offset tables and fixture offset tables and any change made in jog mode also affects those tables.

### Loading a new zero offset or fixture offset into the table.

With an active offset, use the cursor to select an offset from the list and press [ENTER] to save the current offset in that zero offset. The position of all the axes of the channel are updated at the selected zero offset.

### Applying a zero offset or fixture offset stored in the table.

Use the cursor to select a zero offset or fixture offset from the list and press the [START] key to activate. The new zero offset is applied to all the axes of the channel.

# 2.

**OPERATING IN JOG MODE**

Setting and activating the zero offsets and the fixture offsets.



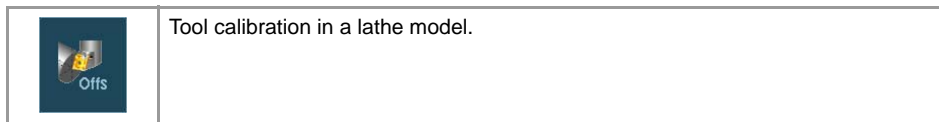
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 2.7 Tool calibration

Tool calibration is available in the jog mode. The softkey to access tool calibration will be different depending on the software installed (lathe model or mill model). To quit the calibration mode and return to jog mode, press the [ESC] key.



The CNC offers in both models the possibility to calibrate lathe tools and milling tools. The CNC will show the necessary data and will update the help graphics according to the selected tool.

### Types of calibration

There are several ways to calibrate a tool. Some ways are only available when using a table-top probe.



Only manual calibration is possible when not using a table-top probe. All types of calibration are available when using a table-top probe. The different calibration methods may be selected from the vertical softkey menu.

The active kinematics are taken into account and do not prevent tool calibration in this mode. Manual or semi-automatic calibration will not be possible if a coordinate ( #CS or #ACS) transformation is active or when either the RTCP or TLC function is active.

#### Manual calibration. Calibration without a probe.

It is done without the table-top probe. A reference part is required to calibrate the tool. All the movements are carried out manually.

#### Semi-automatic calibration. Calibration with a probe.

This calibration mode is available when using a table-top probe. The positioning movements are carried out manually and the CNC executes the probing movements.

#### Automatic calibration. Calibration with a probe and a canned cycle.

This calibration mode is available when using a table-top probe. The CNC executes all the movements using the calibration canned cycle #PROBE.

### Probe selection

Two probes may be configured at the CNC. The probe active at the time is used for calibration. The active probe may be changed via part-program or MDI using the instruction #SELECT PROBE.

```
#SELECT PROBE [1]
  Selects the first probe.
#SELECT PROBE [2]
  Selects the second probe.
```

# 2.

OPERATING IN JOG MODE  
Tool calibration

**FAGOR** 

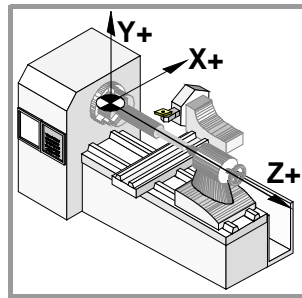
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## Geometrical configuration of the axes on a lathe: "plane" or "trihedron".

At the lathe model, the geometrical configuration of the axes may be either of the "plane" or "trihedron" type depending on the availability of a third main axis, usually the  $\cdot Y \cdot$  axis. The different calibration modes adapt to the current configuration showing the necessary data for each one of them.

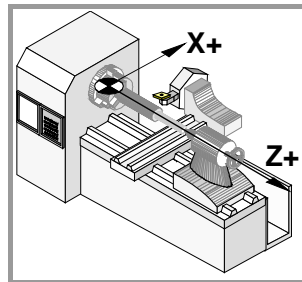


### Geometrical configuration of "trihedron" type axes.

It is the typical configuration of a milling machine or of a lathe that has a third main axis ( $\cdot Y \cdot$  axis).

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

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



### Geometrical configuration of "plane" type axes.

It is the typical configuration of a lathe.

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

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

The work plane is always G18; the plane cannot be changed via part-program.

### Configuration of "plane" type axes. The longitudinal axis.

In this configuration, the second axis of the channel is considered as longitudinal axis. If the X (first) and Z (second) axes have been defined, the work plane will be the ZX and Z will be the longitudinal axis. Tool length compensation is applied on this longitudinal axis when using milling tools. With lathe tools, tool length compensation is applied on all the axes where a tool offset has been defined.

When using milling tools on a lathe, the longitudinal compensation axis may be changed by means of the #TOOL AX instruction or the G20 function.

2.

OPERATING IN JOG MODE  
Tool calibration



FAGOR AUTOMATION

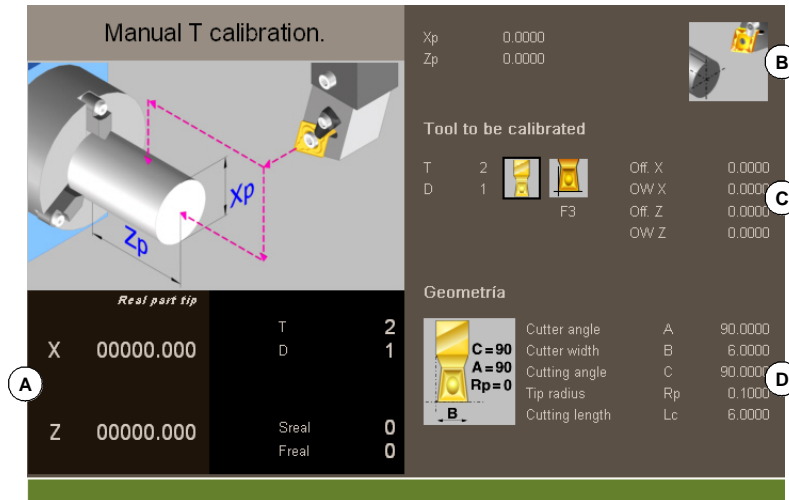
CNC 8060  
CNC 8065

(REF: 1709)



## 2.7.1 Manual calibration. Calibration without a probe

In this mode, only the active tool can be calibrated and it may be a milling tool or a lathe tool. The CNC will show the necessary data and will update the help graphics according to the selected tool.



- A Machine data. Position of the axes, tool and active tool offset, real spindle speed and real feedrate of the axes.
- B Data of the part used for calibration and drawing showing that calibration is possible. If the window does not show this drawing, some of the data is missing.
- C Necessary data for calibration.
- D Tool data.

### Tool calibration

Since there is no probe, a reference part is required to calibrate the tool. The calibration consists in moving the tool manually until it touches the part and then validating the calibration on each axis. After validating them, the new values are saved in the tool table.

#### Selecting a tool

The tool and the active tool offset may be changed from the calibration mode. After defining the new tool or tool offset in the cycle data, press [CYCLE START] and the CNC will execute the tool change.

Bear in mind that if the defined tool is the active tool, when pressing [START] the CNC assumes the values that the offset has at the time.

#### Tool calibration in a lathe model (Configuration of plane type axes).

For lathe and mill tools, it calibrates the tool offsets on each axis. When validating the calibration in one of the offsets, the wear of that offset is reset to zero.

#### Tool calibration in a lathe model (Configuration of trihedron type axes).

- For the lathe tools, it calibrates the tool offsets on each axis. When validating the calibration in one of the offsets, the wear of that offset is reset to zero.
- There are two options for the milling tools and may be selected with the following icons.






Tool length calibration. This option may be used to update the length value and resets the wear value to zero. It also updates the tool table data.



Tool offset calibration. This option may be used to update the value of the offsets on each axis. The offset wears are set to zero.

**Validating the calibration.**

They are validated from the vertical softkey menu. Once the tool has been calibrated, when pressing [START] the CNC assumes the new values of the offset.

Softkey.	Description.
	Validating the length calibration of a milling tool.
	Validating the offsets of a milling tool.
	Validating the offsets of a lathe tool.

When on a lathe the axis have a "trihedron" type configuration, the calibration on the axis perpendicular to the work plane is done using the horizontal softkey menu.

**Definition of data**

To define the data, place the focus on the relevant data, key in the desired value and press [ENTER].

**For a lathe tool.**

The nomenclature of the axes depends on the geometrical configuration of the "plane" or "trihedron" axes. For a "plane" configuration, the names of the axes assume the DIN standard for lathes; the Z axis as the abscissa axis and the X axis as the ordinate axis.

Data	Meaning
Zp Xp	Dimensions of the reference part being used in the calibration. These coordinates are referred to the main axes of the tool.
T	Tool to be calibrated.
D	Tool offset to be calibrated.
Off Y Off X	Tool offsets on each axis.
Lw	Offset wear on each axis.

When a lathe has a third axis perpendicular to the work plane ("trihedron" geometrical configuration), the CNC will also show its data and calibration will be possible on that axis. The data of the third axis may be hidden or shown using the horizontal softkey menu.

**For a milling tool.**

Data	Meaning
Zp	Coordinate of the reference part being used in the calibration. This coordinate is referred to the longitudinal axis of the tool.
T	Tool to be calibrated.
D	Tool offset to be calibrated.
L	Tool length.
Lw	Length wear.
R	Tool radius.
Rw	Radius wear.

**2.**

OPERATING IN JOG MODE  
Tool calibration



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## Tool calibration steps

To calibrate the length, follow these steps.

- 1 Define the dimensions of the reference part being used in the calibration.
- 2 Select the tool and the offset to be calibrated. After the selection, the CNC shows the dimensions defined in the tool table for that offset.

To calibrate a tool, it must be the active tool. When selecting a tool and pressing [ENTER], the CNC only shows the data for that tool. Press [CYCLE START] for the CNC to make the tool change so it becomes the active tool. See "[Selecting a tool](#)" on page 49.

- 3 Calibrate the tool. Approach the tool manually until touching the part and then validate the calibration using the softkey menu.

After validating the calibration, it updates the values and initializes the wear value to zero. Then, the new values are saved in the tool table.

- 4 Press [START] for the CNC to assume the new values of the offset.

To calibrate another tool, repeat steps 2 and 3.

## Considerations for the offsets and their wear.

It must be borne in mind that the offset of a tool on an axis is the distance between the base of the tool and its tip (nose). This means that when calculating the offset of a milling tool on an axis that includes the radius dimension, that radius is included in the offset. The same is true for the tool length.

When calibrating the offsets of a milling tool, the length value is deleted but not the radius value.

### Sign criteria for the offsets and their wear.

The sign criterion for the offsets and their wear is established by machine parameter TOOLOFSG.

TOOLOFSG	Meaning.
Negative.	Tool calibration returns a negative offset. The offset wear must be entered with a positive value.
Positive.	Tool calibration returns a positive offset. The offset wear must be entered with a negative value.

### Entering wear values incrementally or absolutely.

In the tool table, it is possible to define whether the wear value being entered must be incremental or absolute.

Using incremental wear, the value entered by the user will be added (or subtracted if it is negative) to the absolute value of the wear. After pressing [ENTER] to accept the new value, the wear field will show the resulting absolute value.

Initial wear	Incremental wear	Total wear
1	0.2	1.2
1	-0.2	0.8
-1	0.2	-0.8
-1	-0.2	-1.2

2.

OPERATING IN JOG MODE  
Tool calibration



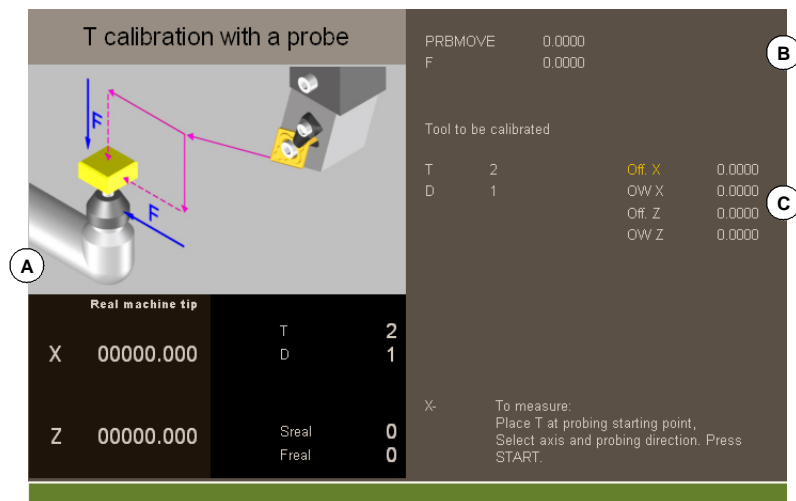
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 2.7.2 Semi-automatic calibration. Calibration with a probe

This option is only available when using a tabletop probe installed on the machine. On a lathe model, it may be used to calibrate the offsets of any tool.



- A Machine data. Position of the axes, tool and active tool offset, real spindle speed and real feedrate of the axes.
- B Data of probing movement.
- C Necessary data for calibration.

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

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

### Tool calibration

The calibration consists in manually approaching the tool to the probe and then command the CNC to execute the probing movement. The CNC will move the tool on the selected axis until touching the probe. After touching the probe, it concludes the calibration on that axis and updates the values.

#### Selecting a tool

The tool and the active tool offset may be changed from the calibration mode. After defining the new tool or tool offset in the cycle data, press [CYCLE START] and the CNC will execute the tool change.

Bear in mind that in this calibration mode, the [CYCLE START] key has two functions. If a new tool has been selected, it executes the tool change. If the selected tool is the active one, pressing [CYCLE START] initiates the calibration.

#### Tool calibration

For lathe and mill tools, it calibrates the tool offsets on each axis. When validating the calibration in one of the offsets, the wear of that offset is reset to zero.

#### Validating the calibration

Use the horizontal softkey menu to select the axis and the moving direction for the calibration. Once selected and after placing the tool in the spindle, press [CYCLE START] to start the calibration. The tool will move in the indicated direction until touching the probe and it will then conclude the calibration updating the tool data with the measured values.

Once the tool has been calibrated, the CNC shows a message proposing to press [START] so the CNC assumes the new values of the offset. When pressing [START] while this message is displayed, the CNC assumes the new values of the offset; if the message is not displayed, pressing [START] executes the probing movement again.

Once a movement has been selected, the window will show a help drawing indicating the type of calibration to be done.

2.

OPERATING IN JOG MODE  
Tool calibration



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## Definition of data

To define the data, place the focus on the relevant data, key in the desired value and press [ENTER].

Data	Meaning
PRBMOVE	Maximum probing distance. If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes.
F	Probing feedrate.
T	Tool to be calibrated.
D	Tool offset to be calibrated.
Off X Off Z	Tool offsets on each axis.

## Tool calibration steps

To calibrate the tool, follow these steps:

- 1 Define the probing distance and feedrate. If the feedrate is not defined, the probing movement will be made at the feedrate set by the OEM.
- 2 Select the tool and the offset to be calibrated. After the selection, the CNC shows the dimensions defined in the tool table for that offset.  
To calibrate a tool, it must be the active tool. When selecting a tool and pressing [ENTER], the CNC only shows the data for that tool. Press [CYCLE START] for the CNC to make the tool change so it becomes the active tool. See "[Tool calibration](#)" on page 52.
- 3 Manually approach the tool to the probe until it is placed on the path that will be used for probing.
- 4 Calibrate the tool. Select the axis and the probing direction on the softkey menu and press [START].  
The probe moves in parallel to the axis and in the selected direction until touching the probe. It updates the measured value and resets the wear value to zero. The data is stored in the tool table.
- 5 Press [START] again for the CNC to assume the new values of the offset. For the new values to be assumed, press [START] while the bottom message is displayed; otherwise, it executes the probing movement again.

## Considerations for the offsets and their wear.

It must be borne in mind that the offset of a tool on an axis is the distance between the base of the tool and its tip (nose). This means that when calculating the offset of a milling tool on an axis that includes the radius dimension, that radius is included in the offset. The same is true for the tool length.

When calibrating the offsets of a milling tool, the length value is deleted but not the radius value.

### Sign criteria for the offsets and their wear.

The sign criterion for the offsets and their wear is established by machine parameter TOOLOFSG.

TOOLOFSG	Meaning.
Negative.	Tool calibration returns a negative offset. The offset wear must be entered with a positive value.
Positive.	Tool calibration returns a positive offset. The offset wear must be entered with a negative value.

### Entering wear values incrementally or absolutely.

In the tool table, it is possible to define whether the wear value being entered must be incremental or absolute.

2.

OPERATING IN JOG MODE  
Tool calibration

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

Using incremental wear, the value entered by the user will be added (or subtracted if it is negative) to the absolute value of the wear. After pressing [ENTER] to accept the new value, the wear field will show the resulting absolute value.

Initial wear	Incremental wear	Total wear
1	0.2	1.2
1	-0.2	0.8
-1	0.2	-0.8
-1	-0.2	-1.2

# 2.

## OPERATING IN JOG MODE

Tool calibration

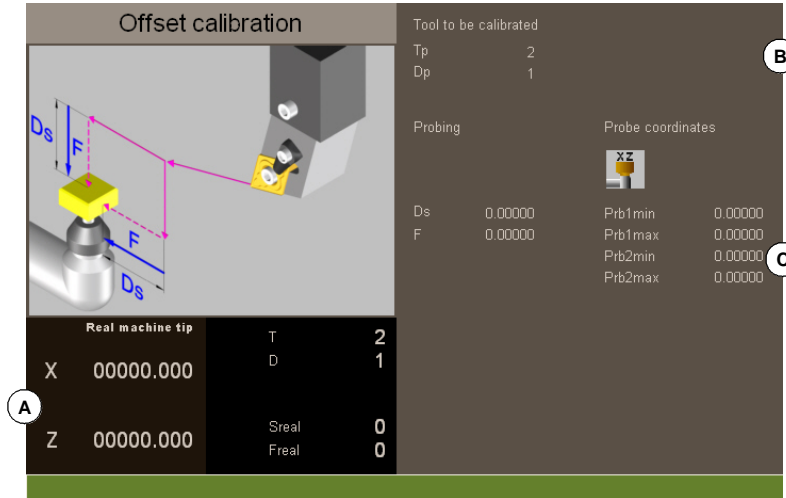


CNC 8060  
CNC 8065

(REF: 1709)

### 2.7.3 Automatic calibration with a probe and a canned cycle (“plane” geometric configuration)

This option is only available when using a tabletop probe installed on the machine. This mode may be used to calibrate both milling and lather tools. The CNC will show the necessary data and will update the help graphics according to the selected tool.



- A Machine data. Position of the axes, tool and active tool offset, real spindle speed and real feedrate of the axes.
- B Tool to be calibrated.
- C Data for probe calibration and position.

### Tool calibration

The calibration is done using a probing canned cycle. The CNC moves the tool until touching the probe and validates the calibration on each axis. The tool is calibrated on the two axes of the plane.

The calibration begins when pressing the [CYCLE START] key. When the CNC finishes the calibration on the selected axes, it updates the dimensions and the wears. Then, the new values are saved in the tool table.

#### Selecting a tool

In this calibration mode, the cycle itself changes the tool and the tool offset. There is no need to previously place the tool in the spindle.

Bear in mind that pressing the [CYCLE START] key starts the calibration cycle.

#### Select another position for the probe.



The calibration uses the probe position defined in the machine parameters. Optionally, another position may be defined for the probe which will only be valid for the defined calibration. The new position does not affect the values defined in the machine parameters.

### Tool calibration

For milling and lathe tools, it calibrates the tool offsets on each axis. The offset wears are set to zero.

## Definition of data

To define the data, place the focus on the relevant data, key in the desired value and press [ENTER]. To change icons, place the focus on it and press [SPACE].

Data	Meaning
T	Tool to be calibrated.
D	Tool offset to be calibrated.
Ds	Safety distance.
F	Probing feedrate. If not defined, the movements are carried out at the default feedrate, set by the machine manufacturer.
PRB1MAX ... PRB2MIN	Probe position. The values defined here are only taken into account during the calibration cycle; they do not modify the machine parameter values.

## Tool calibration steps

To calibrate the length, follow these steps.

- 1 Select the tool and the offset to be calibrated. It is not necessary to insert the tool into the spindle; the CNC carries out this operation if necessary.
- 2 Define the data defining the calibration.
- 3 Press the [CYCLE START] key to start the calibration. The CNC calibrates the tool making all the necessary movements; there is no need to manually approach the tool. If necessary, the CNC makes the tool change.
- 4 After the calibration It updates the tool table data.

# 2.

OPERATING IN JOG MODE  
Tool calibration



FAGOR AUTOMATION

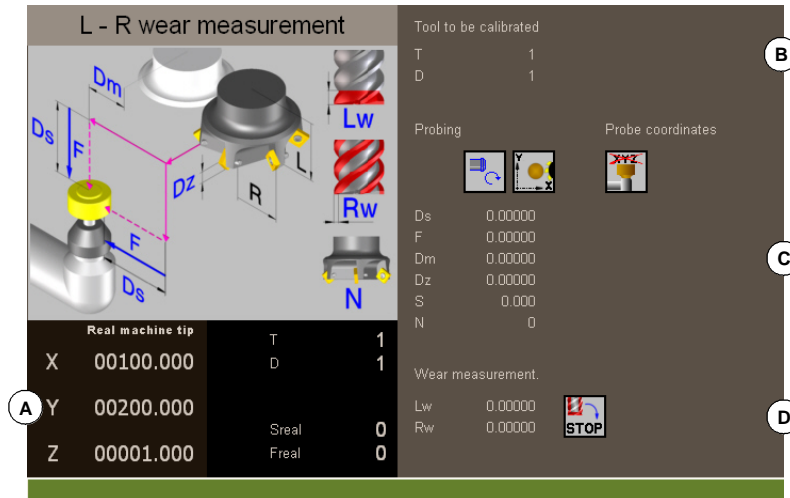
CNC 8060  
CNC 8065

(REF: 1709)



## 2.7.4 Automatic calibration with a probe and a canned cycle (“trihedron” geometric configuration)

This option is only available when using a tabletop probe installed on the machine. This mode may be used to calibrate both milling and lather tools. The CNC will show the necessary data and will update the help graphics according to the selected tool.



- A Machine data. Position of the axes, tool and active tool offset, real spindle speed and real feedrate of the axes.
- B Tool to be calibrated.
- C Data for probe calibration and position.
- D Data for tool wear measurement.

### Tool calibration

The calibration is done using a probing canned cycle. The CNC moves the tool until touching the probe and validates the calibration on each axis. The tool may be calibrated on both axes of the plane or on the three axes of the trihedron.

The calibration begins when pressing the [CYCLE START] key. When the CNC finishes the calibration on the selected axes, it updates the tool table with the measured values. Also, the CNC assumes the new values.

#### Selecting a tool

In this calibration mode, the cycle itself changes the tool and the tool offset. There is no need to previously place the tool in the spindle.

Bear in mind that pressing the [CYCLE START] key starts the calibration cycle.

#### Select another position for the probe.



The calibration uses the probe position defined in the machine parameters. Optionally, another position may be defined for the probe which will only be valid for the defined calibration. The new position does not affect the values defined in the machine parameters.

#### Tool calibration

There are two options for the milling tools and may be selected with the following icons.



- Calibrate the offsets and resets the wears to zero.



- Calibrate the length and radius and measure the wears.

For the lathe tools, it calibrates the tool offsets on each axis. The offset wears are set to zero.

2.

OPERATING IN JOG MODE  
Tool calibration

**FAGOR**

FAGOR AUTOMATION

CNC 8060  
CNC 8065




(REF: 1709)

## Definition of data


To define the data, place the focus on the relevant data, key in the desired value and press [ENTER]. To change icons, place the focus on it and press [SPACE].

### To calibrate the length, radius and wears of a milling tool.

The data shown depends on the calibration option selected with the horizontal softkey menu. This menu may be used to select the length and/or radius calibration and whether to calculate their wear or not. If the wears are not calculated, they are reset to zero after the calibration.

Data	Meaning
T	Tool to be calibrated.
D	Tool offset to be calibrated.
Ds	Safety distance.
F	Probing feedrate. If not defined, the movements are carried out at the default feedrate, set by the machine manufacturer.
N	Number of cutters of the tool. If defined with a -0- value, the CNC knows the location of a cutter and it will only make the movement once. The spindle turning speed must be -0-. If defined with a value other than -0-, all cutters will be calibrated. The CNC makes an initial movement to locate a cutter; then, stops the spindle and makes a precise measurement of each cutter. It is necessary to define the spindle speed and the Dm distance.
Dm	Distance the edge of the tool separates from the center of the probe to position the next cutter.
S	Spindle speed.
	Probe side to be touched.
 	Behavior when exceeding the maximum wear permitted; reject the tool or change it with another one from the same family.
Lw	Maximum length wear allowed.
Rw	Maximum radius wear allowed.
PRB1MAX ... PRB2MIN	Probe position. The values defined here are only taken into account during the calibration cycle; they do not modify the machine parameter values.

### To calibrate the offsets of a milling or lathe tool.

Data	Meaning
T	Tool to be calibrated.
D	Tool offset to be calibrated.
Ds	Safety distance.
F	Probing feedrate. If not defined, the movements are carried out at the default feedrate, set by the machine manufacturer.
PRB1MAX ... PRB2MIN	Probe position. The values defined here are only taken into account during the calibration cycle; they do not modify the machine parameter values.
	This icon sets the number of axes to calibrate on.

2.

OPERATING IN JOG MODE  
Tool calibration



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## Tool calibration steps

To calibrate the length, follow these steps.

- 1 Select the tool and the offset to be calibrated. It is not necessary to insert the tool into the spindle; the CNC carries out this operation if necessary.
- 2 Define the data defining the calibration. To calibrate a milling tool, use the horizontal softkey menu to select the desired operation.
- 3 Press the [CYCLE START] key to start the calibration. The CNC calibrates the tool making all the necessary movements; there is no need to manually approach the tool. If necessary, the CNC makes the tool change.
- 4 After the calibration It updates the tool table data.

2.

OPERATING IN JOG MODE  
Tool calibration

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**2.**

**OPERATING IN JOG MODE**

Tool calibration



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 3.1 Canned cycles available in the editor.

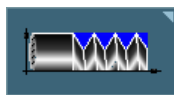
### Select the machining cycles.

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.



#### Turning.

Simple turning, turning with vertex rounding, simple facing, facing with vertex rounding, vertex chamfering, chamfering between points, vertex rounding and rounding between points.



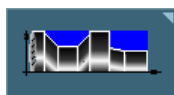
#### Tapping.

Longitudinal threading, taper threading, face threading, thread repair and multiple-entry (multi-start) threads.



#### Grooving.

Simple longitudinal grooving, simple face grooving, inclined longitudinal grooving, inclined face grooving and cut-off.



#### Profiles.

Point-to-point turning, profile turning, ZC plane profile and XC plane profile.



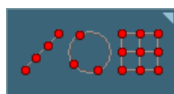
#### Z axis machining.

Drilling, tapping, multiple drilling, multiple threading, multiple slot milling



#### Positioning.

Positioning and positioning with M functions.



#### Multiple machining.

Points in line, points in arc, points in a rectangular pattern, points in a grid pattern, points in a random pattern (several points defined by the user).

### Activating the Teach-in mode.



The "+" softkey shows the softkey to activate the teach-in mode that allows jogging the axes of the machine and entering the actual (real) position of the axes into the data. See "[3.3 Teach-in mode.](#)" on page 63.

### Configuring the cycle editor.



The "+" softkey shows the softkey to configure some of the options of the cycles of the editor.

### Accessing the probing cycles.



The "+" softkey shows the softkey to access the probing cycles or the mill model cycles (if they are available).

### 3.2 Configuring the cycle editor.



The "+" softkey shows the softkey to configure some of the options of the cycles of the editor.

#### Programming M functions in each operation.

Enable the programming of M functions in the canned cycles, to execute them before each machining operation. This permits, for example, to execute subroutines associated with M functions before the various operations.

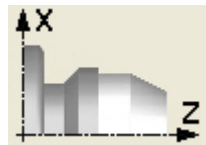
Being this option active, the editor offers in each operation of the cycle the option to edit up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



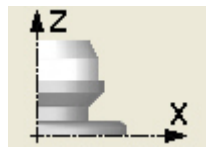
On the cycle screens, the display must be activated in order to see and define the M function data; otherwise, the data will not be displayed.

#### Select the graphics for a vertical lathe.

Enable the cycles for vertical lathe.

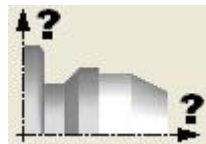


Cycle editor configured for horizontal lathe.



Cycle editor configured for vertical lathe.

#### Select axis configuration.



Setting an axis configuration for the cycle editor. The defined axis configuration is only valid for making it easier to edit the cycle because it shows the data related to the coordinates according to the chosen axis configuration.

The canned cycles have no work plane associated with them, they are executed in the current active work plane.

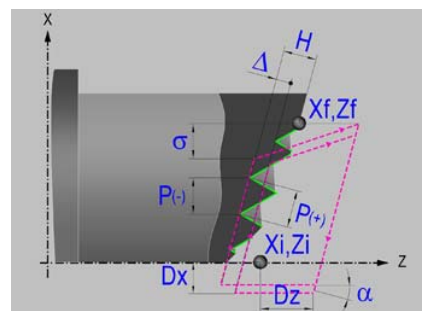
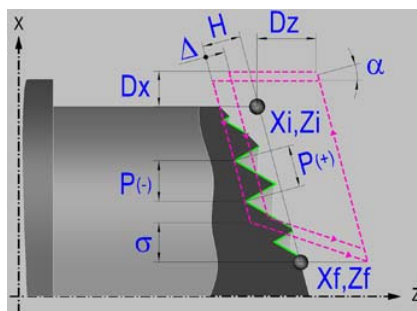
#### Invert the starting and end points in face threading.



Invert the starting and end points.



Do not invert the starting and end points.



**3.**  
**WORKING WITH OPERATIONS OR CYCLES**  
Configuring the cycle editor.



CNC 8060  
CNC 8065

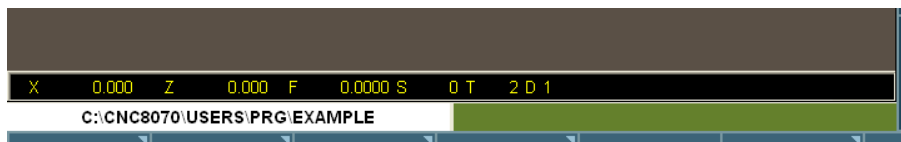
(REF: 1709)

### 3.3 Teach-in mode.



The "+" softkey shows the softkey to activate the teach-in mode that allows jogging the axes of the machine and entering the actual (real) position of the axes into the data. The reset of the data must be edited manually.

Being this mode active, the bottom of the cycle editor shows a window with the actual axis position and the active machining conditions. The information in the window cannot be configured, it is not conditioned by the configuration made in the EDISIMU mode for the teach-in mode.



When teach-in mode is active, it is possible to keep editing the data of the axes directly from the keyboard or they may be assigned the actual position of the axes. Both editing methods may be used indistinctly, even while defining a cycle. To assign the position of its axis to a data, proceed as follows:

- 1 Select one of the data with the cursor.
- 2 Move the axes to the desired position using the jog keys, the handwheels or the MDI/MDA mode.
- 3 Press the [RECALL] key. The editor enters the actual position of the corresponding axis into the selected data.

3.

WORKING WITH OPERATIONS OR CYCLES  
Teach-in mode.

## 3.4 Selecting data, profiles and icons

### Data selection.

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 [←] [→] [↑] [↓] 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 feedrates, [T] for tools, etc. Every time the same key is pressed, it selects the next data of the same type.

### Data entry.

Place the cursor in the corresponding window, key in the desired value and press [ENTER]. If [ENTER] is not pressed, the new value will not be assumed.

If the Teach-in mode is selected, the current position of the machine may be associated with a coordinate. Place the cursor in the relevant window and press the [RECALL] key.

For the X axis parameters, it will take the coordinate of the first axis of the channel where the edit-simulation mode is active. For the Y axis parameters, the coordinate of the second axis and for the Z axis parameters, the coordinate of the third one.

### Changing the state of an icon.

Place the cursor on the desired icon and press the space bar.

### Select - Define a profile.

To select or modify a profile, the corresponding data must be selected; i.e. it must have the editing focus on it.

- To select an existing profile, press the [↓] key to expand the list of defined profiles and select one or type its name.
- To define a new profile, write the desired name and press the [RECALL] key to access the profile editor.
- To modify an existing profile, select it from the list or write its desired name and press the [RECALL] key to access the profile editor.
- To delete a profile, press the [↓] key to expand (drop) the list of profiles and select one. Press the [DEL] key to delete it.

# 3.

WORKING WITH OPERATIONS OR CYCLES  
Selecting data, profiles and icons



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)



# CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

# 4

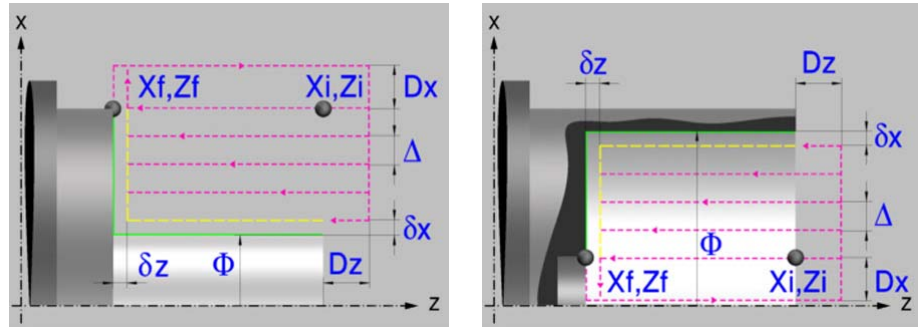
When pressing the softkey, the editor shows the cycle used last. When pressing the same softkey again, the menu shows all the cycles of the group.



- Simple turning.
- Turning with vertex rounding
- Simple facing.
- Facing with vertex rounding
- Vertex chamfer.
- Chamfer between points.
- Vertex rounding.
- Rounding between points.

## 4.1 Simple turning cycle.

The cycle allows outside and inside machining.



### Geometric parameters.

#### Inside or outside machining.



Outside machining.



Inside machining.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Coordinates of the profile's starting and final points.

$X_i, Z_i$  Coordinates of the starting point

$X_f, Z_f$  Coordinates of the end point. By default,  $X_f$  assumes the value set for  $X_i$ .

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

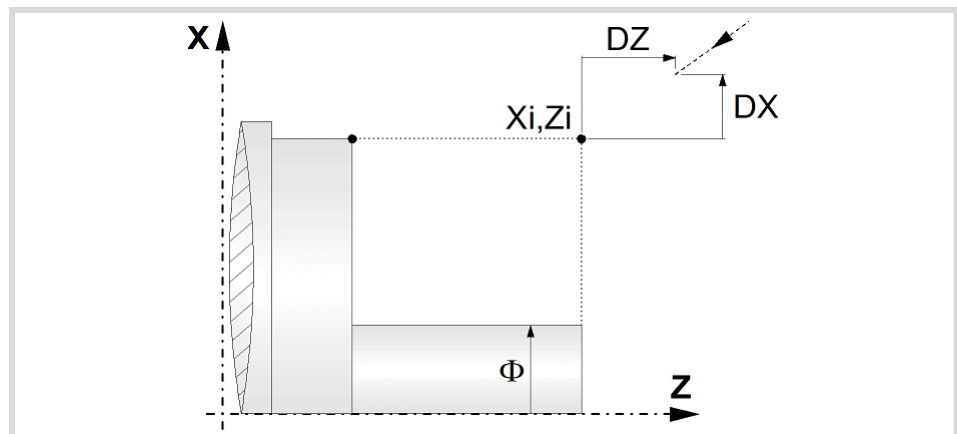
When the surface to be machined is not fully cylindrical, the CNC analyzes the coordinates on X of the starting and end points and takes as starting point on X the coordinate farthest from the final diameter.

#### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the approach point coordinate with respect to the starting point ( $X_i, Z_i$ ).

$D_x, D_z$  Safety distance.

The value of the safety distance on X is always defined in radius.



4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
Simple turning cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Final diameter.**

$\Phi$  Final diameter.

**Spindle parameters.**

**Spindle turning direction**

The cycle lets select the spindle turning direction.



Spindle clockwise.



Spindle counterclockwise.

**Coolant status.**



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

**Spindle work mode; rpm or Constant Surface Speed.**



Constant turning speed.



Constant surface speed.

**Maximum spindle speed.**

Smax Maximum spindle turning speed; if not programmed, the cycle will maintain the maximum active speed.

**Spindle range (gear).**

GEAR Spindle gear; if not programmed, the cycle assumes a value of 0. If defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Machining parameters. Roughing.**

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the roughing operation.



Do not run the roughing operation.

**Maximum machining pass.**

$\Delta$  Maximum machining pass.

- If  $\Delta$  is positive, the cycle recalculates the step so all the passes are identical with the same value as or smaller than the one programmed.
- If  $\Delta$  is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

**Machining feedrate.**

F Machining feedrate.

4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
Simple turning cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 4.

**Spindle speed.**

S Spindle turning speed (rpm) or cutting speed.

**Tool data.**

T Tool. If it is set to 0, the cycle does not execute the roughing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**Machining parameters. Finishing.**

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the finishing operation.



Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes ( $\delta x$ ,  $\delta z$ ).

**Finishing stocks.**

$\delta x$ ,  $\delta z$  Finishing stock on each axis.

**Machining feedrate.**

F Machining feedrate.

**Spindle speed.**

S Spindle turning speed (rpm) or cutting speed.

**Tool data.**

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

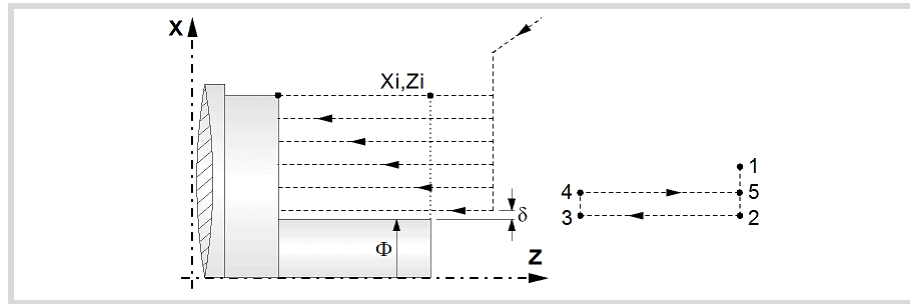
### 4.1.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The spindle starts with the speed selected and in the direction stated.
- 3 The tool approaches the starting point ( $X_i, Z_i$ ) in rapid, keeping the selected safety distance ( $D_x, D_z$ ) according to axes X and Z.
- 4 Roughing operation (if it has been programmed).

The cycle runs successive passes until reaching the final profile plus the finishing stocks. This operation is done with the conditions set for the roughing operation.

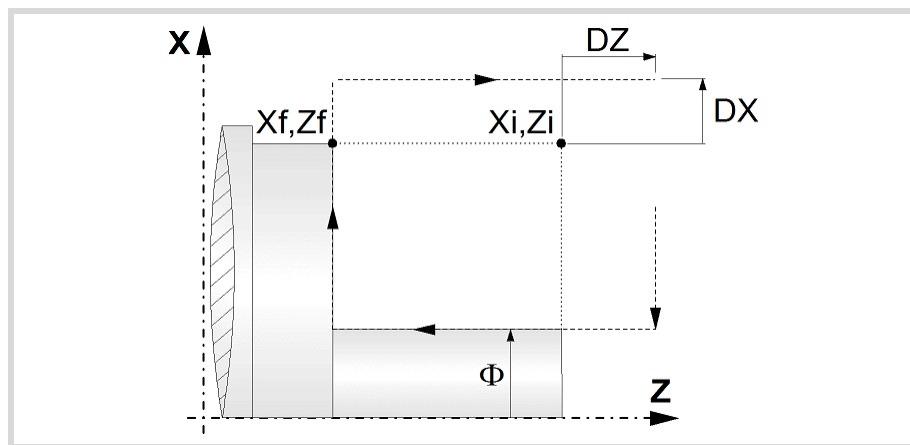
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



- 5 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle runs a single pass to remove the finishing stock. This operation is done with the conditions set for the finishing operation.



- 6 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance).
- 7 The CNC will not stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

### Considerations.

- If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.
- If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the finishing operation, the tool will move to the safety point (starting point plus the safety distance).

# 4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
Simple turning cycle.

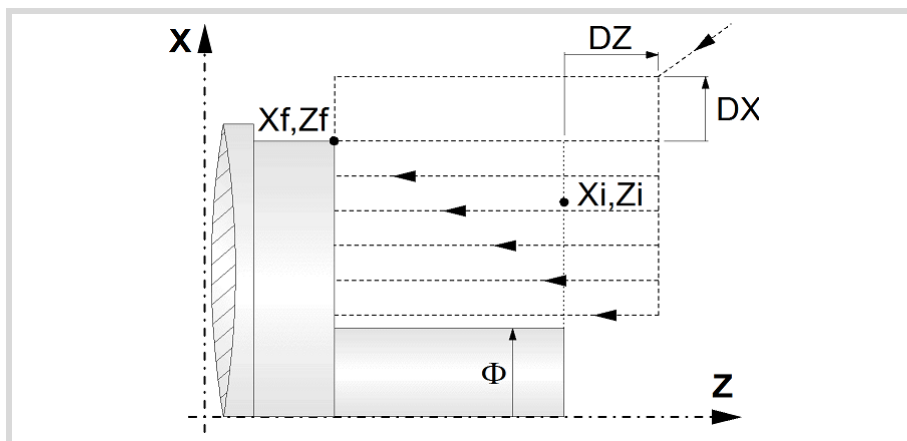


FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

- When the surface to be machined is not fully cylindrical, the CNC analyzes the coordinates on X of the starting and end points and takes as starting point on X the coordinate farthest from the final diameter.



- The roughing operation is carried out in G05, the rounding radius may be modified using the instruction #ROUNDPAR. If #ROUNDPAR is not programmed, it assumes the default rounding radius set by machine parameters.
- The finishing operation is carried out in G07.

## 4.

## CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Simple turning cycle.



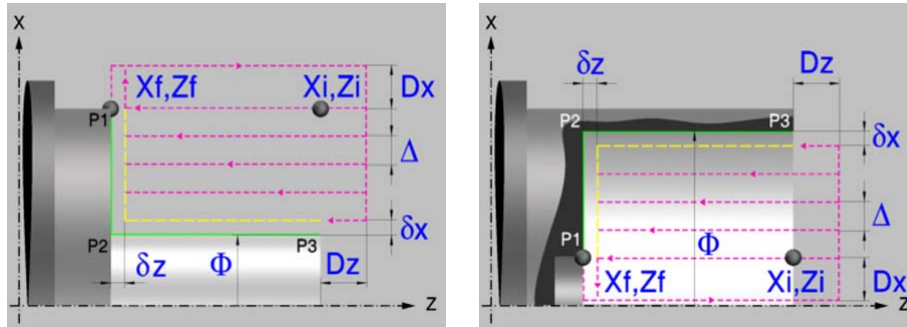
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 4.2 Turning cycle with vertex rounding.

The cycle allows outside and inside machining.



### Geometric parameters.

#### Inside or outside machining.



Outside machining.



Inside machining.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Coordinates of the profile's starting and final points.

$X_i, Z_i$  Coordinates of the starting point

$X_f, Z_f$  Coordinates of the end point. By default,  $X_f$  assumes the value set for  $X_i$ .

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

When the surface to be machined is not fully cylindrical, the CNC analyzes the coordinates on X of the starting and end points and takes as starting point on X the coordinate farthest from the final diameter.

#### Type of corner.

P1..P3 Type of corner; square (sharp), rounded or chamfered.

Each indicated point have an icon to define the type of corner; square (sharp), rounded or chamfered.



Square (sharp) corner.



Rounded corner. The rounding radius must be defined.



Chamfered corner. The size of the chamfer must be defined.

r Rounding radius of the pocket corners.

c Size of the chamfer of the pocket corners.

#### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the approach point coordinate with respect to the starting point ( $X_i, Z_i$ ).

$D_x, D_z$  Safety distance.

4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Turning cycle with vertex rounding.

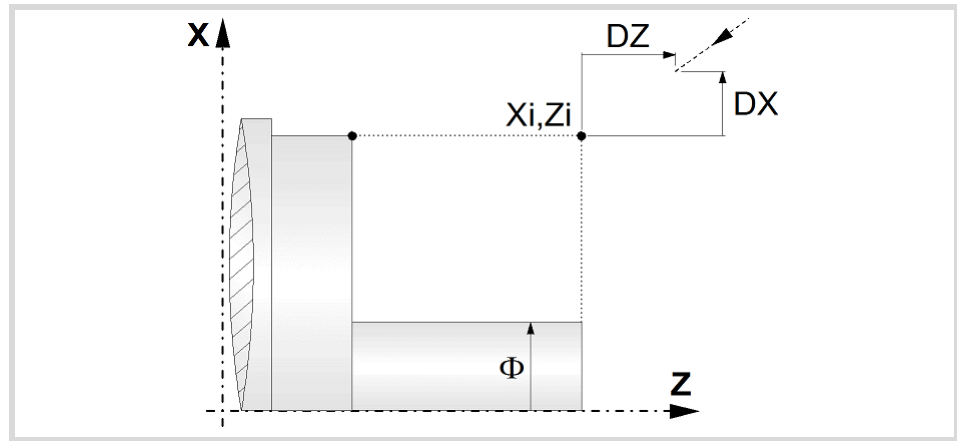
**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

The value of the safety distance on X is always defined in radius.



# 4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
Turning cycle with vertex rounding.

### Final diameter.

$\Phi$  Final diameter.

### Spindle parameters.

#### Spindle turning direction

The cycle lets select the spindle turning direction.



Spindle clockwise.



Spindle counterclockwise.

#### Coolant status.



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

#### Spindle work mode; rpm or Constant Surface Speed.



Constant turning speed.



Constant surface speed.

#### Maximum spindle speed.

$S_{max}$  Maximum spindle turning speed; if not programmed, the cycle will maintain the maximum active speed.

#### Spindle range (gear).

GEAR Spindle gear; if not programmed, the cycle assumes a value of 0. If defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.



## Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the roughing operation.



Do not run the roughing operation.

### Maximum machining pass.

$\Delta$  Maximum machining pass.

- If  $\Delta$  is positive, the cycle recalculates the step so all the passes are identical with the same value as or smaller than the one programmed.
- If  $\Delta$  is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

### Machining feedrate.

F Machining feedrate.

### Spindle speed.

S Spindle turning speed (rpm) or cutting speed.

### Tool data.

T Tool. If it is set to 0, the cycle does not execute the roughing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

### Machining M functions.



Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

## Machining parameters. Finishing.

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the finishing operation.



Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes ( $\delta x$ ,  $\delta z$ ).

### Finishing stocks.

$\delta x$ ,  $\delta z$  Finishing stock on each axis.

### Machining feedrate.

F Machining feedrate.

### Spindle speed.

S Spindle turning speed (rpm) or cutting speed.

### Tool data.

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Turning cycle with vertex rounding.

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

- D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

#### Machining M functions.



Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

# 4.

**CANNED CYCLES OF THE EDITOR. TURNING AND FACING.**

Turning cycle with vertex rounding.



FAGOR AUTOMATION

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

(REF: 1709)

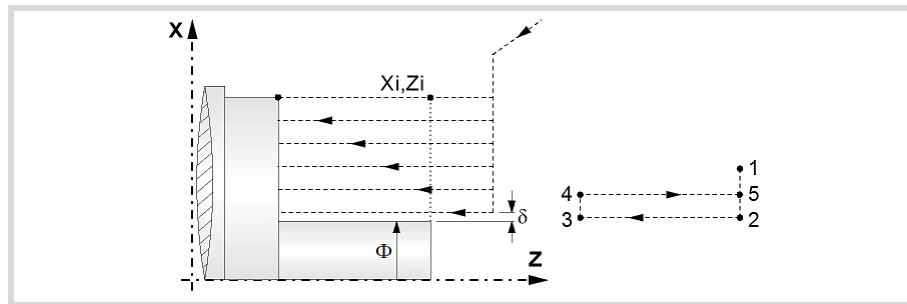
### 4.2.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The spindle starts with the speed selected and in the direction stated.
- 3 The tool approaches the starting point ( $X_i, Z_i$ ) in rapid, keeping the selected safety distance ( $D_x, D_z$ ) according to axes X and Z.
- 4 Roughing operation (if it has been programmed).

The cycle runs successive passes until reaching the final profile plus the finishing stocks. This operation is done with the conditions set for the roughing operation.

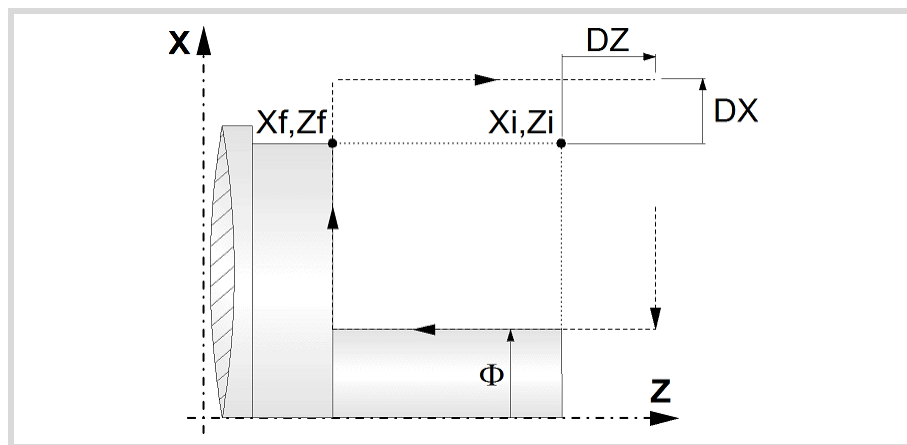
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



- 5 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle runs a single pass to remove the finishing stock. This operation is done with the conditions set for the finishing operation.



- 6 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance).
- 7 The CNC will not stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

### Considerations.

- If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.
- If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the finishing operation, the tool will move to the safety point (starting point plus the safety distance).

# 4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Turning cycle with vertex rounding.

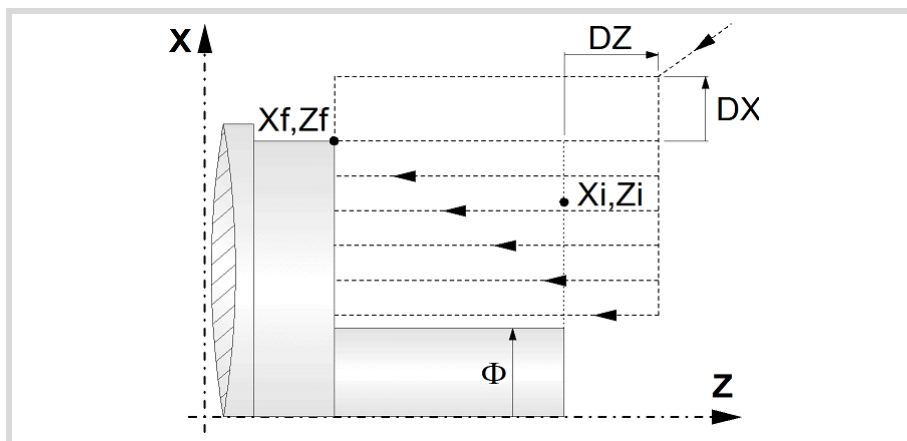


FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

- When the surface to be machined is not fully cylindrical, the CNC analyzes the coordinates on X of the starting and end points and takes as starting point on X the coordinate farthest from the final diameter.



- The roughing operation is carried out in G05, the rounding radius may be modified using the instruction #ROUNDPAR. If #ROUNDPAR is not programmed, it assumes the default rounding radius set by machine parameters.
- The finishing operation is carried out in G07.

## 4.

## CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Turning cycle with vertex rounding.

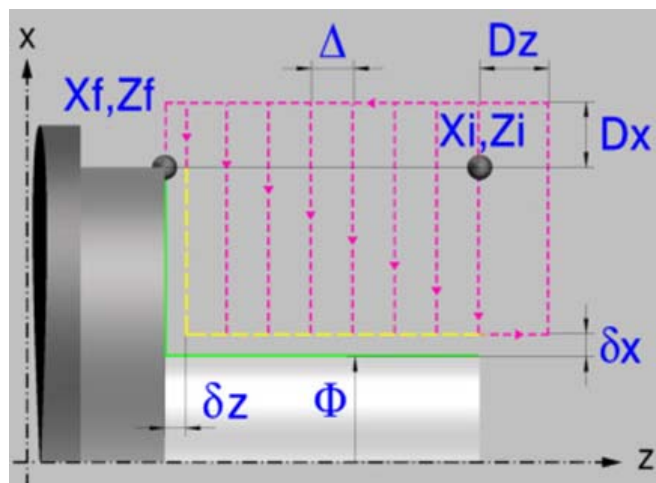


FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

### 4.3 Simple facing cycle.



# 4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
Simple facing cycle.

#### Geometric parameters.

##### Coordinates of the profile's starting and final points.

$X_i, Z_i$  Coordinates of the starting point

$X_f, Z_f$  Coordinates of the end point. By default,  $X_f$  assumes the value set for  $X_i$ .

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

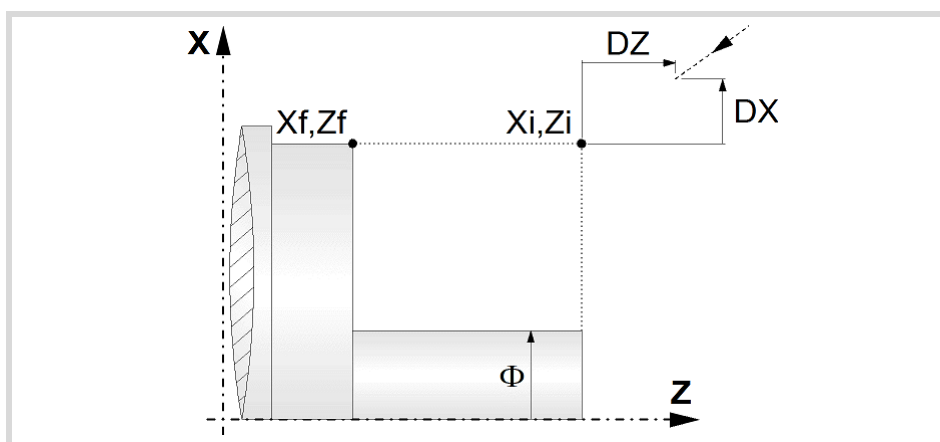
When the surface to be machined is not fully cylindrical, the CNC analyzes the coordinates on X of the starting and end points and takes as starting point on X the coordinate farthest from the final diameter.

##### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the approach point coordinate with respect to the starting point ( $X_i, Z_i$ ).

$D_x, D_z$  Safety distance.

The value of the safety distance on X is always defined in radius.



##### Final diameter.

$\Phi$  Final diameter.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## Spindle parameters.

### Spindle turning direction

The cycle lets select the spindle turning direction.



Spindle clockwise.



Spindle counterclockwise.

### Coolant status.



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

### Spindle work mode; rpm or Constant Surface Speed.



Constant turning speed.



Constant surface speed.

### Maximum spindle speed.

**Smax** Maximum spindle turning speed; if not programmed, the cycle will maintain the maximum active speed.

### Spindle range (gear).

**GEAR** Spindle gear; if not programmed, the cycle assumes a value of 0. If defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

## Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the roughing operation.



Do not run the roughing operation.

### Maximum machining pass.

**Δ** Maximum machining pass.

- If  $\Delta$  is positive, the cycle recalculates the step so all the passes are identical with the same value as or smaller than the one programmed.
- If  $\Delta$  is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

### Machining feedrate.

**F** Machining feedrate.

### Spindle speed.

**S** Spindle turning speed (rpm) or cutting speed.

### Tool data.

**T** Tool. If it is set to 0, the cycle does not execute the roughing operation.

# 4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
Simple facing cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

- D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**



Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**Machining parameters. Finishing.**

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the finishing operation.



Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes ( $\delta x$ ,  $\delta z$ ).

**Finishing stocks.**

$\delta x$ ,  $\delta z$  Finishing stock on each axis.

**Machining feedrate.**

F Machining feedrate.

**Spindle speed.**

S Spindle turning speed (rpm) or cutting speed.

**Tool data.**

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**



Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Simple facing cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

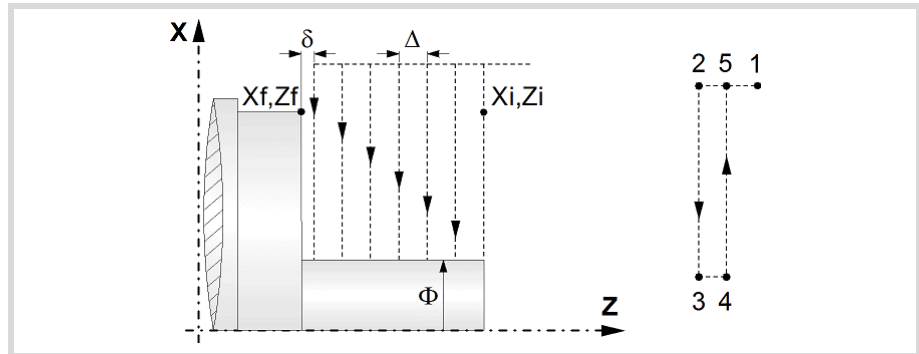
### 4.3.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The spindle starts with the speed selected and in the direction stated.
- 3 The tool approaches the starting point ( $X_i, Z_i$ ) in rapid, keeping the selected safety distance ( $D_x, D_z$ ) according to axes X and Z.
- 4 Roughing operation (if it has been programmed).

The cycle runs successive passes until reaching the final profile plus the finishing stocks. This operation is done with the conditions set for the roughing operation.

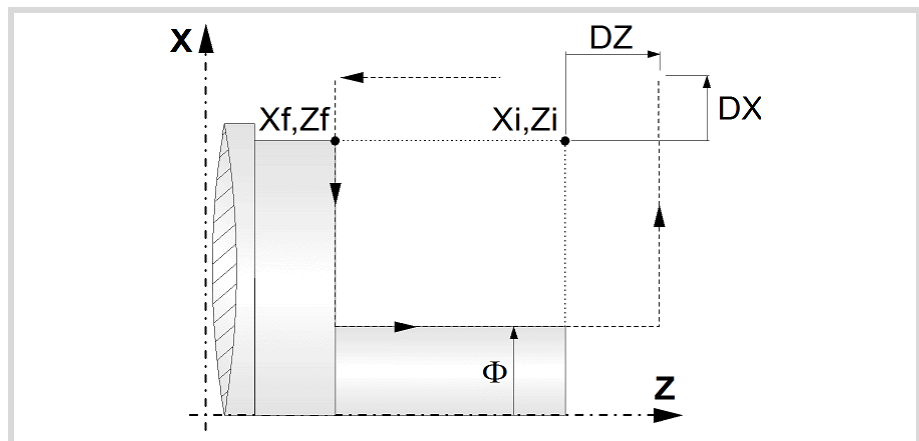
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



- 5 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle runs a single pass to remove the finishing stock. This operation is done with the conditions set for the finishing operation.



- 6 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance).
- 7 The CNC will not stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

### Considerations.

- If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.
- If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the finishing operation, the tool will move to the safety point (starting point plus the safety distance).

# 4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
Simple facing cycle.



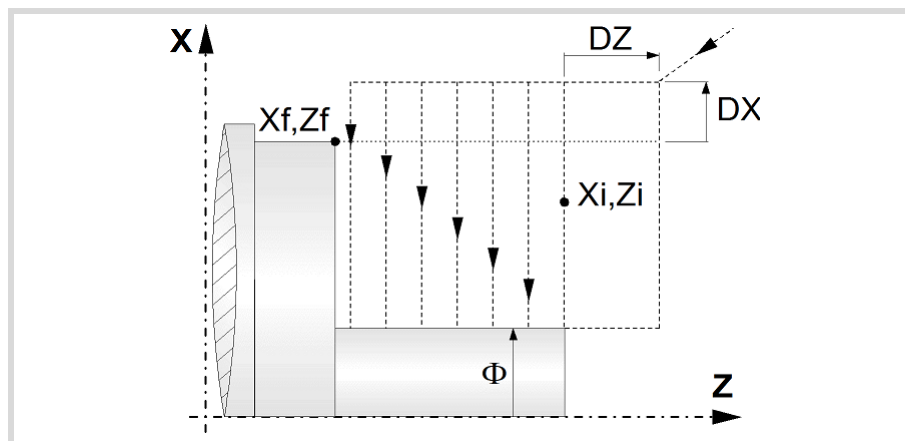
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)



- When the surface to be machined is not fully cylindrical, the CNC analyzes the coordinates on X of the starting and end points and takes as starting point on X the coordinate farthest from the final diameter.



- The roughing operation is carried out in G05, the rounding radius may be modified using the instruction #ROUNDPAR. If #ROUNDPAR is not programmed, it assumes the default rounding radius set by machine parameters.
- The finishing operation is carried out in G07.

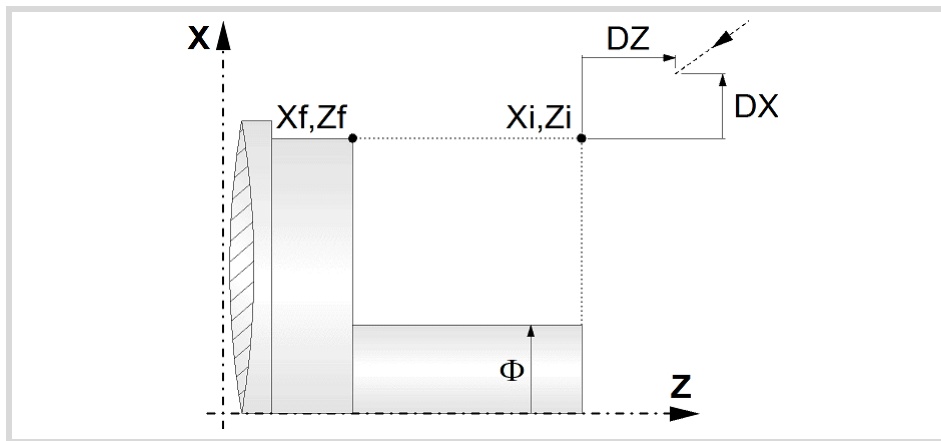
## 4.

### CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Simple facing cycle.



The value of the safety distance on X is always defined in radius.



4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
Facing cycle with vertex rounding.

**Final diameter.**

Φ Final diameter.

**Spindle parameters.**

**Spindle turning direction**

The cycle lets select the spindle turning direction.



Spindle clockwise.



Spindle counterclockwise.

**Coolant status.**



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

**Spindle work mode; rpm or Constant Surface Speed.**



Constant turning speed.



Constant surface speed.

**Maximum spindle speed.**

Smax Maximum spindle turning speed; if not programmed, the cycle will maintain the maximum active speed.

**Spindle range (gear).**

GEAR Spindle gear; if not programmed, the cycle assumes a value of 0. If defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.





FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.

-  Run the roughing operation.
-  Do not run the roughing operation.

### Maximum machining pass.

- $\Delta$  Maximum machining pass.
- If  $\Delta$  is positive, the cycle recalculates the step so all the passes are identical with the same value as or smaller than the one programmed.
  - If  $\Delta$  is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

### Machining feedrate.

- F Machining feedrate.



### Spindle speed.

- S Spindle turning speed (rpm) or cutting speed.

### Tool data.

- T Tool. If it is set to 0, the cycle does not execute the roughing operation.
- D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.



### Machining M functions.

-  Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.
-  Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

## Machining parameters. Finishing.

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.

-  Run the finishing operation.
-  Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes ( $\delta x$ ,  $\delta z$ ).

### Finishing stocks.

- $\delta x$ ,  $\delta z$  Finishing stock on each axis.

### Machining feedrate.

- F Machining feedrate.

### Spindle speed.

- S Spindle turning speed (rpm) or cutting speed.

### Tool data.

- T Tool. If it is set to 0, the cycle does not execute the finishing operation.

# 4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
Facing cycle with vertex rounding.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

- D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**



Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

4.

**CANNED CYCLES OF THE EDITOR. TURNING AND FACING.**

Facing cycle with vertex rounding.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

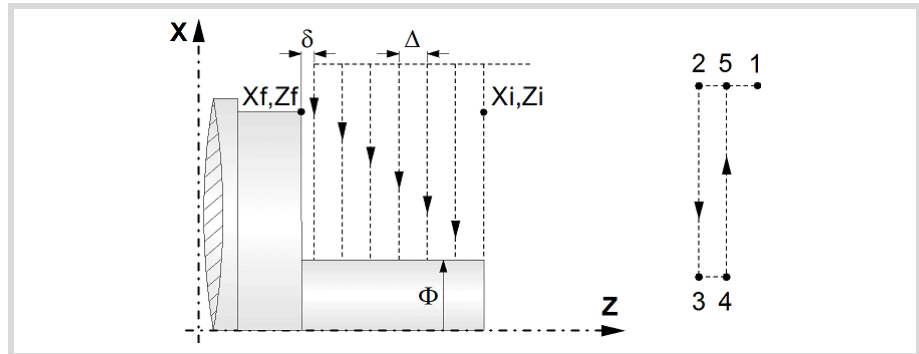
#### 4.4.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The spindle starts with the speed selected and in the direction stated.
- 3 The tool approaches the starting point ( $X_i, Z_i$ ) in rapid, keeping the selected safety distance ( $D_x, D_z$ ) according to axes X and Z.
- 4 Roughing operation (if it has been programmed).

The cycle runs successive passes until reaching the final profile plus the finishing stocks. This operation is done with the conditions set for the roughing operation.

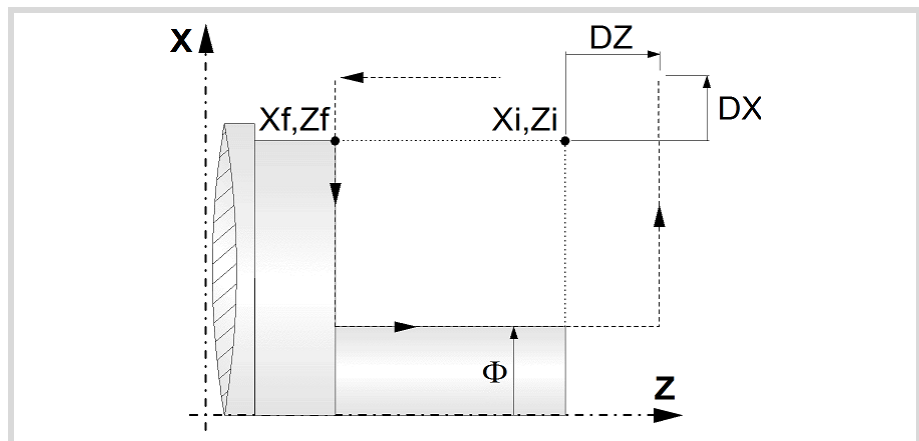
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



- 5 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle runs a single pass to remove the finishing stock. This operation is done with the conditions set for the finishing operation.



- 6 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance).
- 7 The CNC will not stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

#### Considerations.

- If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.
- If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the finishing operation, the tool will move to the safety point (starting point plus the safety distance).

# 4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
Facing cycle with vertex rounding.

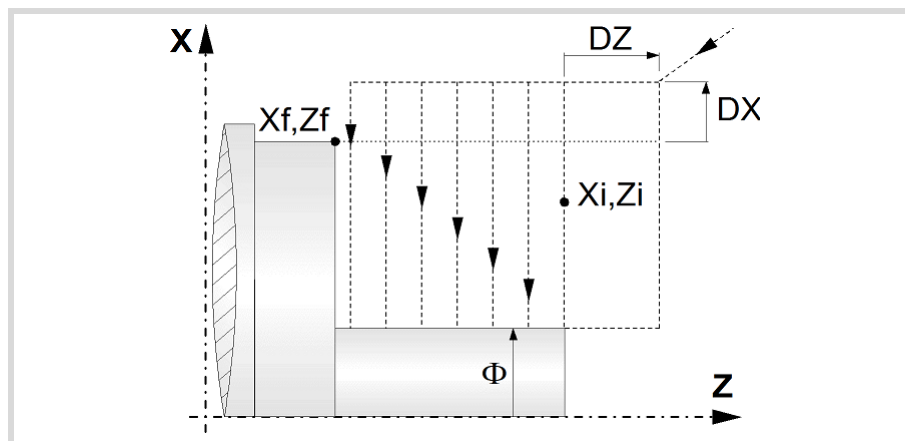


FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

- When the surface to be machined is not fully cylindrical, the CNC analyzes the coordinates on X of the starting and end points and takes as starting point on X the coordinate farthest from the final diameter.



- The roughing operation is carried out in G05, the rounding radius may be modified using the instruction #ROUNDPAR. If #ROUNDPAR is not programmed, it assumes the default rounding radius set by machine parameters.
- The finishing operation is carried out in G07.

# 4.

## CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Facing cycle with vertex rounding.



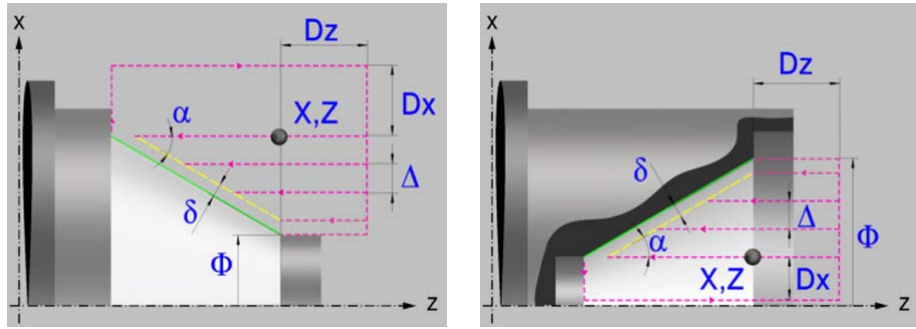
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 4.5 Vertex chamfering cycle.

The cycle allows outside and inside machining.



### Geometric parameters.

#### Inside or outside machining.



Outside machining.



Inside machining.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Shape of the part before the tapered segment.



Initial concave corner.



Initial convex corner.

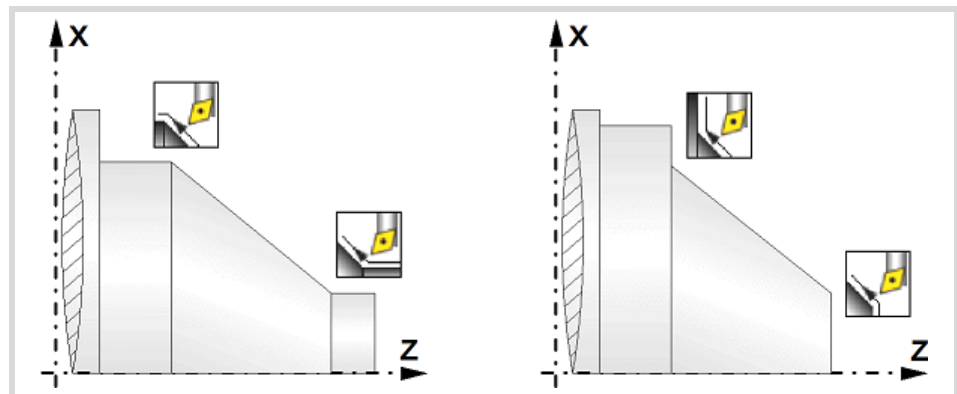
#### Shape of the part after the tapered segment.



Final concave corner.



Final convex corner.



4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
Vertex chamfering cycle.

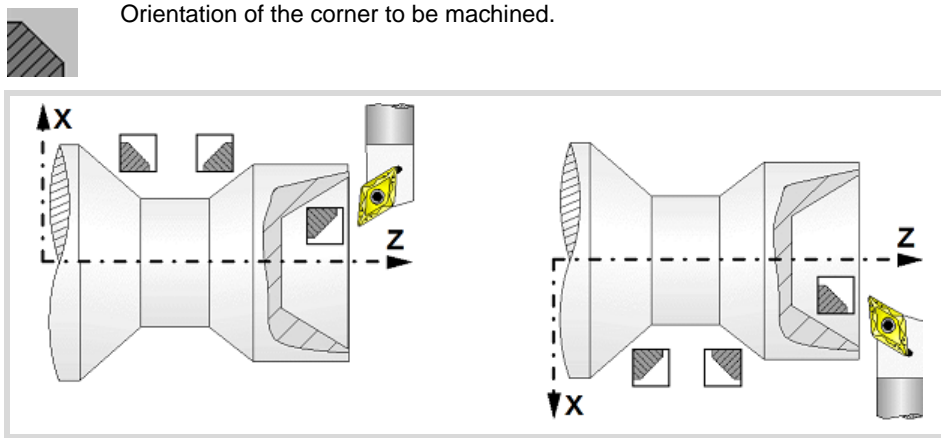
**FAGOR**   
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)



**Work quadrant.**



**Coordinates of the starting point**

X, Z Coordinates of the theoretical corner of the tapered segment.

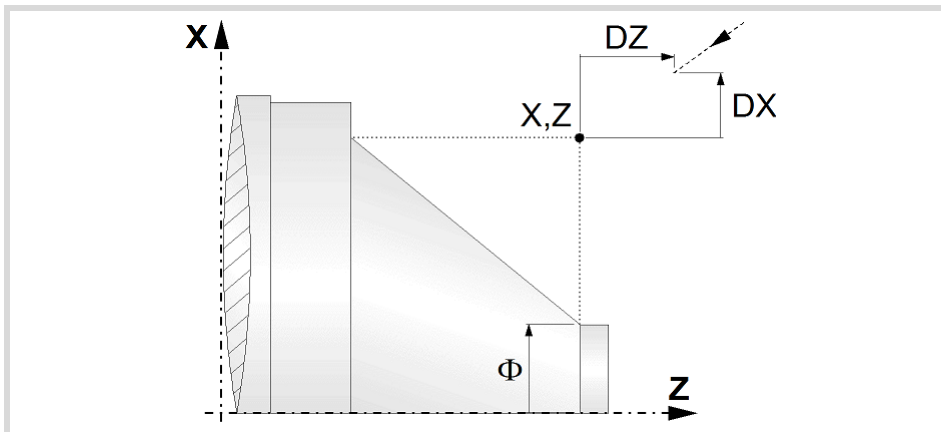
To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

**Safety distance.**

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the approach point coordinate with respect to the theoretical corner (X, Z).

Dx, Dz Safety distance.

The value of the safety distance on X is always defined in radius.



**Final diameter.**

$\Phi$  Final diameter.

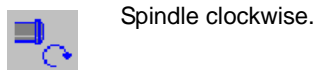
**Taper angle.**

$\alpha$  Taper angle with respect to the abscissa axis.

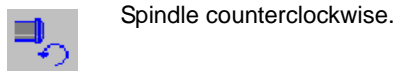
**Spindle parameters.**

**Spindle turning direction**

The cycle lets select the spindle turning direction.



Spindle clockwise.



Spindle counterclockwise.

4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING. Vertex chamfering cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Coolant status.**



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

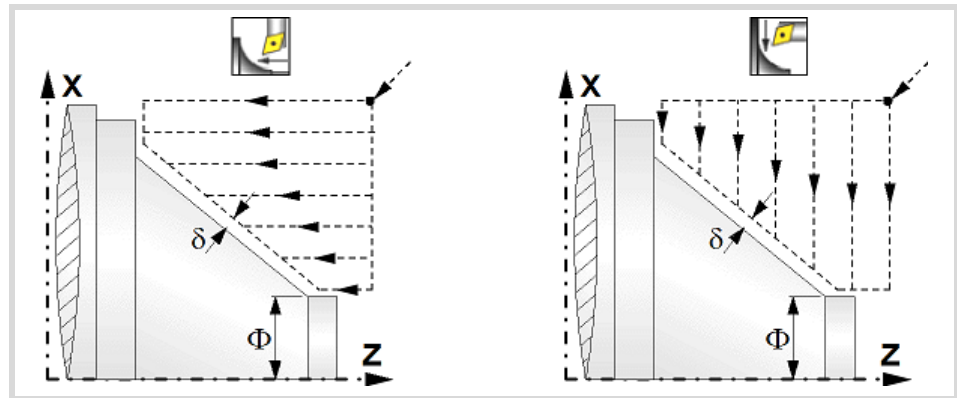
**Machining direction.**



Machining in the turning direction.



Machining in the facing direction.



**Spindle work mode; rpm or Constant Surface Speed.**



Constant turning speed.



Constant surface speed.

**Maximum spindle speed.**

Smax Maximum spindle turning speed; if not programmed, the cycle will maintain the maximum active speed.

**Spindle range (gear).**

GEAR Spindle gear; if not programmed, the cycle assumes a value of 0. If defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Machining parameters. Roughing.**

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the roughing operation.



Do not run the roughing operation.

**Maximum machining pass.**

Δ Maximum machining pass.

- If Δ is positive, the cycle recalculates the step so all the passes are identical with the same value as or smaller than the one programmed.
- If Δ is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING. Vertex chamfering cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Machining feedrate.**

F Machining feedrate.

**Spindle speed.**

S Spindle turning speed (rpm) or cutting speed.

**Tool data.**

T Tool. If it is set to 0, the cycle does not execute the roughing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**



Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**Machining parameters. Finishing.**

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the finishing operation.



Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes ( $\delta x$ ,  $\delta z$ ).

**Finishing stocks.**

The cycle lets define a single finishing stock that will be applied depending on the cutter edge or 2 finishing stocks, one per axis (X, Z).



Define the finishing stock depending on the cutter edge. The finishing stock is measured on the edge of the tool.



Define 2 finishing stocks, one per axis, regardless of the type of tool being used.

$\delta$  Finishing stock depending on the cutter edge.

$\delta x$ ,  $\delta z$  Finishing stock on each axis.

**Machining feedrate.**

F Machining feedrate.

**Spindle speed.**

S Spindle turning speed (rpm) or cutting speed.

**Tool data.**

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
Vertex chamfering cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Machining M functions.**

Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**4.****CANNED CYCLES OF THE EDITOR. TURNING AND FACING.**

Vertex chamfering cycle.



FAGOR AUTOMATION

**CNC 8060  
CNC 8065**

(REF: 1709)

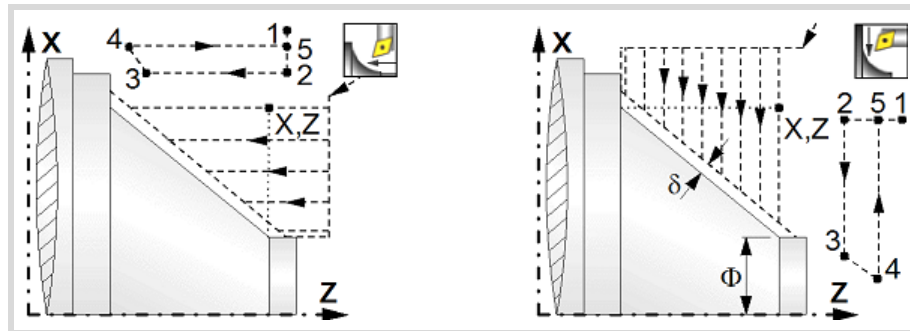
### 4.5.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The spindle starts with the speed selected and in the direction stated.
- 3 The tool approaches the theoretical corner (X, Z) in rapid, keeping the safety distance (Dx, Dz) selected according to axes X and Z.
- 4 Roughing operation (if it has been programmed).

The cycle runs successive turning or facing passes (according to the selected option) until reaching the final profile plus the finishing stocks. This operation is done with the conditions set for the roughing operation.

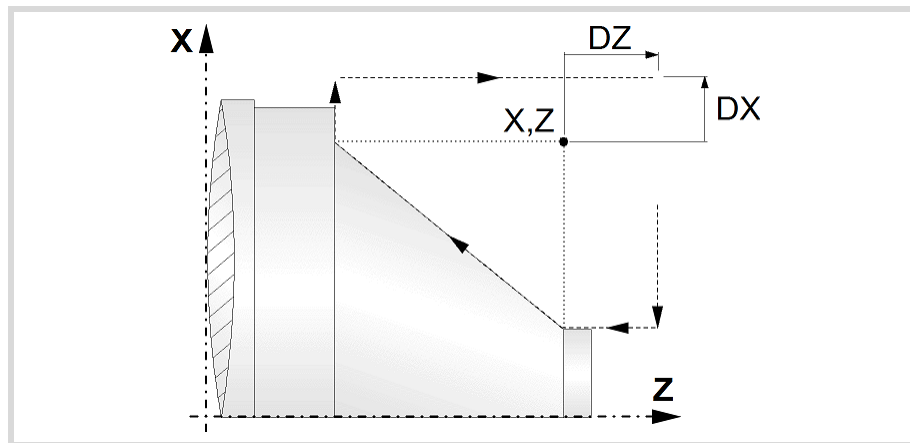
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



- 5 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle runs a single pass to remove the finishing stock. This operation is done with the conditions set for the finishing operation.



- 6 Once the operation or cycle has ended, the tool will return to the safety point (theoretical corner plus the safety distance).
- 7 The CNC will not stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

### Considerations.

- If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.
- If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the finishing operation, the tool will move to the safety point (starting point plus the safety distance).

# 4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING. Vertex chamfering cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

- The roughing operation is carried out in G05, the rounding radius may be modified using the instruction #ROUNDPAR. If #ROUNDPAR is not programmed, it assumes the default rounding radius set by machine parameters.
- The finishing operation is carried out in G07.

# 4.

## CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Vertex chamfering cycle.



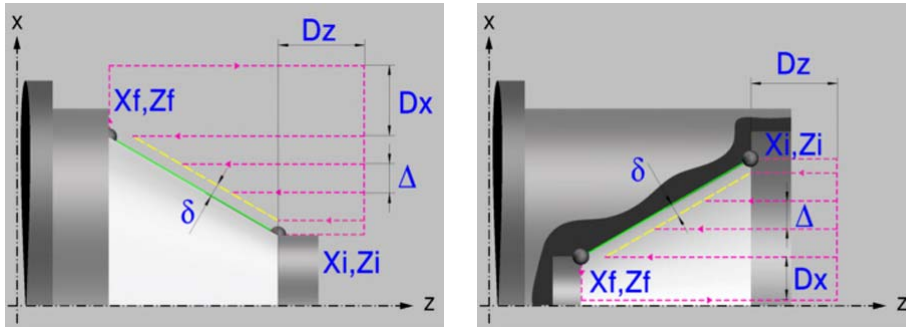
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 4.6 "Chamfering between points" cycle.

The cycle allows outside and inside machining.



### Geometric parameters.

#### Inside or outside machining.



Outside machining.



Inside machining.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Shape of the part before the tapered segment.



Initial concave corner.



Initial convex corner.

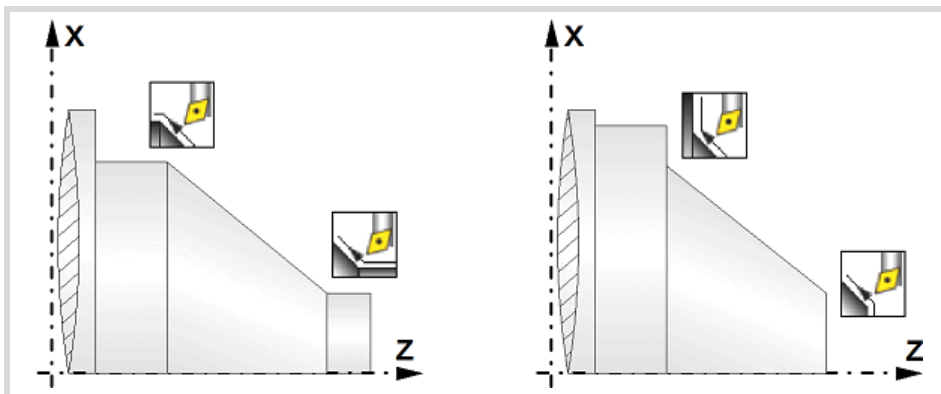
#### Shape of the part after the tapered segment.



Final concave corner.



Final convex corner.



4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
"Chamfering between points" cycle.

**FAGOR** 

FAGOR AUTOMATION

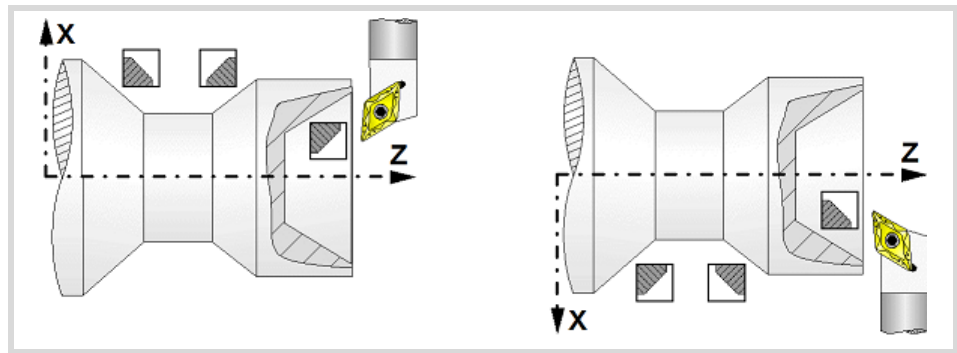
CNC 8060  
CNC 8065

(REF: 1709)

**Work quadrant.**



Orientation of the corner to be machined.



**Coordinates of the profile's starting and final points.**

$X_i, Z_i$  Coordinates of the starting point

$X_f, Z_f$  Coordinates of the end point.

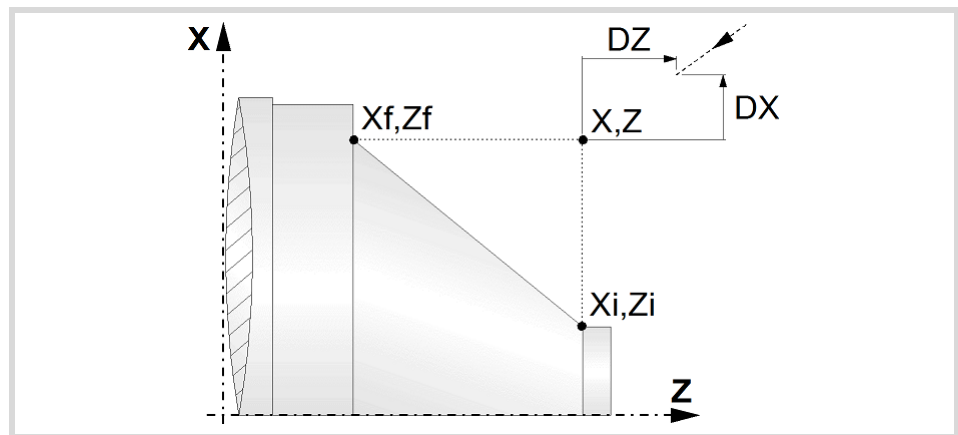
To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

**Safety distance.**

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the approach point coordinate with respect to the theoretical corner.

$D_x, D_z$  Safety distance.

The value of the safety distance on X is always defined in radius.



**Spindle parameters.**

**Spindle turning direction**

The cycle lets select the spindle turning direction.



Spindle clockwise.



Spindle counterclockwise.

4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
"Chamfering between points" cycle.



CNC 8060  
CNC 8065

(REF: 1709)



**Coolant status.**



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

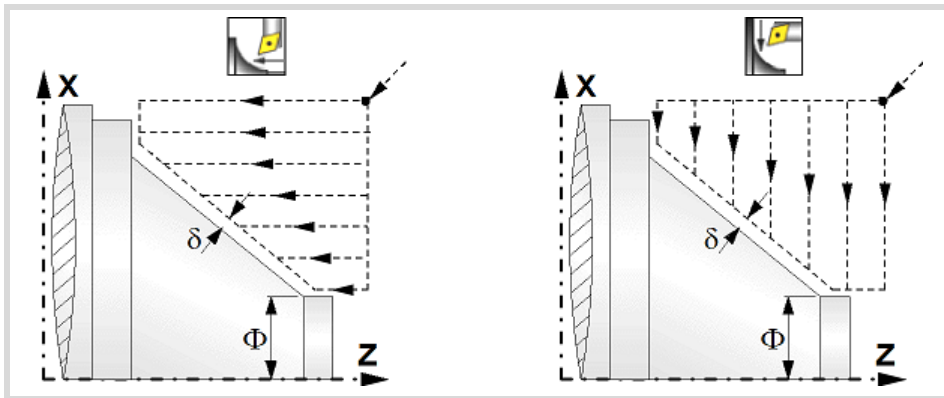
**Machining direction.**



Machining in the turning direction.



Machining in the facing direction.



**Spindle work mode; rpm or Constant Surface Speed.**



Constant turning speed.



Constant surface speed.

**Maximum spindle speed.**

Smax Maximum spindle turning speed; if not programmed, the cycle will maintain the maximum active speed.

**Spindle range (gear).**

GEAR Spindle gear; if not programmed, the cycle assumes a value of 0. If defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Machining parameters. Roughing.**

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the roughing operation.



Do not run the roughing operation.

**Maximum machining pass.**

$\Delta$  Maximum machining pass.

- If  $\Delta$  is positive, the cycle recalculates the step so all the passes are identical with the same value as or smaller than the one programmed.
- If  $\Delta$  is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

## 4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
"Chamfering between points" cycle.

**Machining feedrate.**

F Machining feedrate.

**Spindle speed.**

S Spindle turning speed (rpm) or cutting speed.

**Tool data.**

T Tool. If it is set to 0, the cycle does not execute the roughing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**Machining parameters. Finishing.**

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the finishing operation.



Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes ( $\delta x$ ,  $\delta z$ ).

**Finishing stocks.**

The cycle lets define a single finishing stock that will be applied depending on the cutter edge or 2 finishing stocks, one per axis (X, Z).



Define the finishing stock depending on the cutter edge. The finishing stock is measured on the edge of the tool.



Define 2 finishing stocks, one per axis, regardless of the type of tool being used.

$\delta$  Finishing stock depending on the cutter edge.

$\delta x$ ,  $\delta z$  Finishing stock on each axis.

**Machining feedrate.**

F Machining feedrate.

**Spindle speed.**

S Spindle turning speed (rpm) or cutting speed.

**Tool data.**

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**



Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

4.

**CANNED CYCLES OF THE EDITOR. TURNING AND FACING.**  
"Chamfering between points" cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

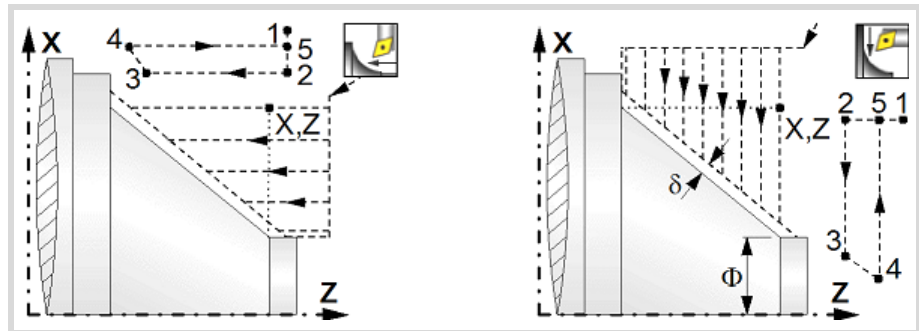
### 4.6.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The spindle starts with the speed selected and in the direction stated.
- 3 The tool approaches the theoretical corner (X, Z) in rapid, keeping the safety distance (Dx, Dz) selected according to axes X and Z.
- 4 Roughing operation (if it has been programmed).

The cycle runs successive turning or facing passes (according to the selected option) until reaching the final profile plus the finishing stocks. This operation is done with the conditions set for the roughing operation.

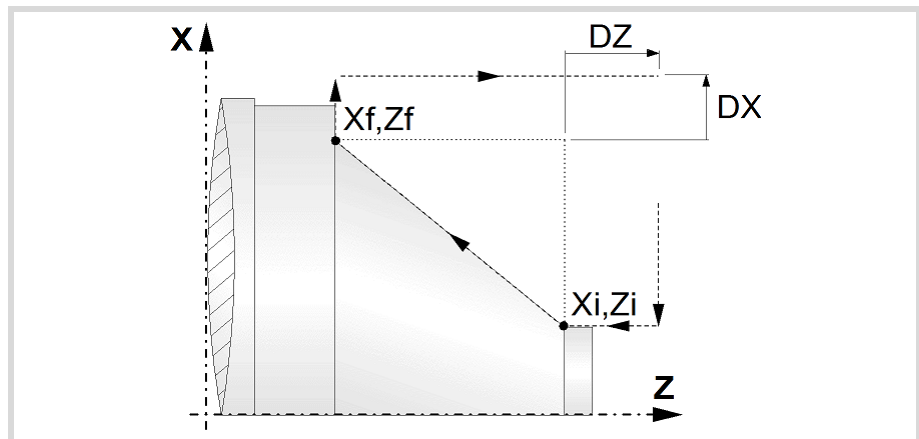
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



- 5 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle runs a single pass to remove the finishing stock. This operation is done with the conditions set for the finishing operation.



- 6 Once the operation or cycle has ended, the tool will return to the safety point (theoretical corner plus the safety distance).
- 7 The CNC will not stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

### Considerations.

- If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.
- If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the finishing operation, the tool will move to the safety point (starting point plus the safety distance).

4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
"Chamfering between points" cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## Operating manual (TC).

- The roughing operation is carried out in G05, the rounding radius may be modified using the instruction #ROUNDPAR. If #ROUNDPAR is not programmed, it assumes the default rounding radius set by machine parameters.
- The finishing operation is carried out in G07.

# 4.

**CANNED CYCLES OF THE EDITOR. TURNING AND FACING.**  
"Chamfering between points" cycle.

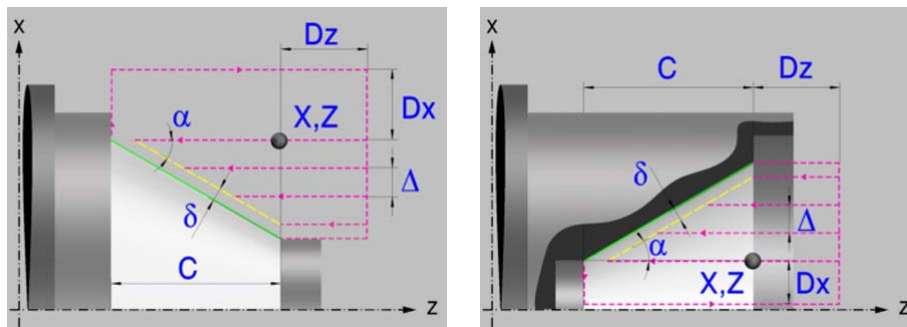
**FAGOR**   
FAGOR AUTOMATION

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

(REF: 1709)

## 4.7 Vertex chamfering cycle 2.

The cycle allows outside and inside machining.



### Geometric parameters.

#### Inside or outside machining.



Outside machining.



Inside machining.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Shape of the part before the tapered segment.



Initial concave corner.



Initial convex corner.

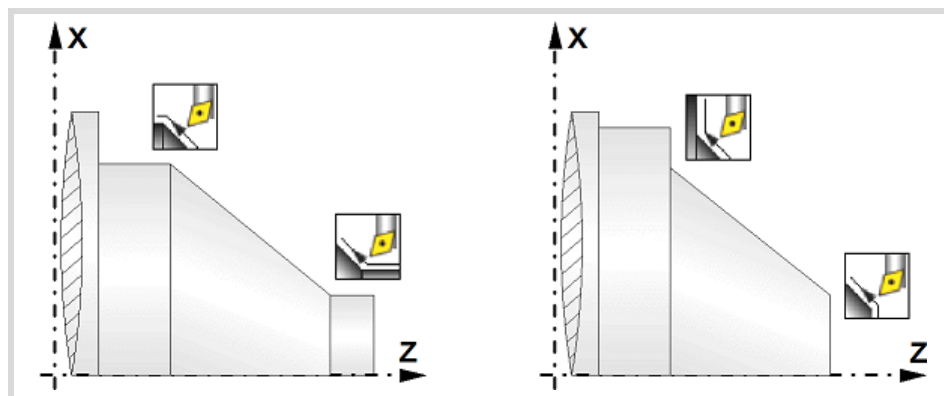
#### Shape of the part after the tapered segment.



Final concave corner.



Final convex corner.



4.


CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
Vertex chamfering cycle 2.

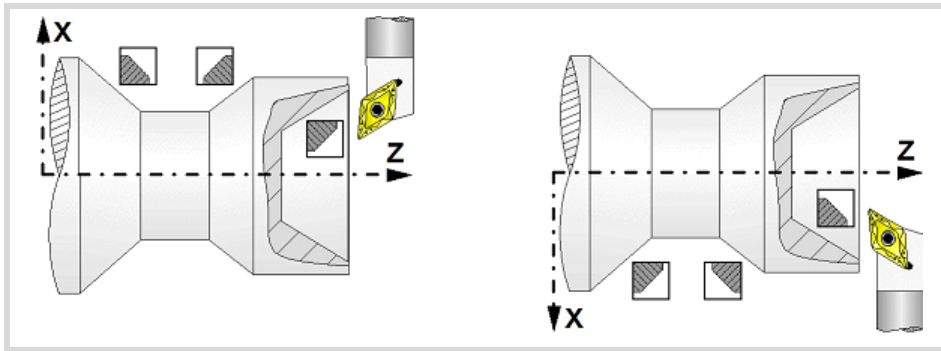
**FAGOR**   
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Work quadrant.**

 Orientation of the corner to be machined.



**Coordinates of the starting point**

X, Z Coordinates of the theoretical corner of the tapered segment.

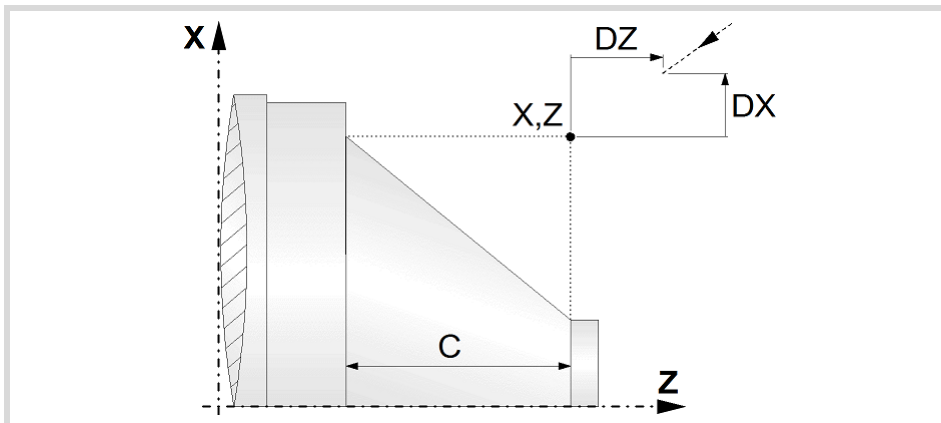
To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

**Safety distance.**

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the approach point coordinate with respect to the theoretical corner.

Dx, Dz Safety distance.

The value of the safety distance on X is always defined in radius.



**Taper length.**

C Taper length.

**Taper angle.**

$\alpha$  Taper angle with respect to the abscissa axis.

**Spindle parameters.**

**Spindle turning direction**

The cycle lets select the spindle turning direction.



Spindle clockwise.



Spindle counterclockwise.

**4.**

**CANNED CYCLES OF THE EDITOR. TURNING AND FACING.**  
Vertex chamfering cycle 2.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Coolant status.**



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

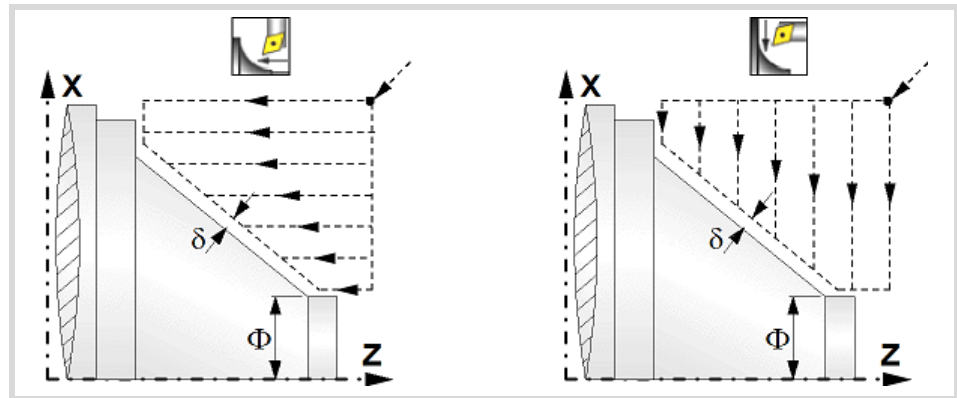
**Machining direction.**



Machining in the turning direction.



Machining in the facing direction.



**Spindle work mode; rpm or Constant Surface Speed.**



Constant turning speed.



Constant surface speed.

**Maximum spindle speed.**

Smax Maximum spindle turning speed; if not programmed, the cycle will maintain the maximum active speed.

**Spindle range (gear).**

GEAR Spindle gear; if not programmed, the cycle assumes a value of 0. If defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Machining parameters. Roughing.**

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the roughing operation.



Do not run the roughing operation.

**Maximum machining pass.**

Δ Maximum machining pass.

- If Δ is positive, the cycle recalculates the step so all the passes are identical with the same value as or smaller than the one programmed.
- If Δ is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING. Vertex chamfering cycle 2.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)



**Machining feedrate.**

F Machining feedrate.

**Spindle speed.**

S Spindle turning speed (rpm) or cutting speed.

**Tool data.**

T Tool. If it is set to 0, the cycle does not execute the roughing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**



Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**Machining parameters. Finishing.**

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the finishing operation.



Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes ( $\delta x$ ,  $\delta z$ ).

**Finishing stocks.**

The cycle lets define a single finishing stock that will be applied depending on the cutter edge or 2 finishing stocks, one per axis (X, Z).



Define the finishing stock depending on the cutter edge. The finishing stock is measured on the edge of the tool.



Define 2 finishing stocks, one per axis, regardless of the type of tool being used.

$\delta$  Finishing stock depending on the cutter edge.

$\delta x$ ,  $\delta z$  Finishing stock on each axis.

**Machining feedrate.**

F Machining feedrate.

**Spindle speed.**

S Spindle turning speed (rpm) or cutting speed.

**Tool data.**

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**4.****CANNED CYCLES OF THE EDITOR. TURNING AND FACING.**

Vertex chamfering cycle 2.



FAGOR AUTOMATION

**CNC 8060  
CNC 8065**

(REF: 1709)

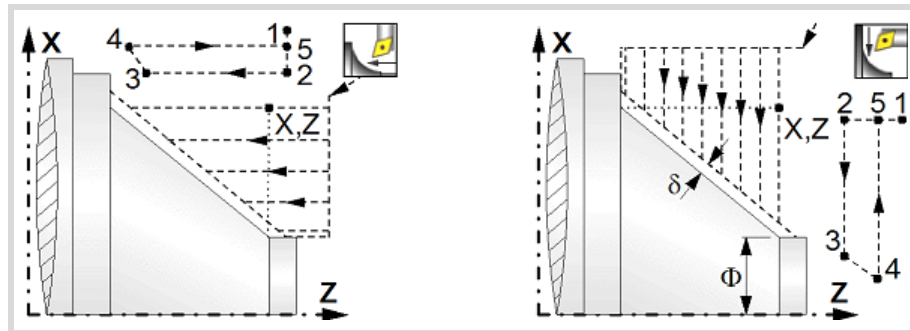
### 4.7.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The spindle starts with the speed selected and in the direction stated.
- 3 The tool approaches the theoretical corner (X, Z) in rapid, keeping the safety distance (Dx, Dz) selected according to axes X and Z.
- 4 Roughing operation (if it has been programmed).

The cycle runs successive turning or facing passes (according to the selected option) until reaching the final profile plus the finishing stocks. This operation is done with the conditions set for the roughing operation.

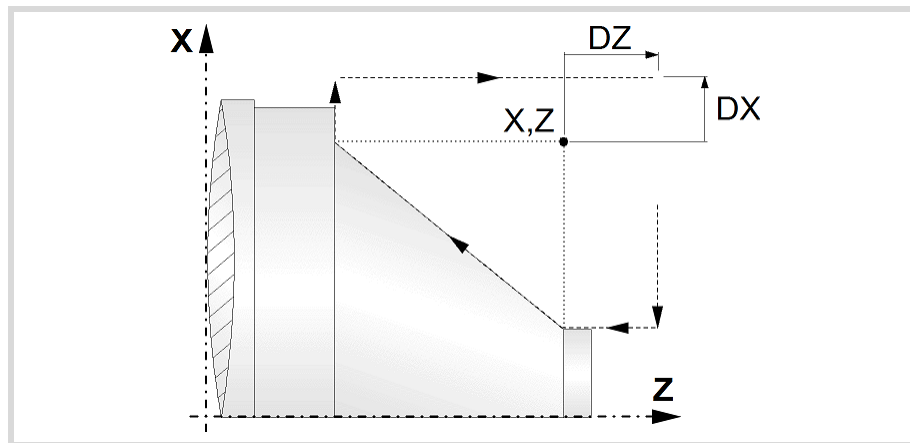
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



- 5 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle runs a single pass to remove the finishing stock. This operation is done with the conditions set for the finishing operation.



- 6 Once the operation or cycle has ended, the tool will return to the safety point (theoretical corner plus the safety distance).
- 7 The CNC will not stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

### Considerations.

- If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.
- If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the finishing operation, the tool will move to the safety point (starting point plus the safety distance).

# 4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
Vertex chamfering cycle 2.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

- The roughing operation is carried out in G05, the rounding radius may be modified using the instruction #ROUNDPAR. If #ROUNDPAR is not programmed, it assumes the default rounding radius set by machine parameters.
- The finishing operation is carried out in G07.

# 4.

## CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Vertex chamfering cycle 2.



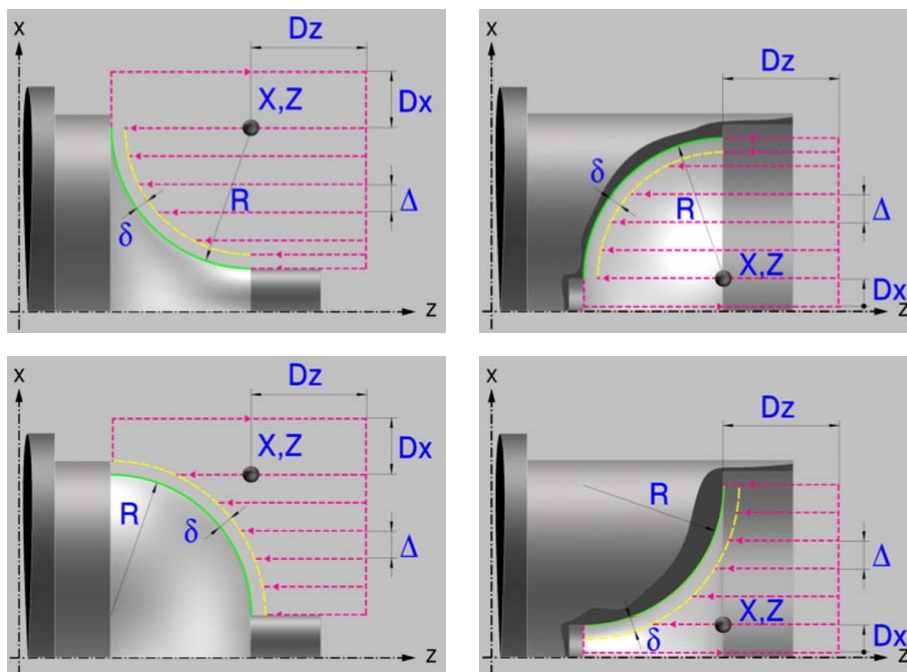
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 4.8 Vertex rounding cycle.

The cycle allows outside and inside, concave and convex machining.



4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
Vertex rounding cycle.

### Geometric parameters.

#### Inside or outside machining.



Outside machining.



Inside machining.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Concave and convex rounding.



Concave rounding.



Convex rounding.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Shape of the part before the tapered segment.



Initial concave corner.



Initial convex corner.

#### Shape of the part after the tapered segment.



Final concave corner.



Final convex corner.

**FAGOR** 

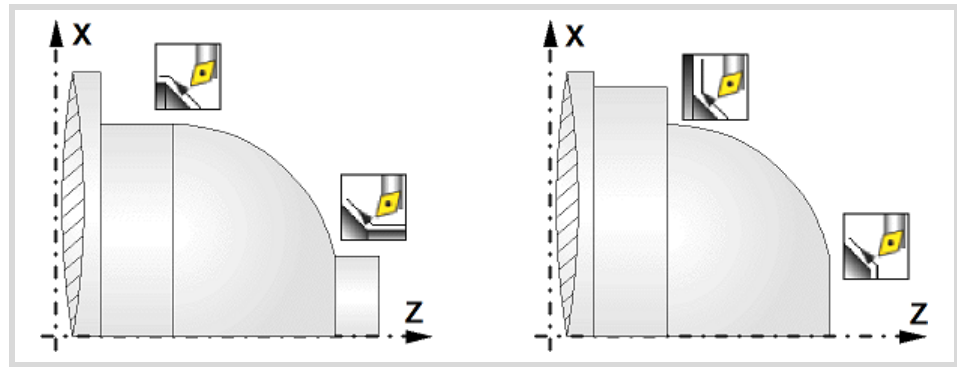
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

# 4.

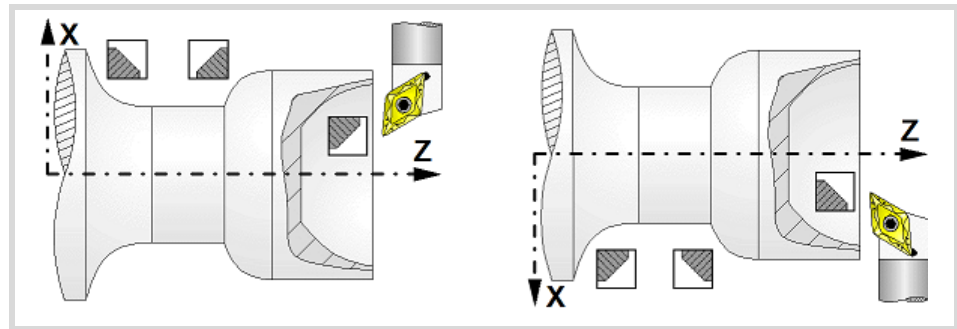
CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
Vertex rounding cycle.



### Work quadrant.



Orientation of the corner to be machined.



### Coordinates of the starting point

X, Z Coordinates of the theoretical corner of the rounding.

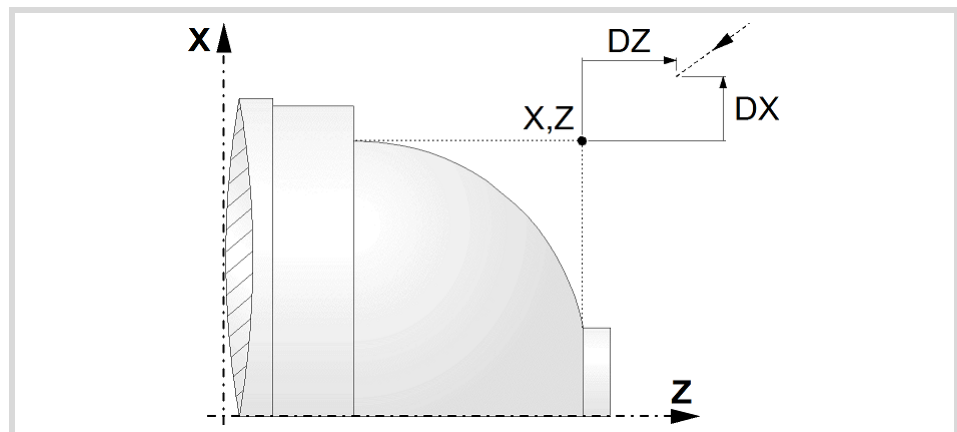
To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the approach point coordinate with respect to the theoretical corner.

Dx, Dz Safety distance.

The value of the safety distance on X is always defined in radius.



### Taper length.

R Rounding radius.

## Spindle parameters.

### Spindle turning direction

The cycle lets select the spindle turning direction.



Spindle clockwise.



Spindle counterclockwise.

### Coolant status.



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

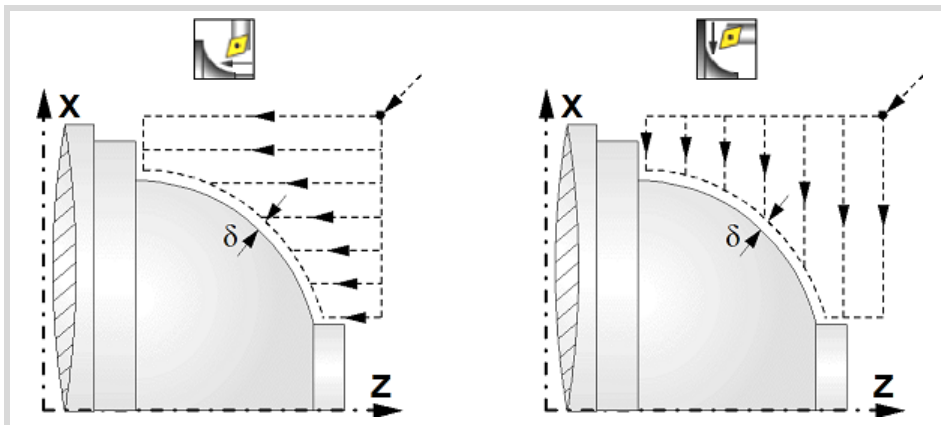
### Machining direction.



Machining in the turning direction.



Machining in the facing direction.



### Spindle work mode; rpm or Constant Surface Speed.



Constant turning speed.



Constant surface speed.

### Maximum spindle speed.

**Smax** Maximum spindle turning speed; if not programmed, the cycle will maintain the maximum active speed.

### Spindle range (gear).

**GEAR** Spindle gear; if not programmed, the cycle assumes a value of 0. If defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
Vertex rounding cycle.

**FAGOR**



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.

-  Run the roughing operation.
-  Do not run the roughing operation.

### Maximum machining pass.

- $\Delta$  Maximum machining pass.
- If  $\Delta$  is positive, the cycle recalculates the step so all the passes are identical with the same value as or smaller than the one programmed.
  - If  $\Delta$  is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

### Machining feedrate.

- F Machining feedrate.



### Spindle speed.

- S Spindle turning speed (rpm) or cutting speed.

### Tool data.

- T Tool. If it is set to 0, the cycle does not execute the roughing operation.
- D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.



### Machining M functions.

-  Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.
-  Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.


## Machining parameters. Finishing.


To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.

-  Run the finishing operation.
-  Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes ( $\delta x$ ,  $\delta z$ ).

### Finishing stocks.

The cycle lets define a single finishing stock that will be applied depending on the cutter edge or 2 finishing stocks, one per axis (X, Z).

-  Define the finishing stock depending on the cutter edge. The finishing stock is measured on the edge of the tool.

-  Define 2 finishing stocks, one per axis, regardless of the type of tool being used.

- $\delta$  Finishing stock depending on the cutter edge.

- $\delta x$ ,  $\delta z$  Finishing stock on each axis.

4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
Vertex rounding cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)



**Machining feedrate.**

F Machining feedrate.

**Spindle speed.**

S Spindle turning speed (rpm) or cutting speed.

**Tool data.**

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**



Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Vertex rounding cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

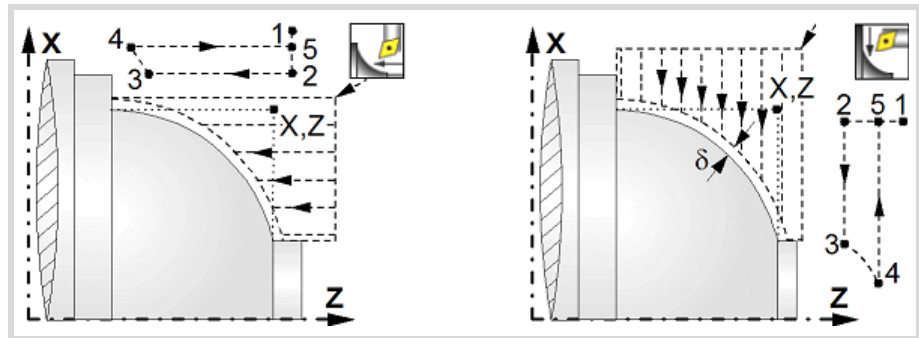
## 4.8.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The spindle starts with the speed selected and in the direction stated.
- 3 The tool approaches the theoretical corner (X, Z) in rapid, keeping the safety distance (Dx, Dz) selected according to axes X and Z.
- 4 Roughing operation (if it has been programmed).

The cycle runs successive turning or facing passes (according to the selected option) until reaching the final profile plus the finishing stocks. This operation is done with the conditions set for the roughing operation.

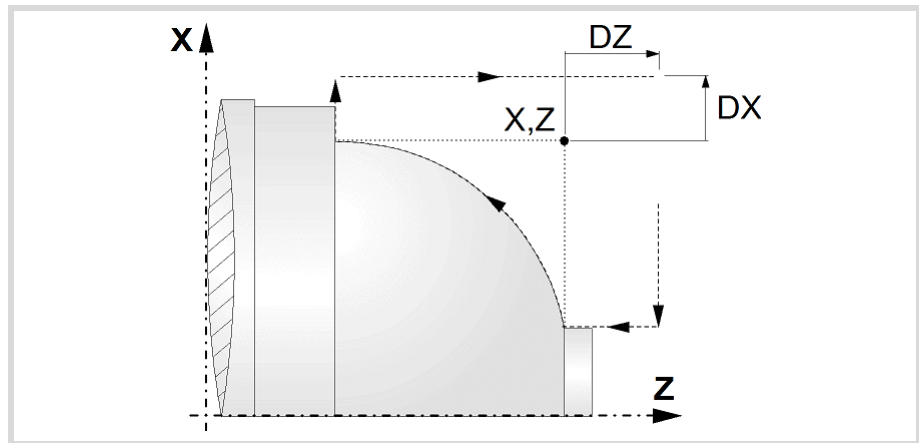
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



- 5 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle runs a single pass to remove the finishing stock. This operation is done with the conditions set for the finishing operation.



- 6 Once the operation or cycle has ended, the tool will return to the safety point (theoretical corner plus the safety distance).
- 7 The CNC will not stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

### Considerations.

- If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.
- If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the finishing operation, the tool will move to the safety point (starting point plus the safety distance).

# 4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
Vertex rounding cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## Operating manual (TC).

- The roughing operation is carried out in G05, the rounding radius may be modified using the instruction #ROUNDPAR. If #ROUNDPAR is not programmed, it assumes the default rounding radius set by machine parameters.
- The finishing operation is carried out in G07.

4.

**CANNED CYCLES OF THE EDITOR. TURNING AND FACING.**  
Vertex rounding cycle.

**FAGOR** 

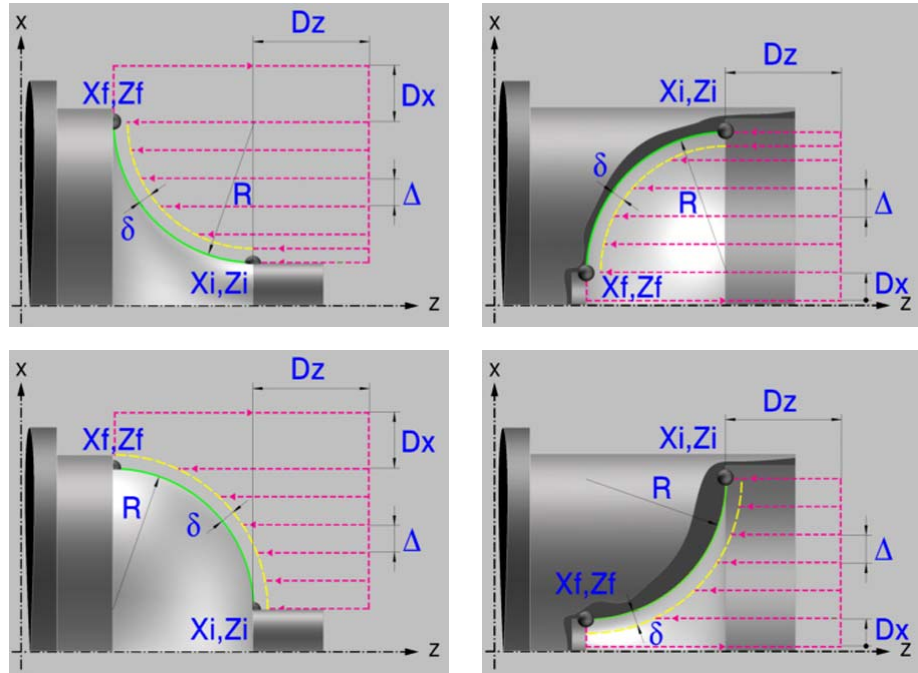
FAGOR AUTOMATION

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

(REF: 1709)

## 4.9 "Rounding between points" cycle.

The cycle allows outside and inside, concave and convex machining.



### Geometric parameters.

#### Inside or outside machining.



Outside machining.



Inside machining.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Concave and convex rounding.



Concave rounding.



Convex rounding.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Shape of the part before the tapered segment.



Initial concave corner.



Initial convex corner.

#### Shape of the part after the tapered segment.



Final concave corner.



Final convex corner.

4.

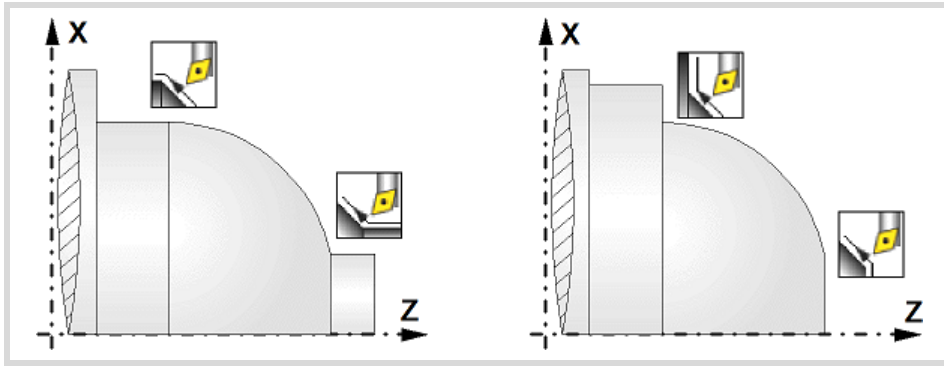
CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
"Rounding between points" cycle.



FAGOR AUTOMATION


CNC 8060  
CNC 8065

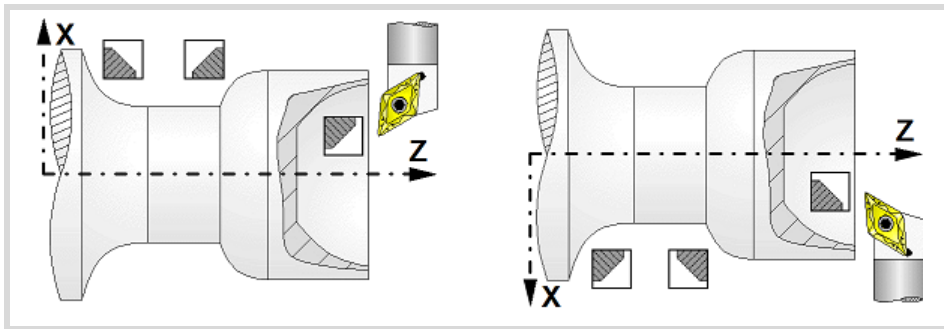
(REF: 1709)



**4.**

**Work quadrant.**

 Orientation of the corner to be machined.



**CANNED CYCLES OF THE EDITOR. TURNING AND FACING.**  
"Rounding between points" cycle.

**Coordinates of the profile's starting and final points.**

$X_i, Z_i$  Coordinates of the starting point

$X_f, Z_f$  Coordinates of the end point.

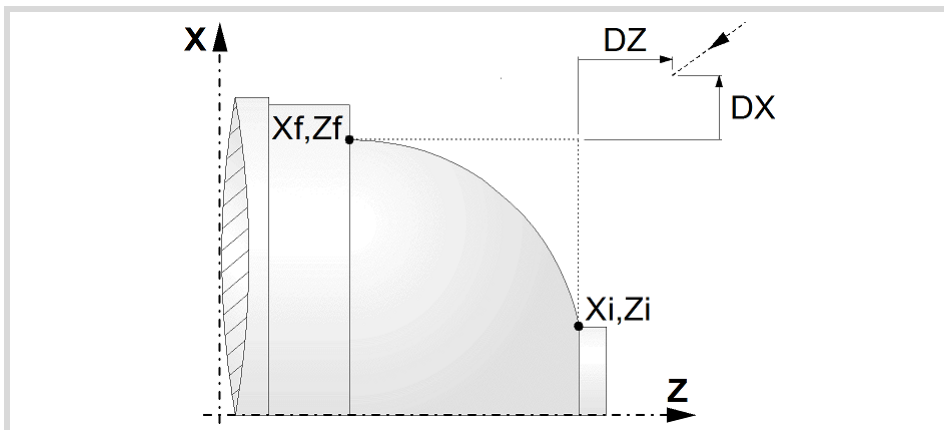
To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

**Safety distance.**

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the approach point coordinate with respect to the theoretical corner.

$D_x, D_z$  Safety distance.

The value of the safety distance on X is always defined in radius.



**Taper length.**

$R$  Rounding radius.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## Spindle parameters.

### Spindle turning direction

The cycle lets select the spindle turning direction.



Spindle clockwise.



Spindle counterclockwise.

### Coolant status.



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

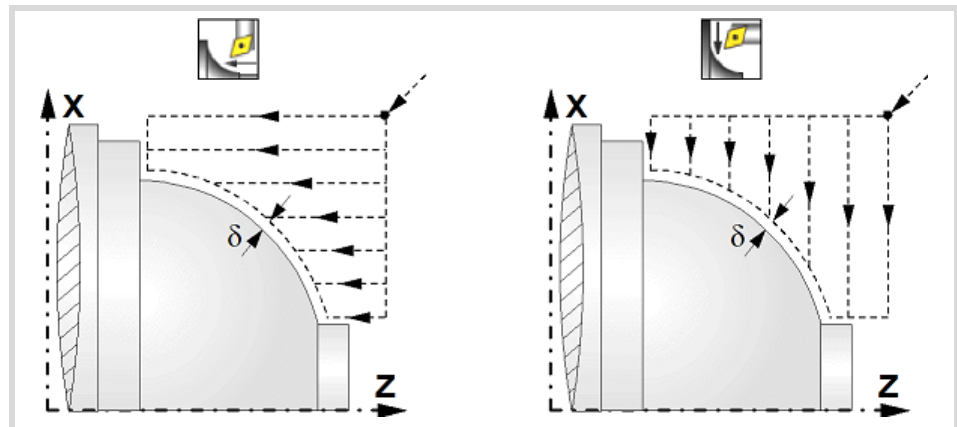
### Machining direction.



Machining in the turning direction.



Machining in the facing direction.



### Spindle work mode; rpm or Constant Surface Speed.



Constant turning speed.



Constant surface speed.

### Maximum spindle speed.

**Smax** Maximum spindle turning speed; if not programmed, the cycle will maintain the maximum active speed.

### Spindle range (gear).

**GEAR** Spindle gear; if not programmed, the cycle assumes a value of 0. If defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
"Rounding between points" cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the roughing operation.



Do not run the roughing operation.

### Maximum machining pass.

$\Delta$  Maximum machining pass.

- If  $\Delta$  is positive, the cycle recalculates the step so all the passes are identical with the same value as or smaller than the one programmed.
- If  $\Delta$  is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

### Machining feedrate.

F Machining feedrate.

### Spindle speed.

S Spindle turning speed (rpm) or cutting speed.

### Tool data.

T Tool. If it is set to 0, the cycle does not execute the roughing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

### Machining M functions.



Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

## Machining parameters. Finishing.

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the finishing operation.



Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes ( $\delta x$ ,  $\delta z$ ).

### Finishing stocks.

The cycle lets define a single finishing stock that will be applied depending on the cutter edge or 2 finishing stocks, one per axis (X, Z).



Define the finishing stock depending on the cutter edge. The finishing stock is measured on the edge of the tool.



Define 2 finishing stocks, one per axis, regardless of the type of tool being used.

$\delta$  Finishing stock depending on the cutter edge.

$\delta x$ ,  $\delta z$  Finishing stock on each axis.

4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
"Rounding between points" cycle.

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Machining feedrate.**

F Machining feedrate.

**Spindle speed.**

S Spindle turning speed (rpm) or cutting speed.

**Tool data.**

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

## 4.

**CANNED CYCLES OF THE EDITOR. TURNING AND FACING.**  
"Rounding between points" cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)



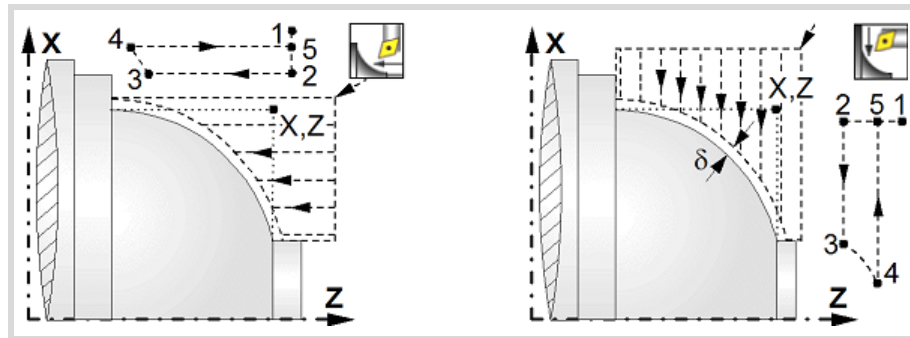
### 4.9.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The spindle starts with the speed selected and in the direction stated.
- 3 The tool approaches the theoretical corner (X, Z) in rapid, keeping the safety distance (Dx, Dz) selected according to axes X and Z.
- 4 Roughing operation (if it has been programmed).

The cycle runs successive turning or facing passes (according to the selected option) until reaching the final profile plus the finishing stocks. This operation is done with the conditions set for the roughing operation.

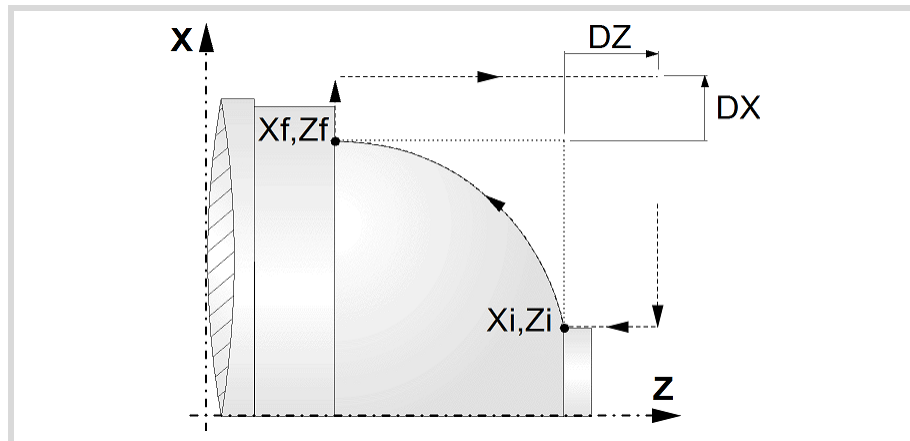
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



- 5 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle runs a single pass to remove the finishing stock. This operation is done with the conditions set for the finishing operation.



- 6 Once the operation or cycle has ended, the tool will return to the safety point (theoretical corner plus the safety distance).
- 7 The CNC will not stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

### Considerations.

- If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.
- If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the finishing operation, the tool will move to the safety point (starting point plus the safety distance).

# 4.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.  
"Rounding between points" cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

- The roughing operation is carried out in G05, the rounding radius may be modified using the instruction #ROUNDPAR. If #ROUNDPAR is not programmed, it assumes the default rounding radius set by machine parameters.
- The finishing operation is carried out in G07.

# 4.

## CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

"Rounding between points" cycle.



FAGOR AUTOMATION

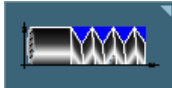
CNC 8060  
CNC 8065

(REF: 1709)

# CANNED CYCLES OF THE EDITOR. TAPPING.

# 5

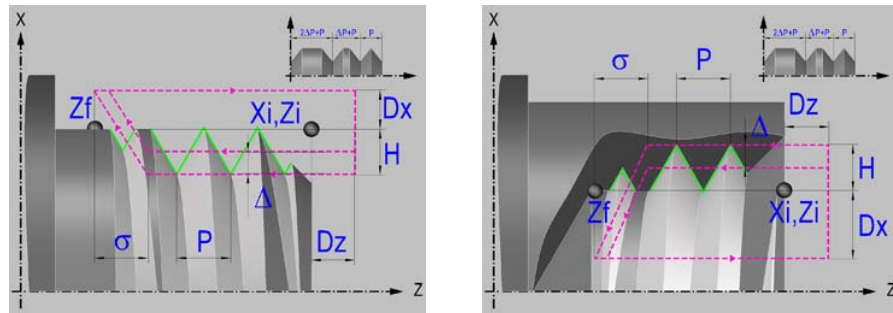
When pressing the softkey, the editor shows the cycle used last. When pressing the same softkey again, the menu shows all the cycles of the group.



- Longitudinal threading.
- Taper threading.
- Face threading.
- Thread repair.
- Multi-entry (start) threading.

## 5.1 Longitudinal threading cycle.

The cycle lets allows outside and inside threading.



### Geometric parameters.

#### Inside or outside machining.



Outside machining.



Inside machining.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Coordinates of the starting and final points.

$X_i, Z_i$  Coordinates of the starting point

$Z_f$  Z coordinate of the end point.

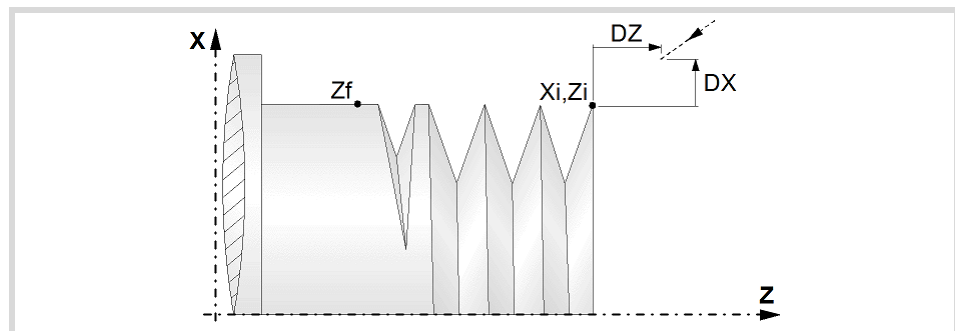
To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the approach point coordinate with respect to the starting point ( $X_i, Z_i$ ).

$D_x, D_z$  Safety distance.

The value of the safety distance on X is always defined in radius.



#### Distance to end of thread.

This parameter indicates at what distance from the end of the thread the tool begins exiting it. In this withdrawal movement threading is continued.

$\sigma$  Distance to end of thread. If programmed with a 0 value, the cycle calculates the distance to the end of the thread according to the dynamics of the machine, losing thread pitches during the exit from the thread.

5.

CANNED CYCLES OF THE EDITOR. TAPPING.  
Longitudinal threading cycle.

### Spindle angular position.

Indicates the angular position of the spindle or the angle referred to home to begin threading. It permits making multiple-entry threads without bringing the starting point back.



Thread without programming the entry angle.



Thread programming the entry angle.

$\omega$  Entry angle.

### Type of thread; free pitch thread or standard threads.

The cycle lets select the following types of standard threads.

P-H.	Free-pitch thread.
M (S.I.)	Metric thread with normal pitch (international system).
M (S.I.F.)	Metric thread with fine pitch (international system).
B.S.W. (W)	Whitworth thread with normal pitch.
B.S.F.	Whitworth thread with fine pitch.
U.N.C.	Unified American thread with normal pitch.
U.N.F.	Unified American thread with fine pitch.
B.S.P.	Whitworth gas thread.
API	API thread (oil pipe).

- If P-H (free-pitch thread) is selected, the pitch and the depth of the thread are selected directly by the user.
- When choosing a standard thread, the user only selects the diameter of the metrics, the pitch and the depth of that thread are calculated automatically. The standard threads are single entry cylindrical threads.
- When choosing an API thread, the user selects the type of thread. The standard threads are single entry cylindrical threads.

### Geometry of the thread (free-pitch threads).

On P-H threads (free-pitch threads), first one must choose between defining the pitch of the thread or the number of threads per inch.

P	Thread pitch in mm, inches or number of threads per inch. For a straight thread, as opposed to taper threads, the sign of the pitch is irrelevant.
H	Total thread depth. The total depth must be programmed in radii and with positive value.



Constant-pitch thread.



Variable-pitch thread.

$\Delta P$  On variable-pitch threads, the increment or decrement of the thread pitch per spindle turn. If not programmed, the thread will have a constant pitch.

### Geometry of the thread (standard threads).

On standard threads, the pitch and the depth of the thread are calculated automatically.

$\phi$  Thread diameter (except in API threads).



Units used to enter the thread data (mm or inches).

5.

CANNED CYCLES OF THE EDITOR. TAPPING.  
Longitudinal threading cycle.

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## Spindle parameters.

### Spindle turning direction

The cycle lets select the spindle turning direction.



Spindle clockwise.



Spindle counterclockwise.

### Coolant status.



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

### Spindle range (gear).

**GEAR** Spindle gear; if not programmed, the cycle assumes a value of 0. If defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

## Machining parameters.

### Spindle speed.

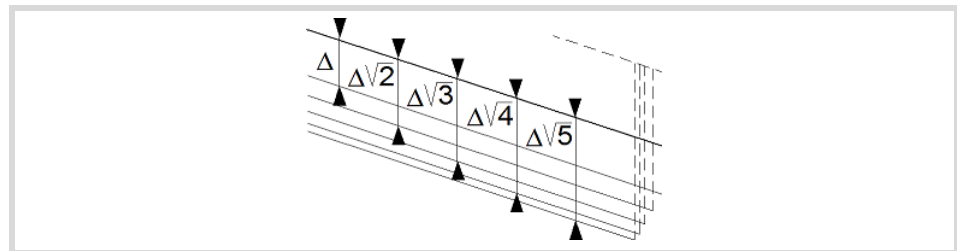
**S** Spindle speed.

### Depth of pass.

$\Delta$  Maximum penetration step.

$\delta$  Minimum penetration step.

The depth of each pass will be a function of the corresponding pass number. The cycle executes decreasing passes, the first one at the maximum penetration step until the minimum step is reached. If the penetrating increment (difference between penetrations) calculated by the CNC is smaller than the minimum decreasing penetrating step, the CNC assumes this latter value. The penetrations are calculated as follows  $\Delta$ ,  $\Delta\sqrt{2}$ ,  $\Delta\sqrt{3}$ ,  $\Delta\sqrt{4}$ ...



### Repeat the last pass.



The cycle repeats the last pass.



The cycle does not repeat the last pass.

5.

CANNED CYCLES OF THE EDITOR. TAPPING.  
Longitudinal threading cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Normal thread or blind thread.**



Normal thread. Parameter  $\sigma$  indicates at what distance from the end of the thread the tool begins exiting it. During the withdrawal movement, the cycle keeps threading.



Blind thread. The cycle finishes the thread trying to maintain the pitch until the final point is reached and then it exits the thread. In this case, the exit from the thread is more abrupt than in the rest of the path blends (joints).

When programming  $\sigma=0$ , unlike a blind thread, the exit from the thread is smoother, but it misses some thread steps in the exit segment. The cycle gives priority to the dynamics of the machine over the geometry of the thread.

When programming blind thread, unlike programming  $\sigma=0$ , the exit from the thread is more abrupt, but the threaded segment is larger. The cycle gives priority to the geometry of the thread over the dynamics of the machine.

**Tool data.**

- T Tool.
- D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**



Activating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivating the execution of M functions before machining.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**5.**

**CANNED CYCLES OF THE EDITOR. TAPPING.**  
 Longitudinal threading cycle.



FAGOR AUTOMATION

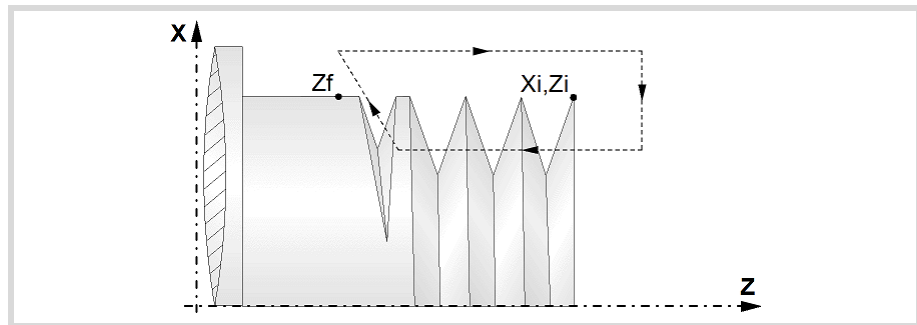
CNC 8060  
CNC 8065

(REF: 1709)

### 5.1.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 If the operation was programmed with another tool, the CNC will make a tool change moving to the change point if the machine so requires.
- 2 The spindle starts with the speed selected and in the direction stated. Depending on the spindle turning direction, the thread will be clockwise or counterclockwise.
- 3 The tool approaches in rapid up to the starting point, keeping the safety distance selected according to axes X and Z.
- 4 Tapping. It is done with radial penetration radial and by successive passes, until the full depth is reached. The depth of each pass will be a function of the corresponding pass number. Each of the threading passes is done in the following way:



First, rapid move to the corresponding programmed depth. Then, threading of the programmed section, along the Z up to the distance from the end of the thread (normal thread) or up to the final point (blind thread) and exit threading up to the final coordinate. And last, withdrawal in rapid to the approach point.

The electronic threading is carried out at 100% of the calculated feedrate and these values cannot be modified from the CNC's operator panel or via PLC. If the OEM allows it, (parameter THREADOVR), the user can modify the speed override from the operator panel and, in that case, the CNC will adapt the feedrate automatically respecting the thread pitch. In order to be able to modify the override, the active feed forward must be higher than 90%.

Neither the feedrate override nor the spindle speed override can be changed in the last pass; this pass will be run with the override set in the previous pass.

- 5 Once the operation or cycle has ended, the tool will return to the safety point. When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
- 6 The CNC will not stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

# 5.

CANNED CYCLES OF THE EDITOR. TAPPING.  
Longitudinal threading cycle.



FAGOR AUTOMATION

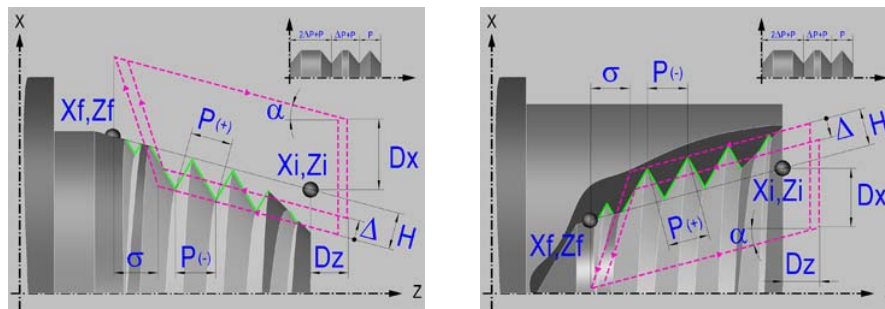
CNC 8060  
CNC 8065

(REF: 1709)



## 5.2 Taper threading cycle.

The cycle lets allows outside and inside threading.



### Geometric parameters.

#### Inside or outside machining.



Outside machining.



Inside machining.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Type of end point.

The end point may be defined in several ways.



Coordinates of the end point ( $X_f, Z_f$ ).



Angle and length of the thread ( $\alpha, \Delta Z$ ).



Angle of the thread and end coordinate ( $\alpha, Z_f$ ).

#### Coordinates of the starting and final points.

$X_i, Z_i$  Coordinates of the starting point

$X_f, Z_f$  Coordinates of the end point. By default,  $X_f$  assumes the value set for  $X_i$ .

$\alpha$  Thread angle.

$\Delta Z$  Thread length.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the approach point coordinate with respect to the starting point ( $X_i, Z_i$ ).

$D_x, D_z$  Safety distance.

The value of the safety distance on X is always defined in radius.

5.

CANNED CYCLES OF THE EDITOR. TAPPING.

Taper threading cycle.

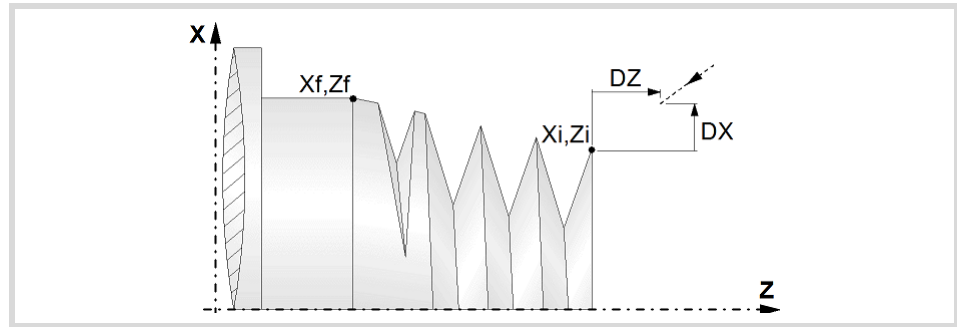
**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 5.

CANNED CYCLES OF THE EDITOR. TAPPING.  
Taper threading cycle.**Distance to end of thread.**

This parameter indicates at what distance from the end of the thread the tool begins exiting it. In this withdrawal movement threading is continued.

- σ Distance to end of thread. If programmed with a 0 value, the cycle calculates the distance to the end of the thread according to the dynamics of the machine, losing thread pitches during the exit from the thread.

**Spindle angular position.**

Indicates the angular position of the spindle or the angle referred to home to begin threading. It permits making multiple-entry threads without bringing the starting point back.



Thread without programming the entry angle.



Thread programming the entry angle.

- ω Entry angle.

**Type of thread; free pitch thread or standard threads.**

The cycle lets select the following types of standard threads. It is possible to select a taper thread and also a standard thread, in this case, it will calculate the pitch and depth that would correspond to the standard cylindrical thread and it will be assigned to that taper thread.

- P-H. Free-pitch thread.
- M (S.I.) Metric thread with normal pitch (international system).
- M (S.I.F.) Metric thread with fine pitch (international system).
- B.S.W. (W) Whitworth thread with normal pitch.
- B.S.F. Whitworth thread with fine pitch.
- U.N.C. Unified American thread with normal pitch.
- U.N.F. Unified American thread with fine pitch.
- B.S.P. Whitworth gas thread.
- API API thread (oil pipe).

- If P-H (free-pitch thread) is selected, the pitch and the depth of the thread are selected directly by the user.
- When choosing a standard thread, the user only selects the diameter of the metrics, the pitch and the depth of that thread are calculated automatically. The standard threads are single entry cylindrical threads.
- When choosing an API thread, the user selects the type of thread. The standard threads are single entry cylindrical threads.

**Geometry of the thread (free-pitch threads).**

On P-H threads (free-pitch threads), first one must choose between defining the pitch of the thread or the number of threads per inch.

- P Thread pitch in mm, inches or number of threads per inch. The thread pitch may be set along the taper of the thread or along its associated axis.
  - $P > 0$ ; When programming the pitch along the taper of the thread.
  - $P < 0$ ; To program the pass along according to the associated axis.

H Total thread depth. The total depth must be programmed in radii and with positive value.



Constant-pitch thread.



Variable-pitch thread.

$\Delta P$  On variable-pitch threads, the increment or decrement of the thread pitch per spindle turn. If not programmed, the thread will have a constant pitch.

### Geometry of the thread (standard threads).

On standard threads, the pitch and the depth of the thread are calculated automatically.

$\phi$  Thread diameter (except in API threads).



Units used to enter the thread data (mm or inches).

## Spindle parameters.

### Spindle turning direction

The cycle lets select the spindle turning direction.



Spindle clockwise.



Spindle counterclockwise.

### Coolant status.



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

### Spindle range (gear).

GEAR Spindle gear; if not programmed, the cycle assumes a value of 0. If defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

## Machining parameters.

### Spindle speed.

S Spindle speed.

### Depth of pass.

$\Delta$  Maximum penetration step.



Constant depth of pass, with a smaller or same value as the maximum programmed.

5.

CANNED CYCLES OF THE EDITOR. TAPPING.

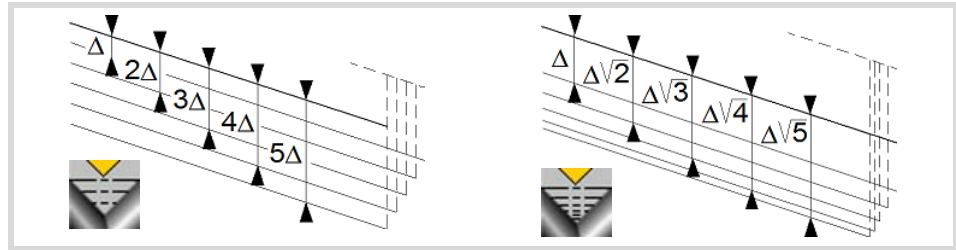
Taper threading cycle.

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)



**5.**

CANNED CYCLES OF THE EDITOR. TAPPING.  
Taper threading cycle.



The depth of each pass will be a function of the corresponding pass number. The cycle executes decreasing passes, the first one at the maximum penetration step until the minimum step is reached. If the penetrating increment (difference between penetrations) calculated by the CNC is smaller than the minimum decreasing penetrating step, the CNC assumes this latter value. The penetrations are calculated as follows  $\Delta, \Delta\sqrt{2}, \Delta\sqrt{3}, \Delta\sqrt{4}...$

$\delta$  Minimum penetration step.

**Repeat the last pass.**



The cycle repeats the last pass.



The cycle does not repeat the last pass.

**Normal thread or blind thread.**



Normal thread. Parameter  $\sigma$  indicates at what distance from the end of the thread the tool begins exiting it. During the withdrawal movement, the cycle keeps threading.



Blind thread. The cycle finishes the thread trying to maintain the pitch until the final point is reached and then it exits the thread. In this case, the exit from the thread is more abrupt than in the rest of the path blends (joints).

When programming  $\sigma=0$ , unlike a blind thread, the exit from the thread is smoother, but it misses some thread steps in the exit segment. The cycle gives priority to the dynamics of the machine over the geometry of the thread.

When programming blind thread, unlike programming  $\sigma=0$ , the exit from the thread is more abrupt, but the threaded segment is larger. The cycle gives priority to the geometry of the thread over the dynamics of the machine.

**Type of tool penetration.**



Radial penetration.



Penetration by starting flank. The CNC will ask for the cutter's penetration angle ( $\alpha$ ).



Penetration in zigzag, alternates between radial and starting flank. The CNC will ask for the cutter's penetration angle ( $\alpha$ ).



Penetration by final flank. The CNC will ask for the cutter's penetration angle ( $\alpha$ ).



Penetration in zigzag, alternates between radial and final flank. The CNC will ask for the cutter's penetration angle ( $\alpha$ ).



Penetration in zigzag, alternates between starting and final flank. The CNC will ask for the cutter's penetration angle ( $\alpha$ ) (leadscrew threading).

$\alpha$  Tool penetration angle.

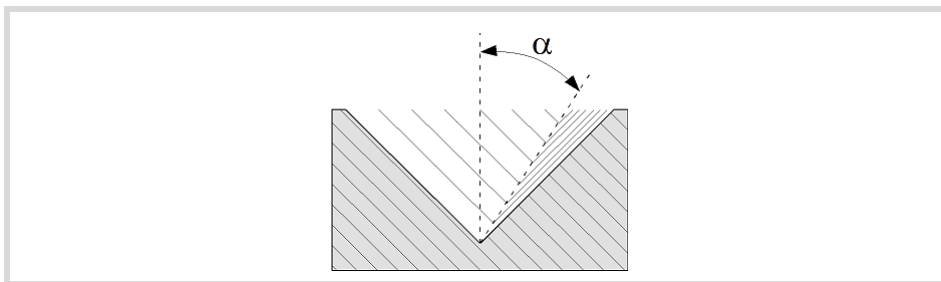
We recommend to use a penetration angle smaller than half the tool angle, never larger. The thread cannot be machined if the penetration angle is larger than half the tool angle. If the



CNC 8060  
CNC 8065

(REF: 1709)

penetration angle is equal to half the tool angle, it will rub the flank of the thread at each cutting pass.



#### Tool data.

T Tool.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

#### Machining M functions.



Activating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivating the execution of M functions before machining.

The editor will only show this option if the user has configured the editor to allow programming M functions.

5.

CANNED CYCLES OF THE EDITOR. TAPPING.

Taper threading cycle.

**FAGOR** 

FAGOR AUTOMATION

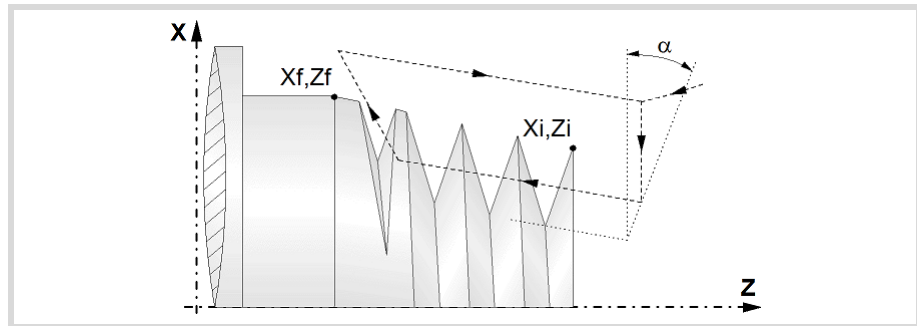
CNC 8060  
CNC 8065

(REF: 1709)

## 5.2.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 If the operation was programmed with another tool, the CNC will make a tool change moving to the change point if the machine so requires.
- 2 The spindle starts with the speed selected and in the direction stated. Depending on the spindle turning direction, the thread will be clockwise or counterclockwise.
- 3 The tool approaches in rapid up to the starting point, keeping the safety distance selected according to axes X and Z.
- 4 Tapping. It is done in successive passes until the full depth is reached. The depth of each pass depends on the selected type of penetration; depending on the corresponding pass number or keeping the increment between passes constant. Each of the threading passes is done in the following way:



First, rapid move to the corresponding programmed depth. This positioning is done according to the selected tool penetration angle ( $\alpha$ ). Then, threading of the programmed section up to the distance from the end of the thread (normal thread) or up to the final point (blind thread) and exit threading up to the final coordinate. And last, withdrawal in rapid to the approach point.

The electronic threading is carried out at 100% of the calculated feedrate and these values cannot be modified from the CNC's operator panel or via PLC. If the OEM allows it, (parameter THREADOVR), the user can modify the speed override from the operator panel and, in that case, the CNC will adapt the feedrate automatically respecting the thread pitch. In order to be able to modify the override, the active feed forward must be higher than 90%. It is recommended not to change the spindle speed override on threads with flank infeed.

Neither the feedrate override nor the spindle speed override can be changed in the last pass; this pass will be run with the override set in the previous pass.

- 5 Once the operation or cycle has ended, the tool will return to the safety point. When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
- 6 The CNC will not stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

# 5.

CANNED CYCLES OF THE EDITOR. TAPPING.  
Taper threading cycle.

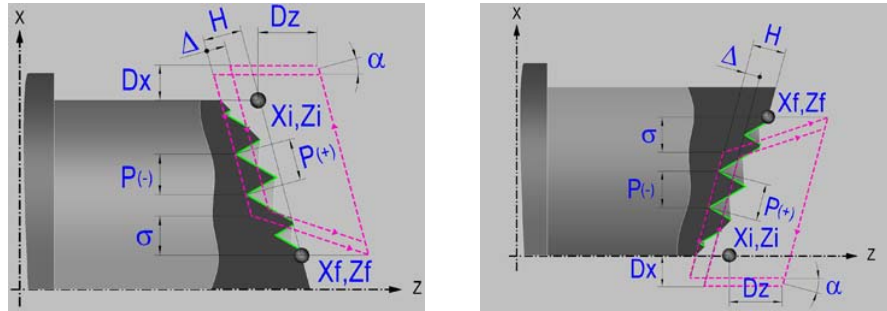


FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

### 5.3 Face threading cycle.



The configurator of the cycle editor lets invert the starting point and the final point of the cycle. The cycle changes the corresponding help screen.

#### Geometric parameters.

##### Coordinates of the starting and final points.

$X_i, Z_i$  Coordinates of the starting point

$X_f, Z_f$  Coordinates of the end point. By default,  $Z_f$  assumes the value set for  $Z_i$ .

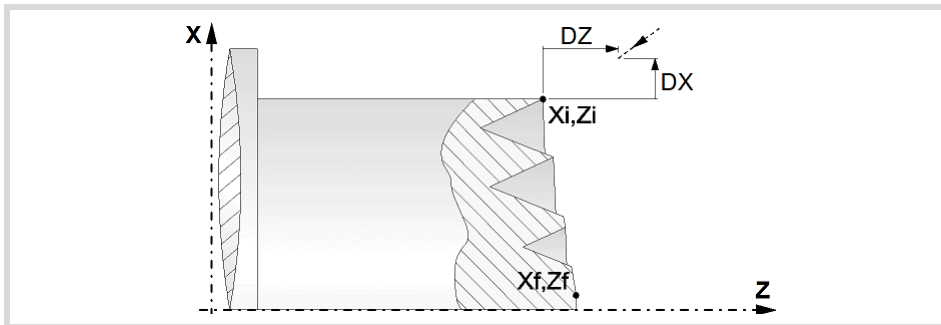
To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

##### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the approach point coordinate with respect to the starting point ( $X_i, Z_i$ ).

$D_x, D_z$  Safety distance.

The value of the safety distance on X is always defined in radius.



##### Distance to end of thread.

This parameter indicates at what distance from the end of the thread the tool begins exiting it. In this withdrawal movement threading is continued.

$\sigma$  Distance to end of thread. If programmed with a 0 value, the cycle calculates the distance to the end of the thread according to the dynamics of the machine, losing thread pitches during the exit from the thread.

##### Spindle angular position.

Indicates the angular position of the spindle or the angle referred to home to begin threading. It permits making multiple-entry threads without bringing the starting point back.



Thread without programming the entry angle.



Thread programming the entry angle.

# 5.

CANNED CYCLES OF THE EDITOR. TAPPING.  
Face threading cycle.

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

ω Entry angle.

**Thread geometry.**

First one must choose between defining the pitch of the thread or the number of threads per inch.

P Thread pitch in mm, inches or number of threads per inch. The thread pitch may be set along the taper of the thread or along its associated axis.

- P > 0; When programming the pitch along the taper of the thread.
- P < 0; To program the pass along according to the associated axis.

H Total thread depth. The total depth must be programmed in radii and with positive value.

**Spindle parameters.**

**Spindle turning direction**

The cycle lets select the spindle turning direction.



Spindle clockwise.



Spindle counterclockwise.

**Coolant status.**



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

**Spindle range (gear).**

GEAR Spindle gear; if not programmed, the cycle assumes a value of 0. If defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Machining parameters.**

**Spindle speed.**

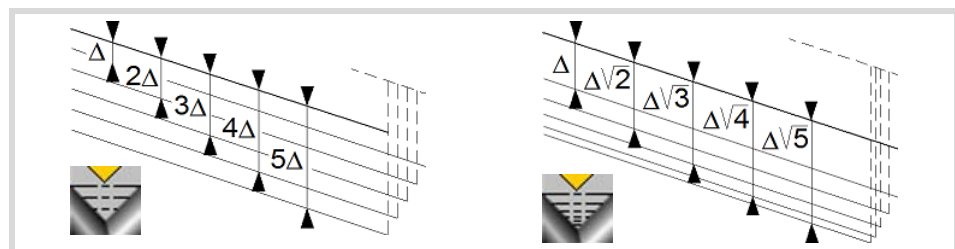
S Spindle speed.

**Depth of pass.**

Δ Maximum penetration step.



Constant depth of pass, with a smaller or same value as the maximum programmed.



**5.**  
CANNED CYCLES OF THE EDITOR. TAPPING.  
Face threading cycle.



CNC 8060  
CNC 8065

(REF: 1709)





The depth of each pass will be a function of the corresponding pass number. The cycle executes decreasing passes, the first one at the maximum penetration step until the minimum step is reached. If the penetrating increment (difference between penetrations) calculated by the CNC is smaller than the minimum decreasing penetrating step, the CNC assumes this latter value. The penetrations are calculated as follows  $\Delta, \Delta\sqrt{2}, \Delta\sqrt{3}, \Delta\sqrt{4}...$

$\delta$  Minimum penetration step.

**Repeat the last pass.**



The cycle repeats the last pass.



The cycle does not repeat the last pass.

**Normal thread or blind thread.**



Normal thread. Parameter  $\sigma$  indicates at what distance from the end of the thread the tool begins exiting it. During the withdrawal movement, the cycle keeps threading.



Blind thread. The cycle finishes the thread trying to maintain the pitch until the final point is reached and then it exits the thread. In this case, the exit from the thread is more abrupt than in the rest of the path blends (joints).

When programming  $\sigma=0$ , unlike a blind thread, the exit from the thread is smoother, but it misses some thread steps in the exit segment. The cycle gives priority to the dynamics of the machine over the geometry of the thread.

When programming blind thread, unlike programming  $\sigma=0$ , the exit from the thread is more abrupt, but the threaded segment is larger. The cycle gives priority to the geometry of the thread over the dynamics of the machine.

**Type of tool penetration.**



Radial penetration.



Penetration by starting flank. The CNC will ask for the cutter's penetration angle ( $\alpha$ ).



Penetration in zigzag, alternates between radial and starting flank. The CNC will ask for the cutter's penetration angle ( $\alpha$ ).



Penetration by final flank. The CNC will ask for the cutter's penetration angle ( $\alpha$ ).



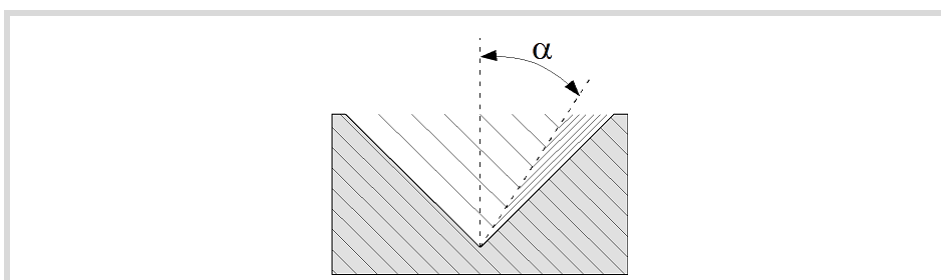
Penetration in zigzag, alternates between radial and final flank. The CNC will ask for the cutter's penetration angle ( $\alpha$ ).



Penetration in zigzag, alternates between starting and final flank. The CNC will ask for the cutter's penetration angle ( $\alpha$ ) (leadscrew threading).

$\alpha$  Tool penetration angle.

We recommend to use a penetration angle smaller than half the tool angle, never larger. The thread cannot be machined if the penetration angle is larger than half the tool angle. If the penetration angle is equal to half the tool angle, it will rub the flank of the thread at each cutting pass.



**5.**

CANNED CYCLES OF THE EDITOR. TAPPING.  
Face threading cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Tool data.**

T Tool.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

Activating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivating the execution of M functions before machining.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**5.**

**CANNED CYCLES OF THE EDITOR. TAPPING.**  
Face threading cycle.



FAGOR AUTOMATION

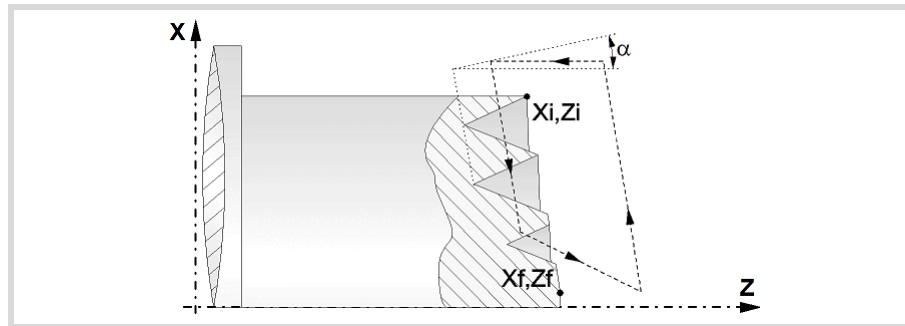
CNC 8060  
CNC 8065

(REF: 1709)

### 5.3.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 If the operation was programmed with another tool, the CNC will make a tool change moving to the change point if the machine so requires.
- 2 The spindle starts with the speed selected and in the direction stated. Depending on the spindle turning direction, the thread will be clockwise or counterclockwise.
- 3 The tool approaches in rapid up to the starting point, keeping the safety distance selected according to axes X and Z.
- 4 Tapping. It is done in successive passes until the full depth is reached. The depth of each pass depends on the selected type of penetration; depending on the corresponding pass number or keeping the increment between passes constant. Each of the threading passes is done in the following way:



First, rapid move to the corresponding programmed depth. This positioning is done according to the selected tool penetration angle ( $\alpha$ ). Then, threading of the programmed section up to the distance from the end of the thread (normal thread) or up to the final point (blind thread) and exit threading up to the final coordinate. And last, withdrawal in rapid to the approach point.

The electronic threading is carried out at 100% of the calculated feedrate and these values cannot be modified from the CNC's operator panel or via PLC. If the OEM allows it, (parameter THREADOVR), the user can modify the speed override from the operator panel and, in that case, the CNC will adapt the feedrate automatically respecting the thread pitch. In order to be able to modify the override, the active feed forward must be higher than 90%. It is recommended not to change the spindle speed override on threads with flank infeed.

Neither the feedrate override nor the spindle speed override can be changed in the last pass; this pass will be run with the override set in the previous pass.

- 5 Once the operation or cycle has ended, the tool will return to the safety point. When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
- 6 The CNC will not stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

# 5.

CANNED CYCLES OF THE EDITOR. TAPPING.  
Face threading cycle.

**FAGOR** 

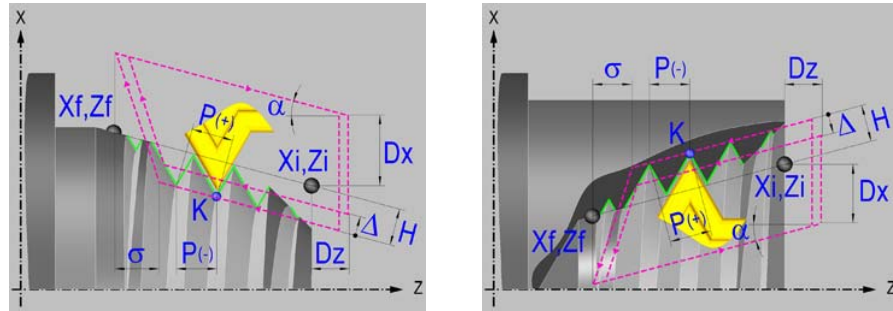
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 5.4 Thread repair cycle.

The cycle allows outside and inside thread repair.



For face thread repair, the configurator of the cycle editor lets invert the starting point and the final point of the cycle. The cycle changes the corresponding help screen.

### Geometric parameters.

#### Inside or outside machining.



Outside machining.



Inside machining.



Face threading.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Type of end point (except face threading).

The end point may be defined in several ways.



Coordinates of the end point ( $X_f, Z_f$ ).



Angle and length of the thread ( $\alpha, \Delta Z$ ).



Angle of the thread and end coordinate ( $\alpha, Z_f$ ).

#### Coordinates of the starting and final points.

$X_i, Z_i$  Coordinates of the starting point

$X_f, Z_f$  Coordinates of the end point. By default,  $X_f$  assumes the value set for  $X_i$ .

$\alpha$  Thread angle.

$\Delta Z$  Thread length.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

5.

CANNED CYCLES OF THE EDITOR. TAPPING.  
Thread repair cycle.

**Full or partial thread repair.**

The cycle offers the possibility of making threads with the start and/or end in different directions to that of their geometry. This option may be used to make threads that do not start or end in "edges" and execute partial thread repairs.



**Full thread repair.**

The cycle makes a standard thread starting in the same direction as its geometry and ending in the same direction; or in the direction defined for the programmed exit from the thread.



**Partial thread repair.**

The cycle starts each pass with a segment at an angle, needed so during the first step of the last pass reaches the depth of the thread. Likewise, the cycle ends each pass with a segment at an angle, needed so during the last step of the last pass reaches the surface of the thread.

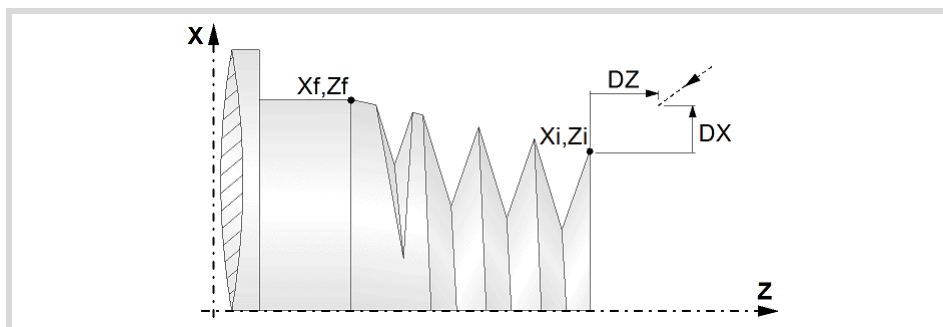
In partial thread repair, the starting point and the final (end) point correspond to the segment to be machined. The entry to the thread (start) is calculated automatically.

**Safety distance.**

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the approach point coordinate with respect to the starting point ( $X_i, Z_i$ ).

$D_x, D_z$  Safety distance.

The value of the safety distance on X is always defined in radius.



**Distance to end of thread.**

This parameter indicates at what distance from the end of the thread the tool begins exiting it. In this withdrawal movement threading is continued.

$\sigma$  Distance to end of thread. If programmed with a 0 value, the cycle calculates the distance to the end of the thread according to the dynamics of the machine, losing thread pitches during the exit from the thread.

**Number of thread entries**

$N$  Number of thread entries

**Spindle angular position.**

$K$  Z coordinate of the root (valley) of the thread.

$W$  Angular position of the spindle in K position.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

**5.**

**CANNED CYCLES OF THE EDITOR. TAPPING.**  
Thread repair cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Type of thread; free-pitch thread or standard thread (the standard threads are not available in face thread repair).**

The cycle lets select the following types of standard threads. It is possible to select a taper thread and also a standard thread, in this case, it will calculate the pitch and depth that would correspond to the standard cylindrical thread and it will be assigned to that taper thread.

P-H.	Free-pitch thread.
M (S.I.)	Metric thread with normal pitch (international system).
M (S.I.F.)	Metric thread with fine pitch (international system).
B.S.W. (W)	Whitworth thread with normal pitch.
B.S.F.	Whitworth thread with fine pitch.
U.N.C.	Unified American thread with normal pitch.
U.N.F.	Unified American thread with fine pitch.
B.S.P.	Whitworth gas thread.
API	API thread (oil pipe).

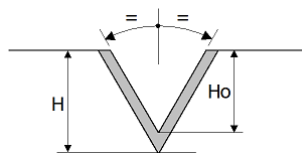
- If P-H (free-pitch thread) is selected, the pitch and the depth of the thread are selected directly by the user.
- When choosing a standard thread, the user only selects the diameter of the metrics, the pitch and the depth of that thread are calculated automatically. The standard threads are single entry cylindrical threads.
- When choosing an API thread, the user selects the type of thread. The standard threads are single entry cylindrical threads.

**Geometry of the thread (free-pitch threads).**

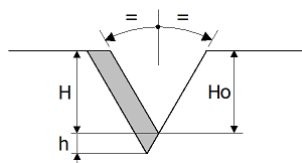
On P-H threads (free-pitch threads), first one must choose between defining the pitch of the thread or the number of threads per inch.

- P Thread pitch in mm, inches or number of threads per inch. The thread pitch may be set along the taper of the thread or along its associated axis.
- $P > 0$ ; When programming the pitch along the taper of the thread.
  - $P < 0$ ; To program the pass along according to the associated axis.
- H Total thread depth. The total depth must be programmed in radii and with positive value.
- h Penetration step in thread repair maintaining the geometry. This parameter must be programmed in radius and with a positive value. This parameter is useful for asymmetrical cutters. The penetrating direction is defined by the cutting angle.
- When programming parameter "h", the tool data (cutter angle and the cutting angle) must be correctly defined in the tool table.

Starting with a thread whose depth is  $H_0$ , the following examples show the difference between increasing the value of parameter H and programming parameter h.



When programming a depth (H) greater than the initial one ( $H_0$ ), the cycle increases the depth of the thread ignoring the existing flanks.



When programming the original depth ( $H=H_0$ ) plus an increment (h), the cycle increases the depth but maintaining the entry (start) flank.

5.

CANNED CYCLES OF THE EDITOR. TAPPING.  
Thread repair cycle.

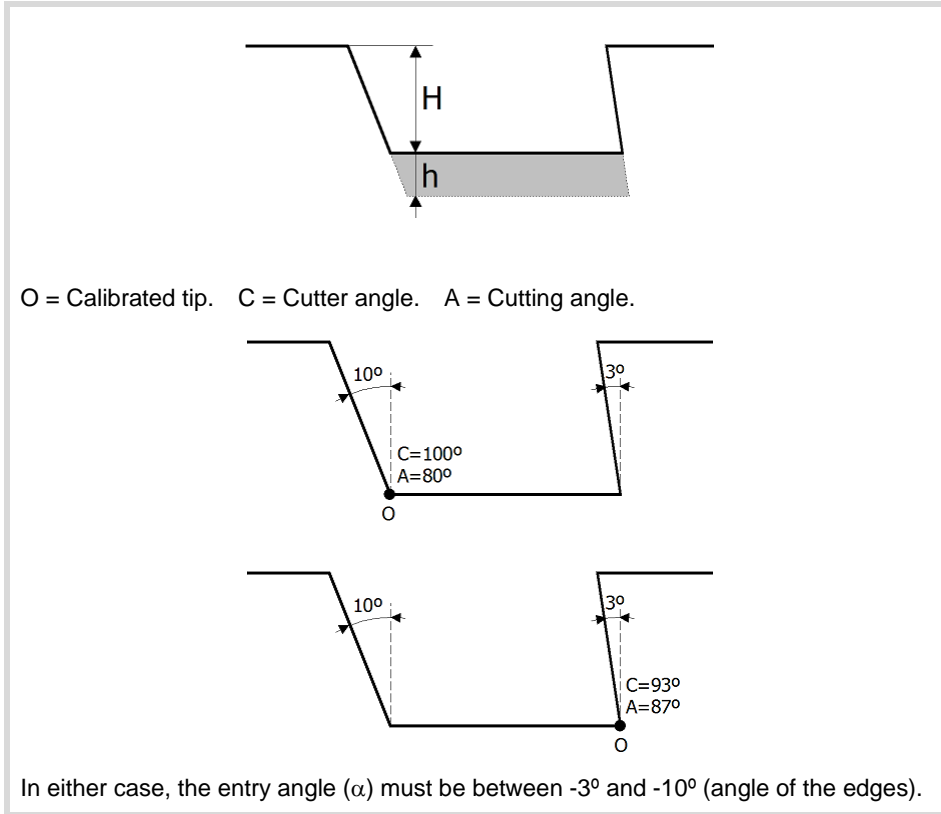


FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

- This feature is also valid for asymmetrical dual-tip tools where the entry angle must always be respected (not ignored). The tool data (cutter angle and the cutting angle) must be correctly defined in the tool table depending on the calibrated tip.




# 5.

**CANNED CYCLES OF THE EDITOR. TAPPING.**  
Thread repair cycle.

### Geometry of the thread (standard threads).

On standard threads, the pitch and the depth of the thread are calculated automatically.


$\phi$  Thread diameter (except in API threads).


 Units used to enter the thread data (mm or inches).

### Spindle parameters.


#### Spindle turning direction


The cycle lets select the spindle turning direction.

 Spindle clockwise.

 Spindle counterclockwise.

#### Coolant status.

 Coolant ON. The CNC outputs the M8 function to the PLC.

 Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Spindle range (gear).**

GEAR Spindle gear; if not programmed, the cycle assumes a value of 0. If defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Machining parameters.**

**Spindle speed.**

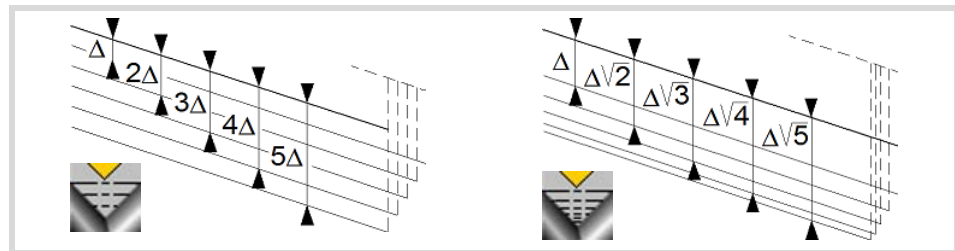
S Spindle speed.

**Depth of pass.**

$\Delta$  Maximum penetration step.



Constant depth of pass, with a smaller or same value as the maximum programmed.



The depth of each pass will be a function of the corresponding pass number. The cycle executes decreasing passes, the first one at the maximum penetration step until the minimum step is reached. If the penetrating increment (difference between penetrations) calculated by the CNC is smaller than the minimum decreasing penetrating step, the CNC assumes this latter value. The penetrations are calculated as follows  $\Delta, \Delta\sqrt{2}, \Delta\sqrt{3}, \Delta\sqrt{4}...$

$\delta$  Minimum penetration step.

**Repeat the last pass.**



The cycle repeats the last pass.



The cycle does not repeat the last pass.

**Normal thread or blind thread.**



Normal thread. Parameter  $\sigma$  indicates at what distance from the end of the thread the tool begins exiting it. During the withdrawal movement, the cycle keeps threading.



Blind thread. The cycle finishes the thread trying to maintain the pitch until the final point is reached and then it exits the thread. In this case, the exit from the thread is more abrupt than in the rest of the path blends (joints).

When programming  $\sigma=0$ , unlike a blind thread, the exit from the thread is smoother, but it misses some thread steps in the exit segment. The cycle gives priority to the dynamics of the machine over the geometry of the thread.

When programming blind thread, unlike programming  $\sigma=0$ , the exit from the thread is more abrupt, but the threaded segment is larger. The cycle gives priority to the geometry of the thread over the dynamics of the machine.

5.

CANNED CYCLES OF THE EDITOR. TAPPING.  
Thread repair cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)



**Type of tool penetration.**



Radial penetration.



Penetration by starting flank. The CNC will ask for the cutter's penetration angle ( $\alpha$ ).



Penetration in zigzag, alternates between radial and starting flank. The CNC will ask for the cutter's penetration angle ( $\alpha$ ).



Penetration by final flank. The CNC will ask for the cutter's penetration angle ( $\alpha$ ).



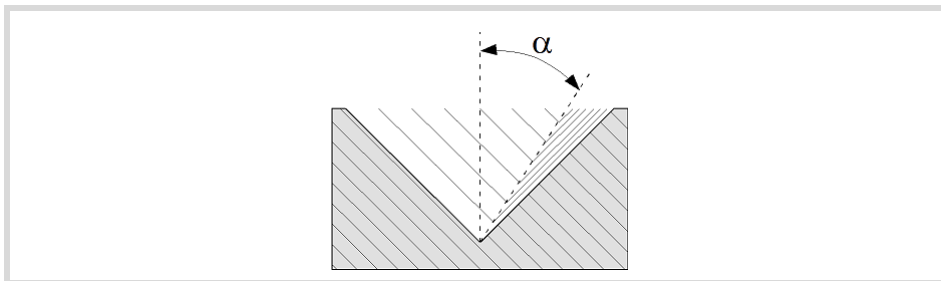
Penetration in zigzag, alternates between radial and final flank. The CNC will ask for the cutter's penetration angle ( $\alpha$ ).



Penetration in zigzag, alternates between starting and final flank. The CNC will ask for the cutter's penetration angle ( $\alpha$ ) (leadscrew threading).

$\alpha$  Tool penetration angle.

We recommend to use a penetration angle smaller than half the tool angle, never larger. The thread cannot be machined if the penetration angle is larger than half the tool angle. If the penetration angle is equal to half the tool angle, it will rub the flank of the thread at each cutting pass.



**Tool data.**

T Tool.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**



Activating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivating the execution of M functions before machining.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**5.**

**CANNED CYCLES OF THE EDITOR. TAPPING.**

Thread repair cycle.



FAGOR AUTOMATION

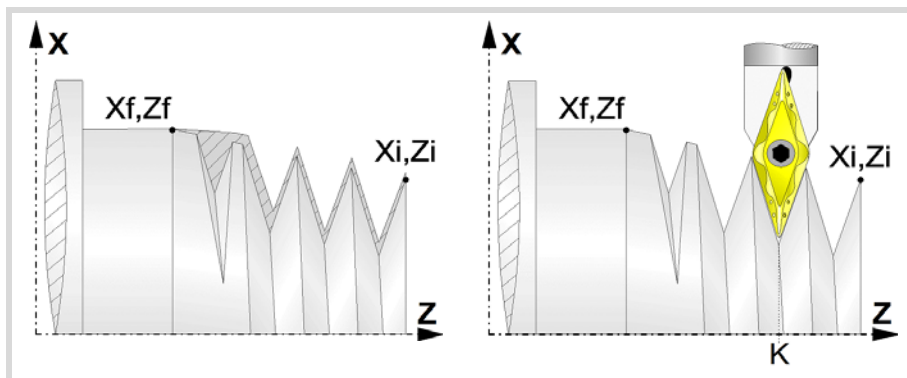
CNC 8060  
CNC 8065

(REF: 1709)

## 5.4.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 Define the dimensions of the thread like at the rest of the levels and the coordinates of one of the roots. To define the coordinates of the root, the CNC must know the spindle position; for that, just orient the spindle (M19) once after power-up.
- 2 With the spindle stopped, take the tool to be used for thread repair to one of the roots of the thread. Once there, take the Z value and the angular position of the spindle at the root (parameters K W). We suggest using the Teach-in mode to define the coordinates.
- 3 The CNC makes a new thread on the existing thread, but maintaining the roots and inclinations of the current thread. The machining steps for this cycle are identical to those for taper threading described earlier.



# 5.

CANNED CYCLES OF THE EDITOR. TAPPING.  
Thread repair cycle.

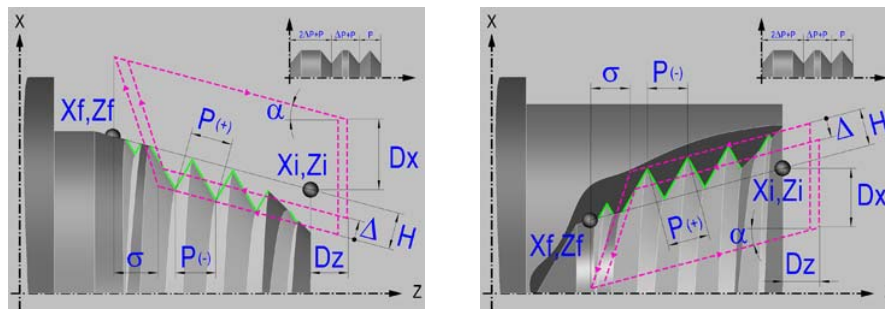
**FAGOR**   
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 5.5 Multi-entry threading cycle.

The cycle lets allows outside and inside threading.



### Geometric parameters.

#### Inside or outside machining.



Outside machining.



Inside machining.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Type of end point.

The end point may be defined in several ways.



Coordinates of the end point ( $X_f, Z_f$ ).



Angle and length of the thread ( $\alpha, \Delta Z$ ).



Angle of the thread and end coordinate ( $\alpha, Z_f$ ).

#### Coordinates of the starting and final points.

$X_i, Z_i$  Coordinates of the starting point

$X_f, Z_f$  Coordinates of the end point. By default,  $X_f$  assumes the value set for  $X_i$ .

$\alpha$  Thread angle.

$\Delta Z$  Thread length.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the approach point coordinate with respect to the starting point ( $X_i, Z_i$ ).

$D_x, D_z$  Safety distance.

The value of the safety distance on X is always defined in radius.

5.

CANNED CYCLES OF THE EDITOR. TAPPING.  
Multi-entry threading cycle.

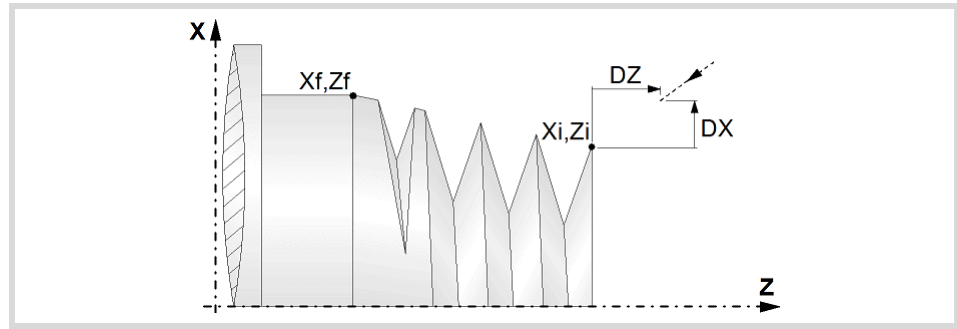


FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 5.

CANNED CYCLES OF THE EDITOR. TAPPING.  
Multi-entry threading cycle.**Distance to end of thread.**

This parameter indicates at what distance from the end of the thread the tool begins exiting it. In this withdrawal movement threading is continued.

- σ Distance to end of thread. If programmed with a 0 value, the cycle calculates the distance to the end of the thread according to the dynamics of the machine, losing thread pitches during the exit from the thread.

**Number of thread entries**

N Number of thread entries

**Spindle angular position.**

Indicates the angular position of the spindle or the angle referred to home to begin threading.



Thread without programming the entry angle.



Thread programming the entry angle.

- ω Entry angle.

**Type of thread; free pitch thread or standard threads.**

The cycle lets select the following types of standard threads. It is possible to select a taper thread and also a standard thread, in this case, it will calculate the pitch and depth that would correspond to the standard cylindrical thread and it will be assigned to that taper thread.

P-H.	Free-pitch thread.
M (S.I.)	Metric thread with normal pitch (international system).
M (S.I.F.)	Metric thread with fine pitch (international system).
B.S.W. (W)	Whitworth thread with normal pitch.
B.S.F.	Whitworth thread with fine pitch.
U.N.C.	Unified American thread with normal pitch.
U.N.F.	Unified American thread with fine pitch.
B.S.P.	Whitworth gas thread.
API	API thread (oil pipe).

- If P-H (free-pitch thread) is selected, the pitch and the depth of the thread are selected directly by the user.
- When choosing a standard thread, the user only selects the diameter of the metrics, the pitch and the depth of that thread are calculated automatically. The standard threads are single entry cylindrical threads.
- When choosing an API thread, the user selects the type of thread. The standard threads are single entry cylindrical threads.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

### Geometry of the thread (free-pitch threads).

On P-H threads (free-pitch threads), first one must choose between defining the pitch of the thread or the number of threads per inch.

- P Thread pitch in mm, inches or number of threads per inch. The thread pitch may be set along the taper of the thread or along its associated axis.
- $P > 0$ ; When programming the pitch along the taper of the thread.
  - $P < 0$ ; To program the pass along according to the associated axis.

H Total thread depth. The total depth must be programmed in radii and with positive value.



Constant-pitch thread.



Variable-pitch thread.

$\Delta P$  On variable-pitch threads, the increment or decrement of the thread pitch per spindle turn. If not programmed, the thread will have a constant pitch.

### Geometry of the thread (standard threads).

On standard threads, the pitch and the depth of the thread are calculated automatically.

$\phi$  Thread diameter (except in API threads).



Units used to enter the thread data (mm or inches).

## Spindle parameters.

### Spindle turning direction

The cycle lets select the spindle turning direction.



Spindle clockwise.



Spindle counterclockwise.

### Coolant status.



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

### Spindle range (gear).

GEAR Spindle gear; if not programmed, the cycle assumes a value of 0. If defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

## Machining parameters.

### Spindle speed.

S Spindle speed.

### Depth of pass.

$\Delta$  Maximum penetration step.

5.

CANNED CYCLES OF THE EDITOR. TAPPING.  
Multi-entry threading cycle.

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

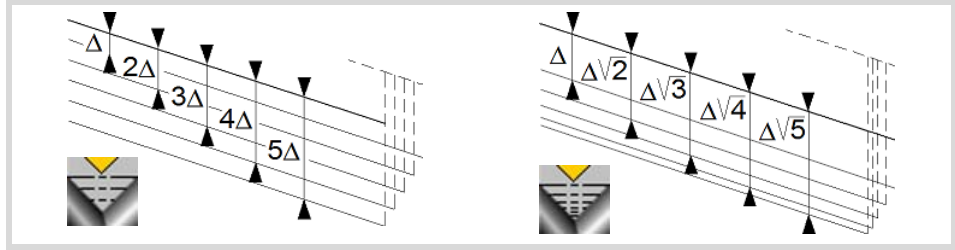
(REF: 1709)

# 5.

CANNED CYCLES OF THE EDITOR. TAPPING.  
Multi-entry threading cycle.



Constant depth of pass, with a smaller or same value as the maximum programmed.



The depth of each pass will be a function of the corresponding pass number. The cycle executes decreasing passes, the first one at the maximum penetration step until the minimum step is reached. If the penetrating increment (difference between penetrations) calculated by the CNC is smaller than the minimum decreasing penetrating step, the CNC assumes this latter value. The penetrations are calculated as follows  $\Delta, \Delta\sqrt{2}, \Delta\sqrt{3}, \Delta\sqrt{4}...$

$\delta$  Minimum penetration step.

**Repeat the last pass.**



The cycle repeats the last pass.



The cycle does not repeat the last pass.

**Normal thread or blind thread.**



Normal thread. Parameter  $\sigma$  indicates at what distance from the end of the thread the tool begins exiting it. During the withdrawal movement, the cycle keeps threading.



Blind thread. The cycle finishes the thread trying to maintain the pitch until the final point is reached and then it exits the thread. In this case, the exit from the thread is more abrupt than in the rest of the path blends (joints).

When programming  $\sigma=0$ , unlike a blind thread, the exit from the thread is smoother, but it misses some thread steps in the exit segment. The cycle gives priority to the dynamics of the machine over the geometry of the thread.

When programming blind thread, unlike programming  $\sigma=0$ , the exit from the thread is more abrupt, but the threaded segment is larger. The cycle gives priority to the geometry of the thread over the dynamics of the machine.

**Type of tool penetration.**



Radial penetration.



Penetration by starting flank. The CNC will ask for the cutter's penetration angle ( $\alpha$ ).



Penetration in zigzag, alternates between radial and starting flank. The CNC will ask for the cutter's penetration angle ( $\alpha$ ).



Penetration by final flank. The CNC will ask for the cutter's penetration angle ( $\alpha$ ).



Penetration in zigzag, alternates between radial and final flank. The CNC will ask for the cutter's penetration angle ( $\alpha$ ).

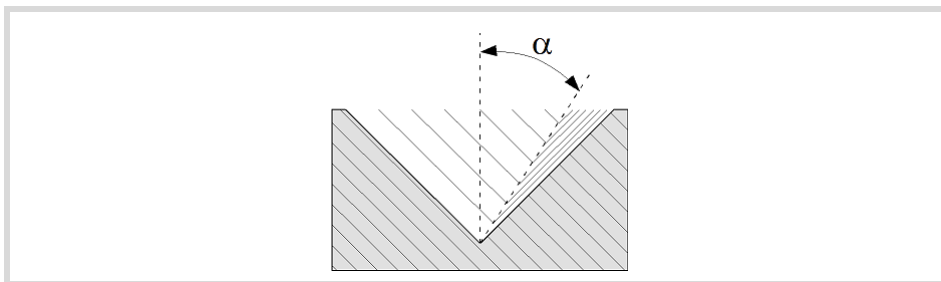


Penetration in zigzag, alternates between starting and final flank. The CNC will ask for the cutter's penetration angle ( $\alpha$ ) (leadscrew threading).

$\alpha$  Tool penetration angle.

We recommend to use a penetration angle smaller than half the tool angle, never larger. The thread cannot be machined if the penetration angle is larger than half the tool angle. If the

penetration angle is equal to half the tool angle, it will rub the flank of the thread at each cutting pass.



**Tool data.**

T Tool.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**



Activating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivating the execution of M functions before machining.

The editor will only show this option if the user has configured the editor to allow programming M functions.

5.

CANNED CYCLES OF THE EDITOR. TAPPING.

Multi-entry threading cycle.



FAGOR AUTOMATION

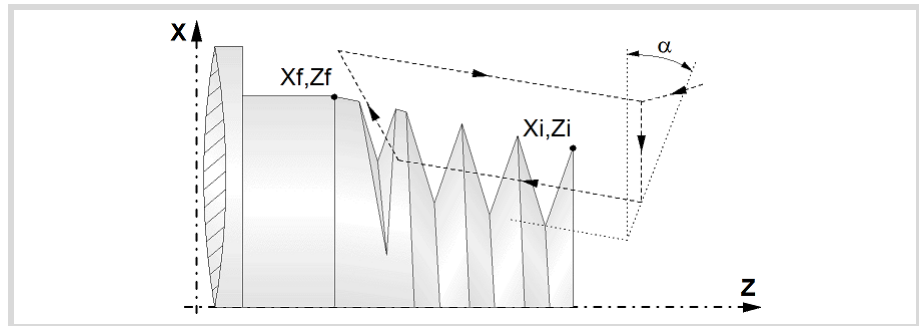
CNC 8060  
CNC 8065

(REF: 1709)

## 5.5.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 If the operation was programmed with another tool, the CNC will make a tool change moving to the change point if the machine so requires.
- 2 The spindle starts with the speed selected and in the direction stated. Depending on the spindle turning direction, the thread will be clockwise or counterclockwise.
- 3 The tool approaches in rapid up to the starting point, keeping the safety distance selected according to axes X and Z.
- 4 Tapping. It is done in successive passes until the full depth is reached. The depth of each pass depends on the selected type of penetration; depending on the corresponding pass number or keeping the increment between passes constant. Each of the threading passes is done in the following way:



First, rapid move to the corresponding programmed depth. This positioning is done according to the selected tool penetration angle ( $\alpha$ ). Then, threading of the programmed section up to the distance from the end of the thread (normal thread) or up to the final point (blind thread) and exit threading up to the final coordinate. And last, withdrawal in rapid to the approach point.

The electronic threading is carried out at 100% of the calculated feedrate and these values cannot be modified from the CNC's operator panel or via PLC. If the OEM allows it, (parameter THREADOVR), the user can modify the speed override from the operator panel and, in that case, the CNC will adapt the feedrate automatically respecting the thread pitch. In order to be able to modify the override, the active feed forward must be higher than 90%. It is recommended not to change the spindle speed override on threads with flank infeed.

Neither the feedrate override nor the spindle speed override can be changed in the last pass; this pass will be run with the override set in the previous pass.

- 5 Once the operation or cycle has ended, the tool will return to the safety point. When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
- 6 The CNC will not stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

# 5.

CANNED CYCLES OF THE EDITOR. TAPPING.  
Multi-entry threading cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

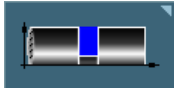
(REF: 1709)



# CANNED CYCLES OF THE EDITOR. GROOVING.

# 6

When pressing the softkey, the editor shows the cycle used last. When pressing the same softkey again, the menu shows all the cycles of the group.



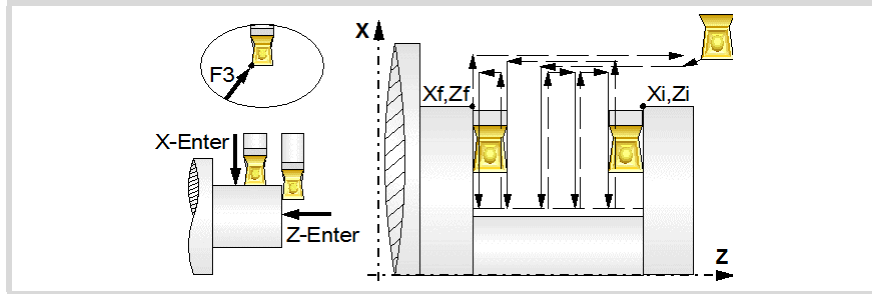
- Simple longitudinal grooving.
- Simple face grooving.
- Inclined (angled) longitudinal grooving.
- Inclined (angled) face grooving.
- Cut-off.

## 6.1 Calibrate the grooving tool (location code, shape).

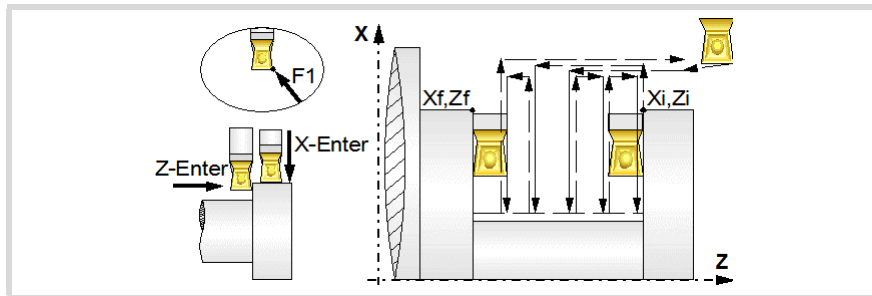
### 6.1.1 Outside longitudinal grooving.

When calibrating the grooving tool proper indication should be made of the location codes for the corner that has been calibrated. For this cycle, the same tool may be calibrated in six different ways, both for the outside and for the inside as shown next:

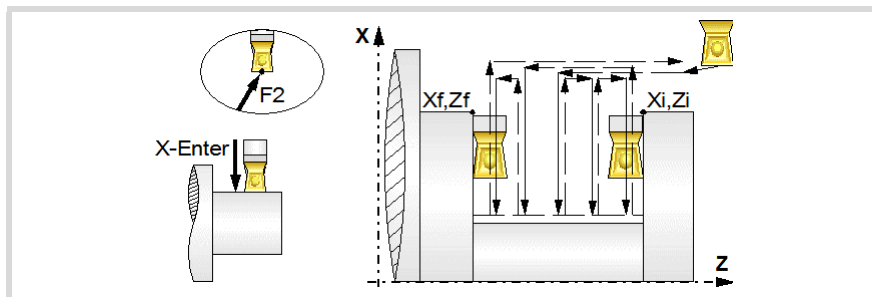
- The lower left-hand corner of the cutter is calibrated. Location code F3.



- The lower right-hand corner of the cutter is calibrated. Location code F1.



- Calibration is made only according to axis X, the CNC assumes the lower center of the cutter as the calibrated point. Location code F2.

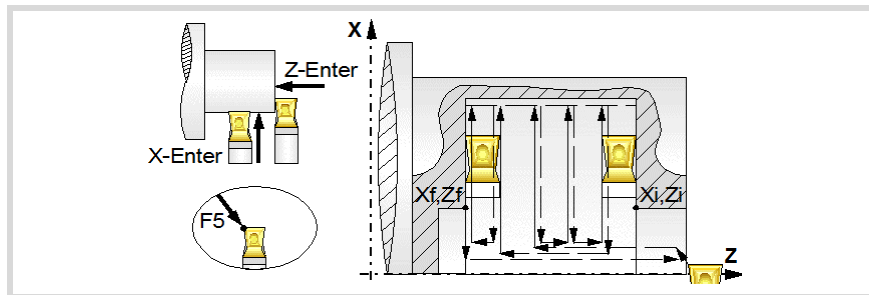


6.

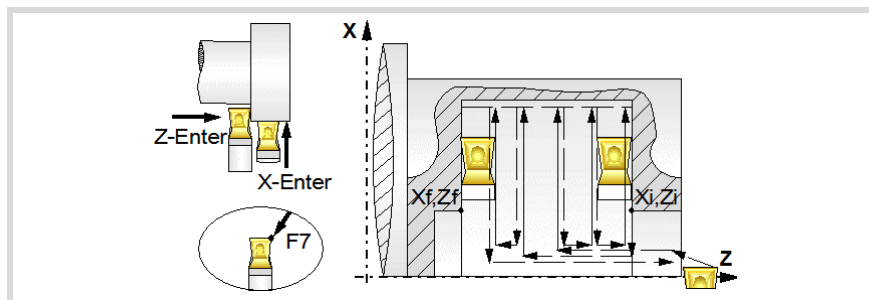
CANNED CYCLES OF THE EDITOR. GROOVING.  
Calibrate the grooving tool (location code, shape).

## 6.1.2 Inside longitudinal grooving.

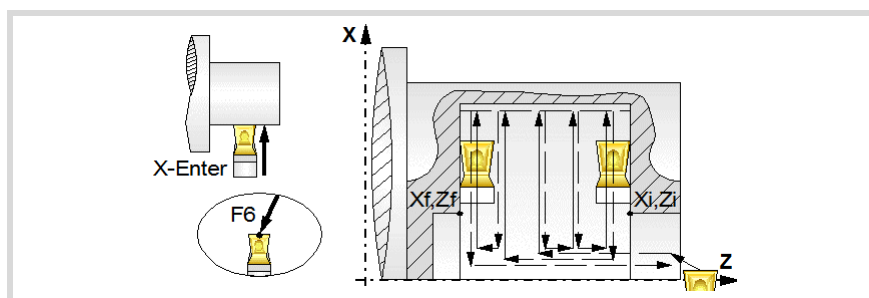
- The upper left-hand corner of the cutter is calibrated. Location code F5.



- The upper right-hand corner of the cutter is calibrated. Location code F7.



- Calibration is made only according to axis X, the CNC assumes the upper center of the cutter as the calibrated point. Location code F6.



# 6.

**CANNED CYCLES OF THE EDITOR. GROOVING.**  
 Calibrate the grooving tool (location code, shape).

**FAGOR** 

FAGOR AUTOMATION

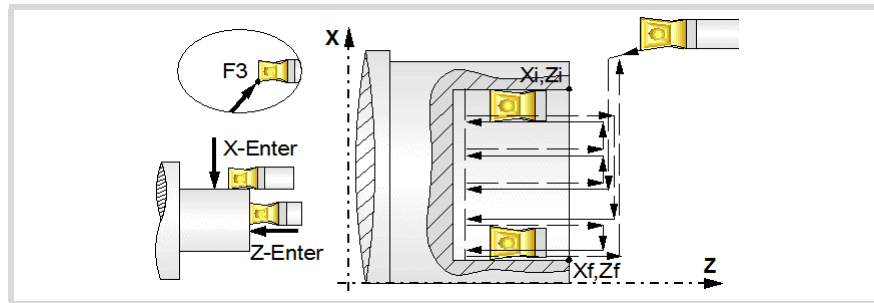
CNC 8060  
 CNC 8065

(REF: 1709)

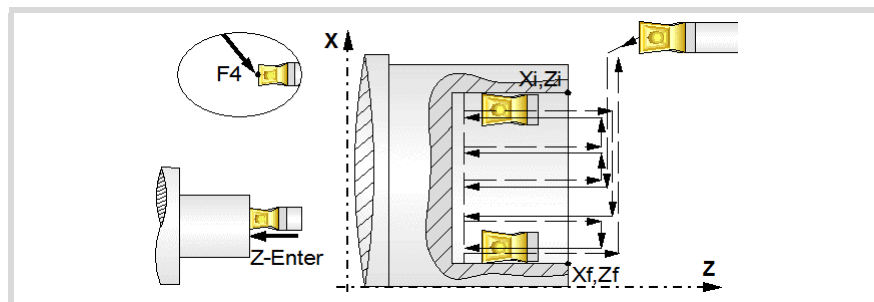
### 6.1.3 Face grooving.

When calibrating the grooving tool proper indication should be made of the location codes for the corner that has been calibrated. For this cycle, the same tool can be calibrated in three different ways, as shown below:

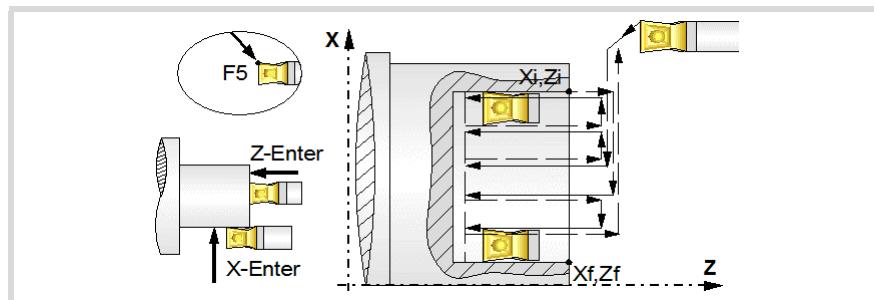
- The lower left-hand corner of the cutter is calibrated. Location code F3.



- Calibration is made only according to the Z axis the CNC assumes the left center of the cutter as the calibrated point. Location code F4.



- The upper left-hand corner of the cutter is calibrated. Location code F5.



**6.**  
**CANNED CYCLES OF THE EDITOR. GROOVING.**  
 Calibrate the grooving tool (location code, shape).

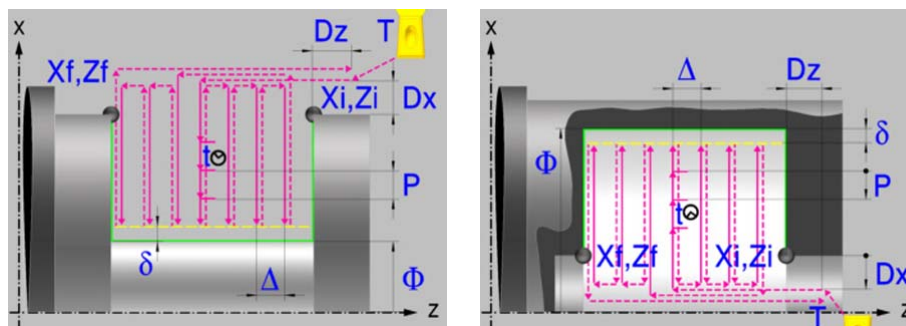


CNC 8060  
 CNC 8065

(REF: 1709)

## 6.2 Simple longitudinal grooving cycle.

The cycle allows outside and inside machining.



### Calibration of the grooving tool.

When calibrating the grooving tool proper indication should be made of the location codes for the corner that has been calibrated. For this cycle, the same tool may be calibrated in six different ways, both for outside machining and for inside machining. See "6.1 Calibrate the grooving tool (location code, shape)." on page 154.

### Geometric parameters.

#### Inside or outside machining.



Outside machining.



Inside machining.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Coordinates of the profile's starting and final points.

$X_i, Z_i$  Coordinates of the starting point

$X_f, Z_f$  Coordinates of the end point. By default,  $X_f$  assumes the value set for  $X_i$ .

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the approach point coordinate with respect to the starting point ( $X_i, Z_i$ ).

$D_x, D_z$  Safety distance.

6.

CANNED CYCLES OF THE EDITOR. GROOVING.  
Simple longitudinal grooving cycle.

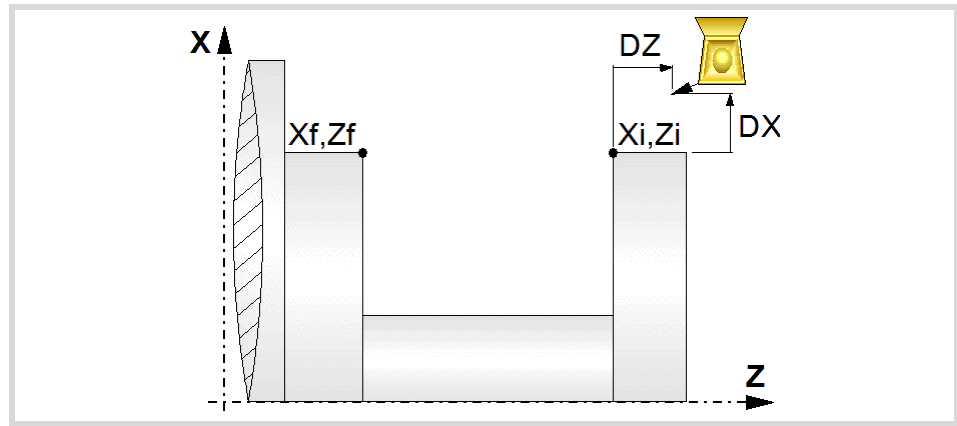
**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

The value of the safety distance on X is always defined in radius.



**Final diameter.**

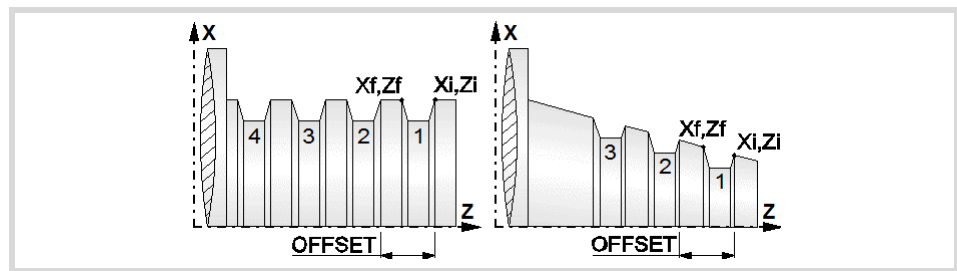
$\Phi$  Final diameter.

**Groove repetition.**

N Number of grooves. If is equal to "0" or "1", the cycle will only run one grooving operation.

I Distance between grooves.

With this data, the same groove may be repeated along the Z axis on longitudinal grooving or along the X axis on face grooving. If the initial groove is tapered (Xi different from Xf) that taper is maintained for the rest of the grooves.



**Spindle parameters.**

**Spindle turning direction**

The cycle lets select the spindle turning direction.



Spindle clockwise.



Spindle counterclockwise.

**Coolant status.**



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

6.

CANNED CYCLES OF THE EDITOR. GROOVING.  
Simple longitudinal grooving cycle.



CNC 8060  
CNC 8065

(REF: 1709)

**Spindle work mode; rpm or Constant Surface Speed.**



Constant turning speed.



Constant surface speed.

**Maximum spindle speed.**

Smax Maximum spindle turning speed; if not programmed, the cycle will maintain the maximum active speed.

**Spindle range (gear).**

GEAR Spindle gear; if not programmed, the cycle assumes a value of 0. If defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Machining parameters. Roughing.**

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the roughing operation.



Do not run the roughing operation.

**Type of penetration.**



Penetration without chip removal.



Penetration with chip removal.



Zig-zag penetration with chip removal.

Two more parameters must be defined when the cycle runs the grooving operation with chip removal.

P Penetration step.

t Dwell for chip removal.

**Maximum machining pass.**

$\Delta$  Maximum machining pass.

- If  $\Delta$  is positive, the cycle recalculates the step so all the passes are identical with the same value as or smaller than the one programmed.
- If  $\Delta$  is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

**Machining feedrate.**

F Machining feedrate.

**Spindle speed.**

S Spindle turning speed (rpm) or cutting speed.

**Tool data.**

T Tool. If it is set to 0, the cycle does not execute the roughing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

6.

CANNED CYCLES OF THE EDITOR. GROOVING.

Simple longitudinal grooving cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 6.

CANNED CYCLES OF THE EDITOR. GROOVING.  
Simple longitudinal grooving cycle.

**Machining M functions.**

Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**Machining parameters. Finishing.**

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the finishing operation.



Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes ( $\delta x$ ,  $\delta z$ ).

**Finishing stock.**

$\delta$  Finishing stock at the bottom.

**Machining feedrate.**

F Machining feedrate.

**Spindle speed.**

S Spindle turning speed (rpm) or cutting speed.

**Tool data.**

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)



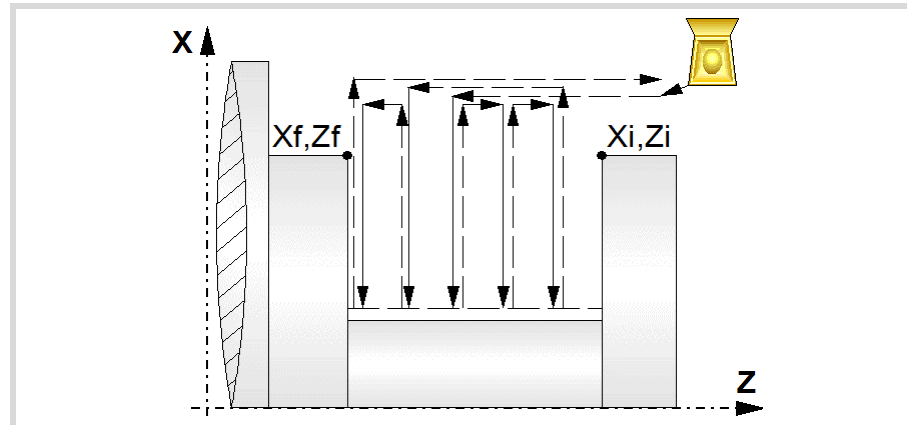
## 6.2.1 Basic operation

The machining steps in this cycle are as follows:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The spindle starts with the speed selected and in the direction stated.
- 3 The tool approaches the starting point ( $X_i, Z_i$ ) in rapid, keeping the selected safety distance ( $D_x, D_z$ ) according to axes X and Z.
- 4 Roughing operation (if it has been programmed).

The cycle runs successive passes until reaching the bottom of the groove plus the finishing stocks. This operation is done with the conditions set for the roughing operation.

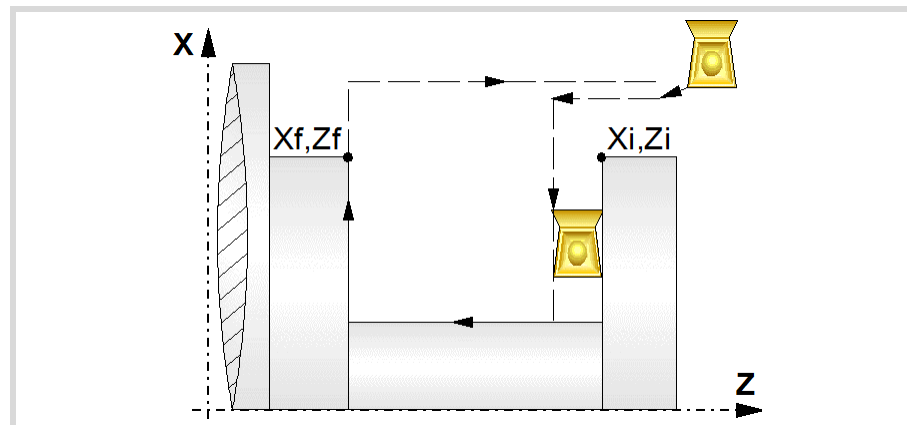
The first penetration pass may be done in two ways; either continuously or in intervals depending on the programmed pass and with a dwell at the bottom of each pass.



- 5 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle runs a single pass to remove the finishing stock. This operation is done with the conditions set for the finishing operation.



- 6 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
- 7 The CNC will not stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

# 6.

**CANNED CYCLES OF THE EDITOR. GROOVING.**  
Simple longitudinal grooving cycle.

**FAGOR** 

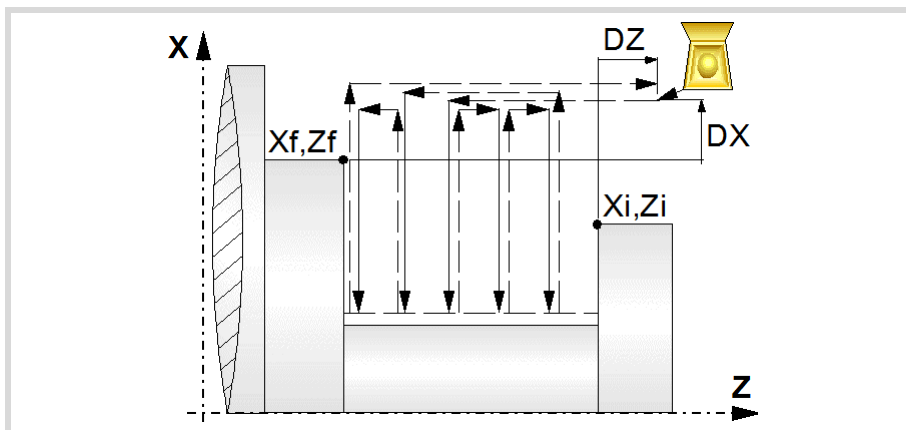
FAGOR AUTOMATION

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

(REF: 1709)

## Considerations.

- If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.
- If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the finishing operation, the tool will move to the safety point (starting point plus the safety distance).
- When the surface to be machined is not fully cylindrical, the CNC analyzes the coordinates on X of the starting and end points and takes as starting point on X the coordinate farthest from the final diameter.



# 6.

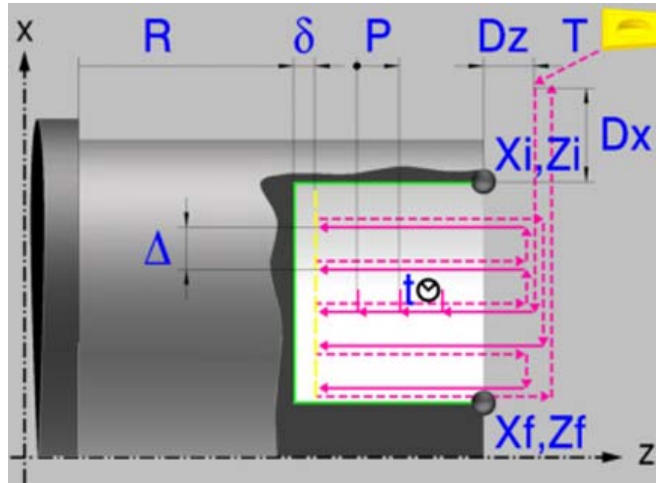
**CANNED CYCLES OF THE EDITOR. GROOVING.**  
Simple longitudinal grooving cycle.

**FAGOR**   
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

### 6.3 Simple face grooving cycle



#### Calibration of the grooving tool.

When calibrating the grooving tool proper indication should be made of the location codes for the corner that has been calibrated. For this cycle, the same tool may be calibrated in three different ways, both for outside machining and for inside machining See "6.1 Calibrate the grooving tool (location code, shape)." on page 154.

#### Geometric parameters.

##### Coordinates of the profile's starting and final points.

$X_i, Z_i$  Coordinates of the starting point

$X_f, Z_f$  Coordinates of the end point. By default,  $Z_f$  assumes the value set for  $Z_i$ .

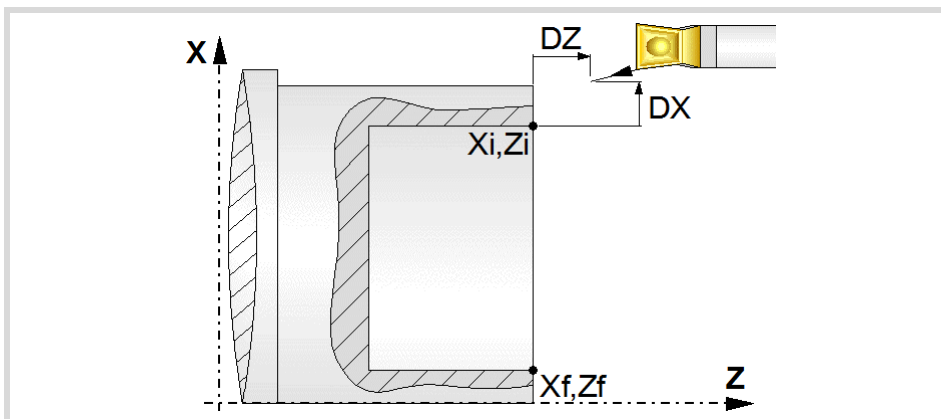
To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

##### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the approach point coordinate with respect to the starting point ( $X_i, Z_i$ ).

$D_x, D_z$  Safety distance.

The value of the safety distance on X is always defined in radius.



# 6.

CANNED CYCLES OF THE EDITOR. GROOVING.  
Simple face grooving cycle



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Coordinate of the bottom of the groove.**

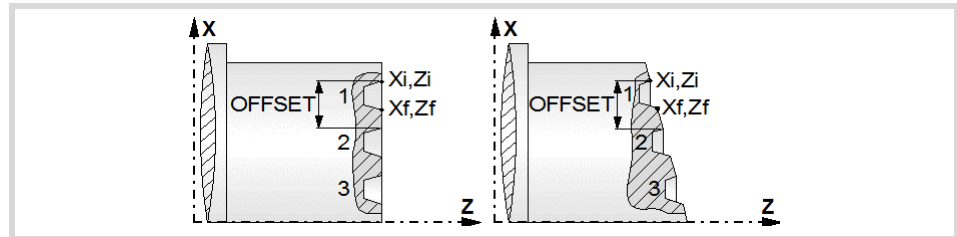
R Coordinate of the bottom of the groove

**Groove repetition.**

N Number of grooves. If is equal to "0" or "1", the cycle will only run one grooving operation.

I Distance between grooves.

With this data, the same groove may be repeated along the Z axis on longitudinal grooving or along the X axis on face grooving. If the initial groove is tapered ( $X_i$  different from  $X_f$ ) that taper is maintained for the rest of the grooves.



**Spindle parameters.**

**Spindle turning direction**

The cycle lets select the spindle turning direction.



Spindle clockwise.



Spindle counterclockwise.

**Coolant status.**



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

**Spindle work mode; rpm or Constant Surface Speed.**



Constant turning speed.



Constant surface speed.

**Maximum spindle speed.**

Smax Maximum spindle turning speed; if not programmed, the cycle will maintain the maximum active speed.

**Spindle range (gear).**

GEAR Spindle gear; if not programmed, the cycle assumes a value of 0. If defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**6.**

CANNED CYCLES OF THE EDITOR. GROOVING.  
Simple face grooving cycle



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

### Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the roughing operation.



Do not run the roughing operation.

#### Type of penetration.



Penetration without chip removal.



Penetration with chip removal.



Zig-zag penetration with chip removal.

Two more parameters must be defined when the cycle runs the grooving operation with chip removal.

P Penetration step.

t Dwell for chip removal.

#### Maximum machining pass.

$\Delta$  Maximum machining pass.

- If  $\Delta$  is positive, the cycle recalculates the step so all the passes are identical with the same value as or smaller than the one programmed.
- If  $\Delta$  is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

#### Machining feedrate.

F Machining feedrate.

#### Spindle speed.

S Spindle turning speed (rpm) or cutting speed.

#### Tool data.

T Tool. If it is set to 0, the cycle does not execute the roughing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

#### Machining M functions.



Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

### Machining parameters. Finishing.

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the finishing operation.



Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes ( $\delta x$ ,  $\delta z$ ).

# 6.

CANNED CYCLES OF THE EDITOR. GROOVING.

Simple face grooving cycle

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Finishing stock.**

δ Finishing stock at the bottom.

**Machining feedrate.**

F Machining feedrate.

**Spindle speed.**

S Spindle turning speed (rpm) or cutting speed.

**Tool data.**

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

6.

CANNED CYCLES OF THE EDITOR. GROOVING.  
Simple face grooving cycle

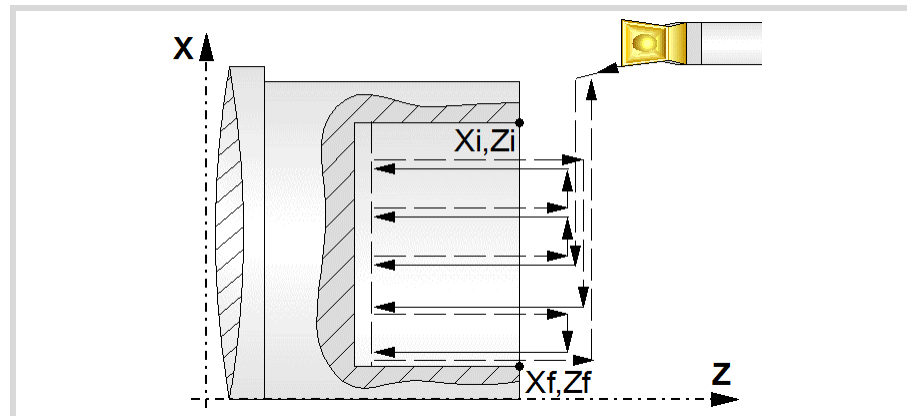
### 6.3.1 Basic operation

The machining steps in this cycle are as follows:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The spindle starts with the speed selected and in the direction stated.
- 3 The tool approaches the starting point ( $X_i, Z_i$ ) in rapid, keeping the selected safety distance ( $D_x, D_z$ ) according to axes X and Z.
- 4 Roughing operation (if it has been programmed).

The cycle runs successive passes until reaching the bottom of the groove plus the finishing stocks. This operation is done with the conditions set for the roughing operation.

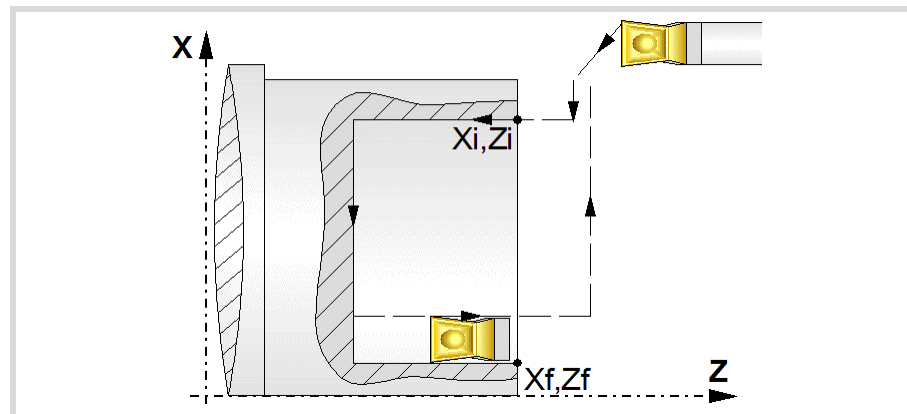
The first penetration pass may be done in two ways; either continuously or in intervals depending on the programmed pass and with a dwell at the bottom of each pass.



- 5 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle runs a single pass to remove the finishing stock. This operation is done with the conditions set for the finishing operation.



- 6 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
- 7 The CNC will not stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

# 6.

CANNED CYCLES OF THE EDITOR. GROOVING.  
Simple face grooving cycle

**FAGOR** 

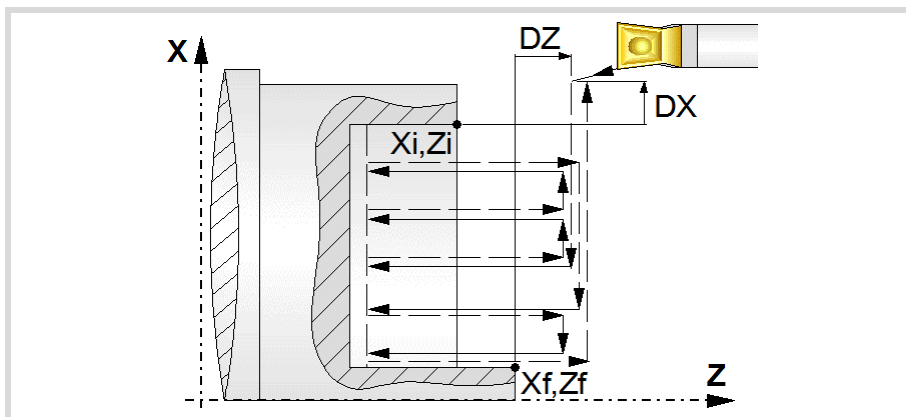
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## Considerations.

- If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.
- If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the finishing operation, the tool will move to the safety point (starting point plus the safety distance).
- When the surface to be machined is not fully cylindrical, the CNC analyzes the coordinates on X of the starting and end points and takes as starting point on X the coordinate farthest from the final diameter.



# 6.

**CANNED CYCLES OF THE EDITOR. GROOVING.**  
Simple face grooving cycle

**FAGOR**   
FAGOR AUTOMATION

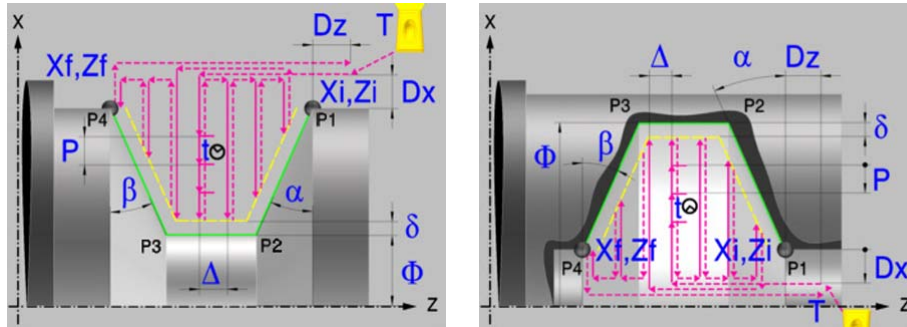
CNC 8060  
CNC 8065

(REF: 1709)



## 6.4 Inclined longitudinal grooving cycle.

The cycle allows outside and inside machining.



### Calibration of the grooving tool.

When calibrating the grooving tool proper indication should be made of the location codes for the corner that has been calibrated. For this cycle, the same tool may be calibrated in six different ways, both for outside machining and for inside machining. See ["6.1 Calibrate the grooving tool \(location code, shape\)."](#) on page 154.

### Geometric parameters.

#### Inside or outside machining.



Outside machining.



Inside machining.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Coordinates of the profile's starting and final points.

$X_i, Z_i$  Coordinates of the starting point

$X_f, Z_f$  Coordinates of the end point. By default,  $X_f$  assumes the value set for  $X_i$ .

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the approach point coordinate with respect to the starting point ( $X_i, Z_i$ ).

$D_x, D_z$  Safety distance.

6.

CANNED CYCLES OF THE EDITOR. GROOVING.  
Inclined longitudinal grooving cycle.

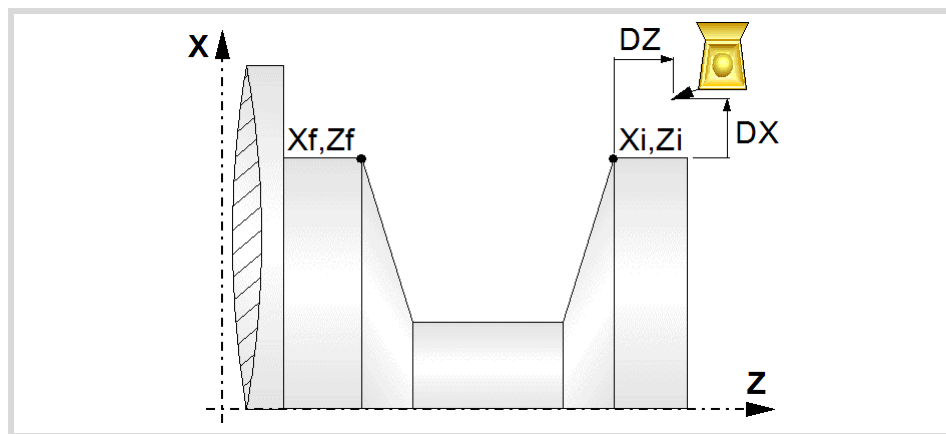
**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

The value of the safety distance on X is always defined in radius.



**Final diameter.**

$\Phi$  Final diameter.

**Type of machining at each corner.**

The type of machining must be defined for all four corners of the groove.



Square corner.

Rounding. The rounding radius must be defined.

Chamfer. The size of the chamfer must be defined.

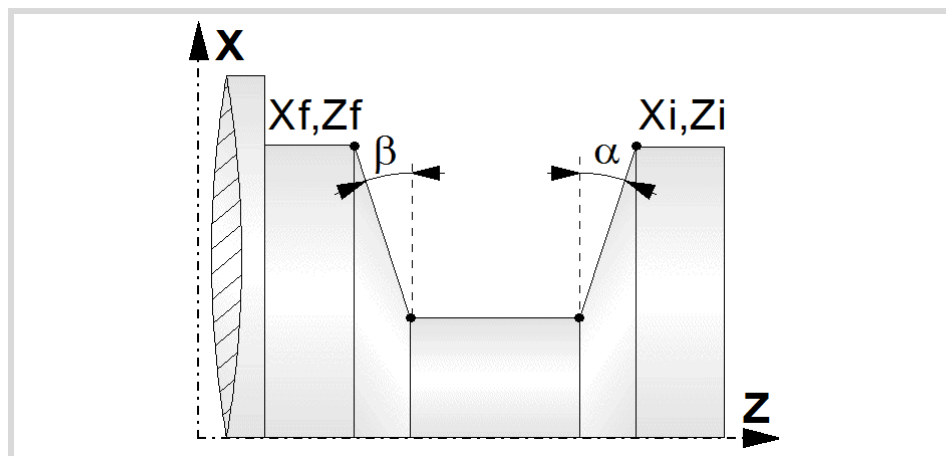
$r$  Rounding radius of the pocket corners.

$c$  Size of the chamfer of the pocket corners.

**Inclination angles.**

$\alpha$  Inclination angle of the starting wall.

$\beta$  Inclination angle of the final wall.



**Groove repetition.**

$N$  Number of grooves. If is equal to "0" or "1", the cycle will only run one grooving operation.

$l$  Distance between grooves.

6.

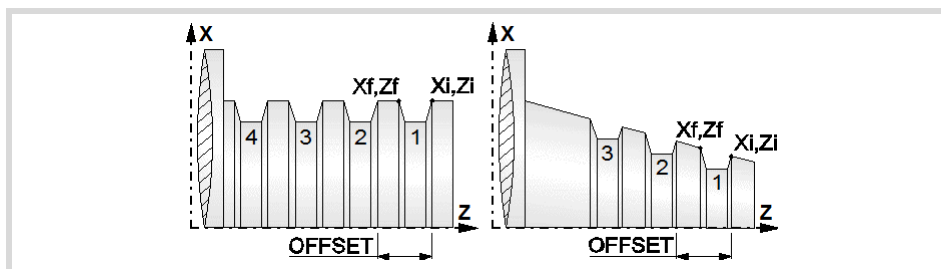
**CANNED CYCLES OF THE EDITOR. GROOVING.**  
Inclined longitudinal grooving cycle.



CNC 8060  
CNC 8065

(REF: 1709)

With this data, the same groove may be repeated along the Z axis on longitudinal grooving or along the X axis on face grooving. If the initial groove is tapered ( $X_i$  different from  $X_f$ ) that taper is maintained for the rest of the grooves.



## Spindle parameters.

### Spindle turning direction

The cycle lets select the spindle turning direction.



Spindle clockwise.



Spindle counterclockwise.

### Coolant status.



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

### Spindle work mode; rpm or Constant Surface Speed.



Constant turning speed.



Constant surface speed.

### Maximum spindle speed.

**Smax** Maximum spindle turning speed; if not programmed, the cycle will maintain the maximum active speed.

### Spindle range (gear).

**GEAR** Spindle gear; if not programmed, the cycle assumes a value of 0. If defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

## Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the roughing operation.



Do not run the roughing operation.

6.

CANNED CYCLES OF THE EDITOR. GROOVING.  
Inclined longitudinal grooving cycle.

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 6.

CANNED CYCLES OF THE EDITOR. GROOVING.  
Inclined longitudinal grooving cycle.

**Type of penetration.**

Penetration without chip removal.



Penetration with chip removal.



Zig-zag penetration with chip removal.

Two more parameters must be defined when the cycle runs the grooving operation with chip removal.

- P Penetration step.  
t Dwell for chip removal.

**Maximum machining pass.**

- $\Delta$  Maximum machining pass.
- If  $\Delta$  is positive, the cycle recalculates the step so all the passes are identical with the same value as or smaller than the one programmed.
  - If  $\Delta$  is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

**Machining feedrate.**

- F Machining feedrate.

**Spindle speed.**

- S Spindle turning speed (rpm) or cutting speed.

**Tool data.**

- T Tool. If it is set to 0, the cycle does not execute the roughing operation.  
D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**Machining parameters. Finishing.**

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the finishing operation.



Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes ( $\delta x$ ,  $\delta z$ ).



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Finishing stock.**

$\delta$  Finishing stock.

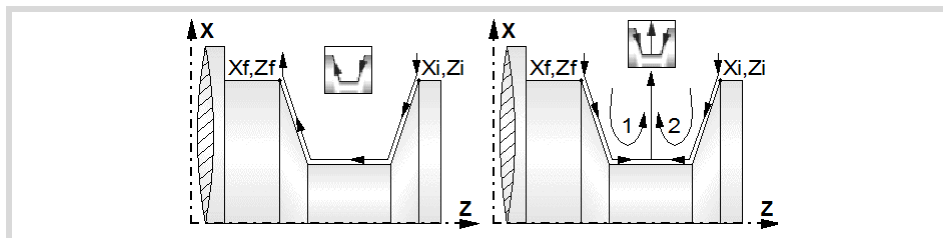
**Type of finishing pass.**



Groove finishing following the profile.



Descending finishing of the groove.



**Machining feedrate.**

F Machining feedrate.

**Spindle speed.**

S Spindle turning speed (rpm) or cutting speed.

**Tool data.**

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**



Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**6.**

**CANNED CYCLES OF THE EDITOR. GROOVING.**

Inclined longitudinal grooving cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

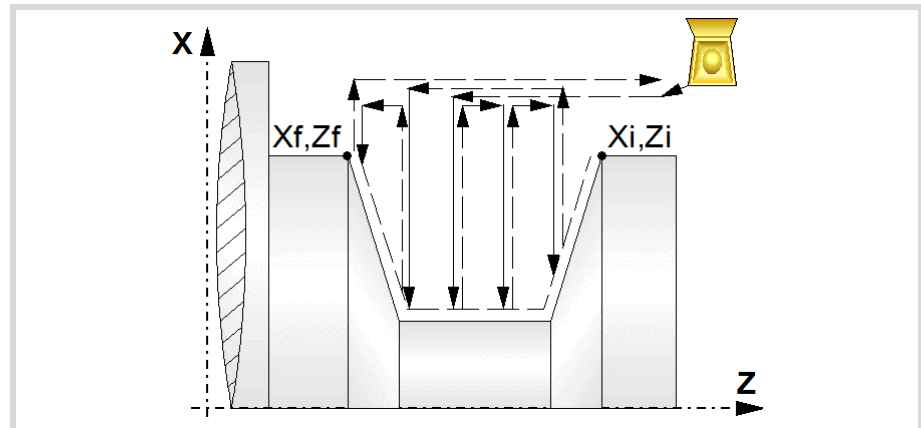
## 6.4.1 Basic operation

The machining steps in this cycle are as follows:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The spindle starts with the speed selected and in the direction stated.
- 3 The tool approaches the starting point ( $X_i, Z_i$ ) in rapid, keeping the selected safety distance ( $D_x, D_z$ ) according to axes X and Z.
- 4 Roughing operation (if it has been programmed).

The cycle runs successive passes until reaching the bottom of the groove plus the finishing stocks. This operation is done with the conditions set for the roughing operation. If no finishing stock is programmed, the cycle applies the dwell in all roughing passes.

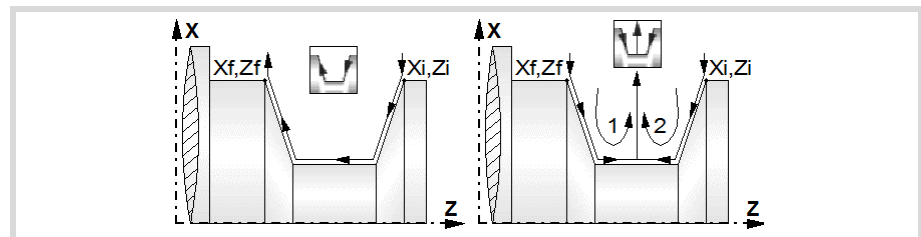
The first penetration pass may be done in two ways; either continuously or in intervals depending on the programmed pass and with a dwell at the bottom of each pass.



- 5 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle runs a single pass to remove the finishing stock. This operation is done with the conditions set for the finishing operation.



- 6 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
- 7 The CNC will not stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

# 6.

CANNED CYCLES OF THE EDITOR. GROOVING.  
Inclined longitudinal grooving cycle.



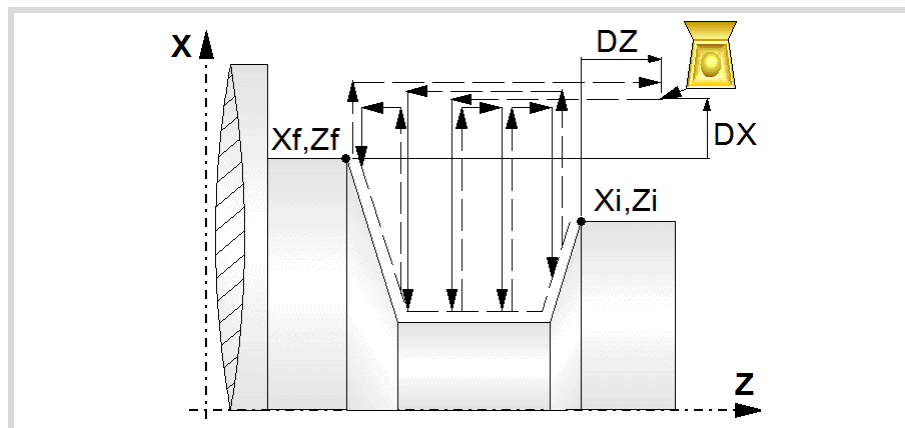
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## Considerations.

- If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.
- If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the finishing operation, the tool will move to the safety point (starting point plus the safety distance).
- When the surface to be machined is not fully cylindrical, the CNC analyzes the coordinates on X of the starting and end points and takes as starting point on X the coordinate farthest from the final diameter.



- The roughing operation is carried out in G05, the rounding radius may be modified using the instruction #ROUNDPAR. If #ROUNDPAR is not programmed, it assumes the default rounding radius set by machine parameters.
- The finishing operation is carried out in G07.

6.

**CANNED CYCLES OF THE EDITOR. GROOVING.**

Inclined longitudinal grooving cycle.

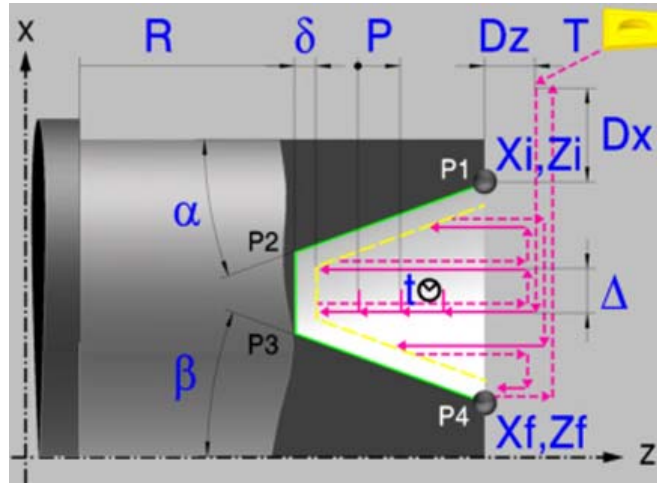
**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 6.5 Inclined face grooving cycle.



### Calibration of the grooving tool.

When calibrating the grooving tool proper indication should be made of the location codes for the corner that has been calibrated. For this cycle, the same tool may be calibrated in three different ways, both for outside machining and for inside machining See "6.1 Calibrate the grooving tool (location code, shape)." on page 154.

### Geometric parameters.

#### Coordinates of the profile's starting and final points.

$X_i, Z_i$  Coordinates of the starting point

$X_f, Z_f$  Coordinates of the end point. By default,  $Z_f$  assumes the value set for  $Z_i$ .

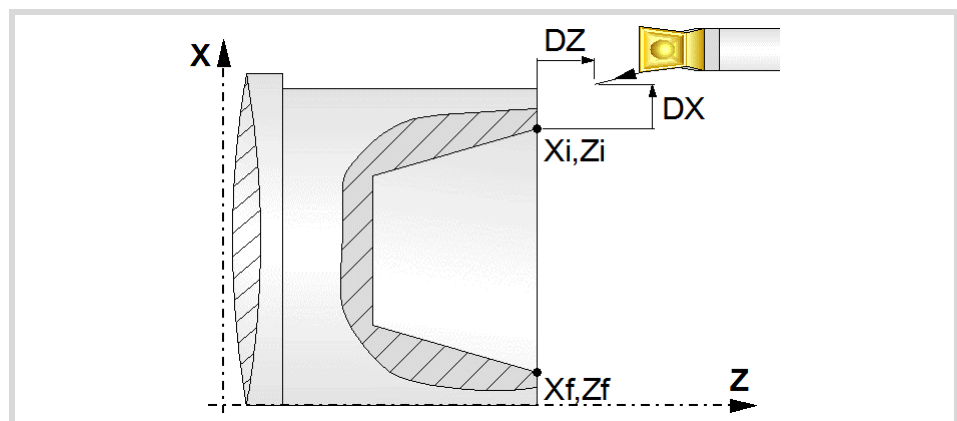
To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the approach point coordinate with respect to the starting point ( $X_i, Z_i$ ).

$D_x, D_z$  Safety distance.

The value of the safety distance on X is always defined in radius.



6.

CANNED CYCLES OF THE EDITOR. GROOVING.  
Inclined face grooving cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)



**Coordinate of the bottom of the groove.**

R Coordinate of the bottom of the groove

**Type of machining at each corner.**

The type of machining must be defined for all four corners of the groove.



Square corner.



Rounding. The rounding radius must be defined.



Chamfer. The size of the chamfer must be defined.

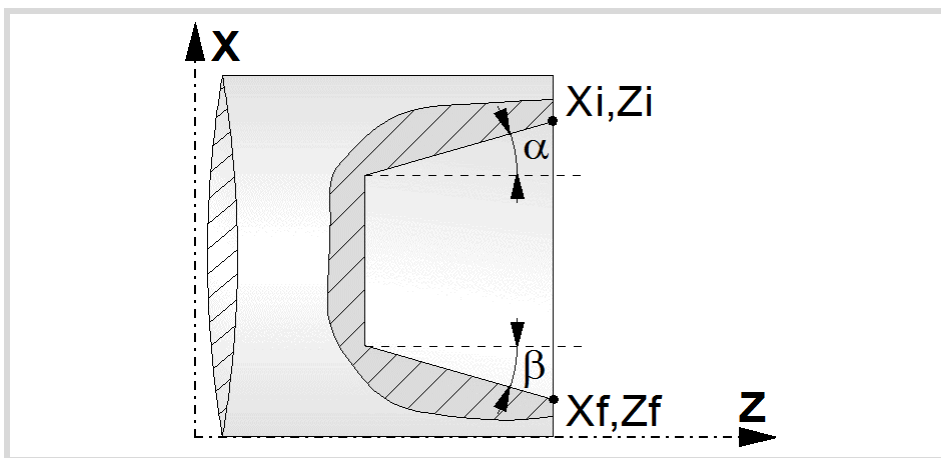
r Rounding radius of the pocket corners.

c Size of the chamfer of the pocket corners.

**Inclination angles.**

$\alpha$  Inclination angle of the starting wall.

$\beta$  Inclination angle of the final wall.

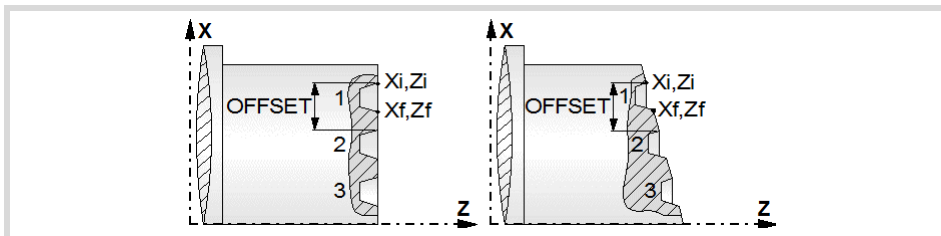


**Groove repetition.**

N Number of grooves. If is equal to "0" or "1", the cycle will only run one grooving operation.

I Distance between grooves.

With this data, the same groove may be repeated along the Z axis on longitudinal grooving or along the X axis on face grooving. If the initial groove is tapered ( $X_i$  different from  $X_f$ ) that taper is maintained for the rest of the grooves.



**Spindle parameters.**

**Spindle turning direction**

The cycle lets select the spindle turning direction.



Spindle clockwise.



Spindle counterclockwise.

**6.**

**CANNED CYCLES OF THE EDITOR. GROOVING.**  
Inclined face grooving cycle.



CNC 8060  
CNC 8065

(REF: 1709)

**Coolant status.**

Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

**Spindle work mode; rpm or Constant Surface Speed.**

Constant turning speed.



Constant surface speed.

**Maximum spindle speed.**

Smax Maximum spindle turning speed; if not programmed, the cycle will maintain the maximum active speed.

**Spindle range (gear).**

GEAR Spindle gear; if not programmed, the cycle assumes a value of 0. If defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Machining parameters. Roughing.**

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the roughing operation.



Do not run the roughing operation.

**Type of penetration.**

Penetration without chip removal.



Penetration with chip removal.



Zig-zag penetration with chip removal.

Two more parameters must be defined when the cycle runs the grooving operation with chip removal.

P Penetration step.

t Dwell for chip removal.

6.

CANNED CYCLES OF THE EDITOR. GROOVING.  
Inclined face grooving cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Maximum machining pass.**

- $\Delta$  Maximum machining pass.
- If  $\Delta$  is positive, the cycle recalculates the step so all the passes are identical with the same value as or smaller than the one programmed.
  - If  $\Delta$  is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

**Machining feedrate.**

F Machining feedrate.


**Spindle speed.**


S Spindle turning speed (rpm) or cutting speed.

**Tool data.**

- T Tool. If it is set to 0, the cycle does not execute the roughing operation.
- D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**


 Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.


 Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**Machining parameters. Finishing.**

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.


 Run the finishing operation.


 Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes ( $\delta x$ ,  $\delta z$ ).

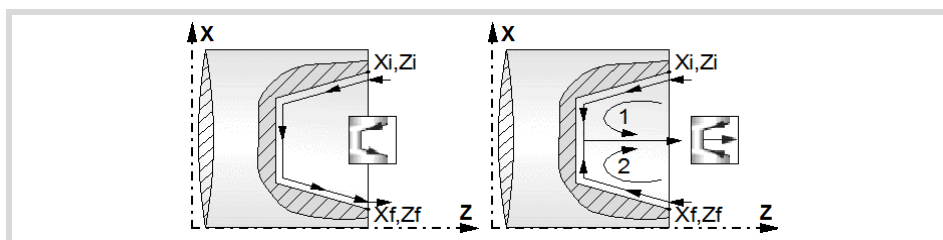
**Finishing stock.**

$\delta$  Finishing stock.

**Type of finishing pass.**

 Groove finishing following the profile.

 Descending finishing of the groove.



**Machining feedrate.**

F Machining feedrate.

**Spindle speed.**

S Spindle turning speed (rpm) or cutting speed.

**Tool data.**

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

6.

**CANNED CYCLES OF THE EDITOR. GROOVING.**  
Inclined face grooving cycle.

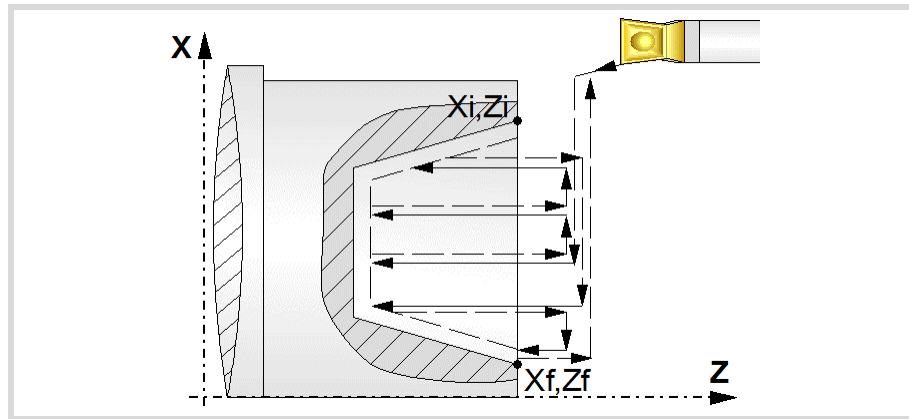
### 6.5.1 Basic operation

The machining steps in this cycle are as follows:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The spindle starts with the speed selected and in the direction stated.
- 3 The tool approaches the starting point ( $X_i, Z_i$ ) in rapid, keeping the selected safety distance ( $D_x, D_z$ ) according to axes X and Z.
- 4 Roughing operation (if it has been programmed).

The cycle runs successive passes until reaching the bottom of the groove plus the finishing stocks. This operation is done with the conditions set for the roughing operation. If no finishing stock is programmed, the cycle applies the dwell in all roughing passes.

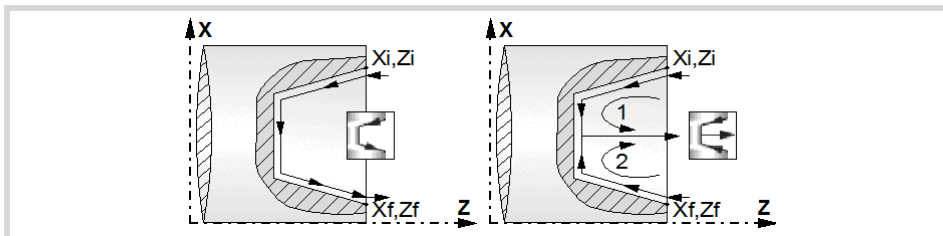
The first penetration pass may be done in two ways; either continuously or in intervals depending on the programmed pass and with a dwell at the bottom of each pass.



- 5 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle runs a single pass to remove the finishing stock. This operation is done with the conditions set for the finishing operation.



- 6 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
- 7 The CNC will not stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

# 6.

CANNED CYCLES OF THE EDITOR. GROOVING.  
Inclined face grooving cycle.



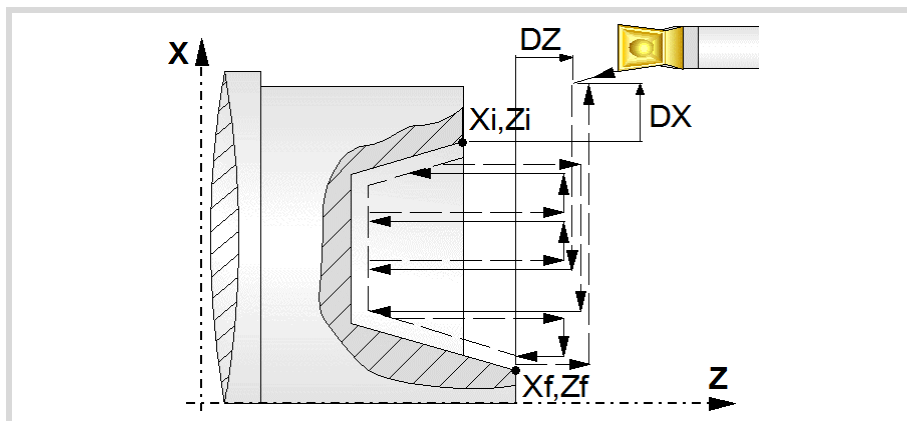
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## Considerations.

- If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.
- If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the finishing operation, the tool will move to the safety point (starting point plus the safety distance).
- When the surface to be machined is not fully cylindrical, the CNC analyzes the coordinates on X of the starting and end points and takes as starting point on X the coordinate farthest from the final diameter.



- The roughing operation is carried out in G05, the rounding radius may be modified using the instruction #ROUNDPAR. If #ROUNDPAR is not programmed, it assumes the default rounding radius set by machine parameters.
- The finishing operation is carried out in G07.

6.

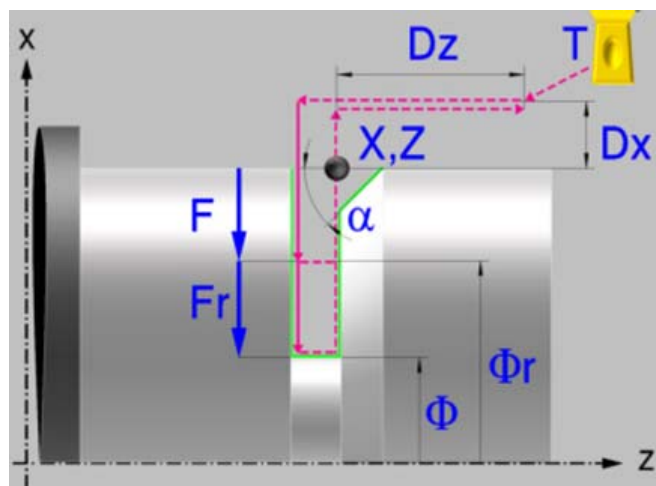
CANNED CYCLES OF THE EDITOR. GROOVING.  
Inclined face grooving cycle.

**FAGOR**   
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 6.6 Cut-off cycle.



6.

CANNED CYCLES OF THE EDITOR. GROOVING.

Cut-off cycle.

### Calibration of the grooving tool.

When calibrating the grooving tool proper indication should be made of the location codes for the corner that has been calibrated. For this cycle, the same tool may be calibrated in six different ways, both for outside machining and for inside machining See "6.1 Calibrate the grooving tool (location code, shape)." on page 154.

### Geometric parameters.

#### Coordinates of the profile's starting and final points.

$X_i, Z_i$  Coordinates of the starting point

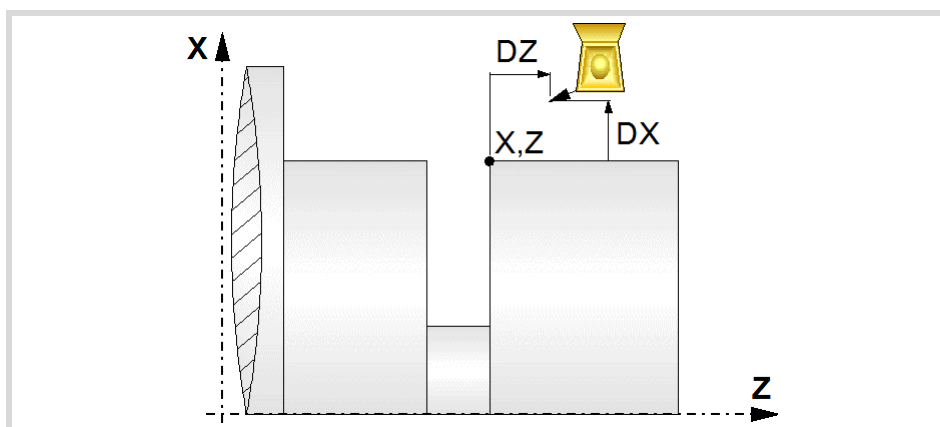
To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the approach point coordinate with respect to the starting point ( $X_i, Z_i$ ).

$D_x, D_z$  Safety distance.

The value of the safety distance on X is always defined in radius.



**FAGOR**

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 6.

CANNED CYCLES OF THE EDITOR. GROOVING. Cut-off cycle.

**Final diameter.**

$\Phi$  Final diameter.

**Type of machining at each corner.**

The type of machining must be defined for all the corner of the groove.



Square corner.



Rounding. The rounding radius must be defined.



Chamfer. The size of the chamfer must be defined.

r Rounding radius of the pocket corners.

c Size of the chamfer of the pocket corners.

**Spindle parameters.****Spindle turning direction**

The cycle lets select the spindle turning direction.



Spindle clockwise.



Spindle counterclockwise.

**Coolant status.**

Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

**Spindle work mode; rpm or Constant Surface Speed.**

Constant turning speed.



Constant surface speed.

**Maximum spindle speed.**

Smax Maximum spindle turning speed; if not programmed, the cycle will maintain the maximum active speed.

**Spindle range (gear).**

GEAR Spindle gear; if not programmed, the cycle assumes a value of 0. If defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)



## Machining parameters.

### Slower feedrate at the end of the machining operation.

It is possible to program two different feedrates for the same cut-off operation. From a particular diameter ( $\Phi r$ ) on, the feedrate (F) slows down gradually to reach the final diameter at the slow feedrate.

- Fr        Slower feedrate at the end of the machining operation.  
 $\Phi r$       Diameter to begin lowering the feedrate.

### Machining feedrate.

- F        Machining feedrate.

### Spindle speed.

- S        Spindle turning speed (rpm) or cutting speed.

### Tool data.

- T        Tool. If it is set to 0, the cycle does not execute the roughing operation.  
 D        Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

### Machining M functions.



Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

6.

CANNED CYCLES OF THE EDITOR. GROOVING.

Cut-off cycle.

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 6.6.1 Basic operation

The machining steps in this cycle are as follows:

- 1 If the cut-off operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.
- 2 The spindle starts with the speed selected and in the direction stated.
- 3 The tool approaches the starting point ( $X_i$ ,  $Z_i$ ) in rapid, keeping the selected safety distance ( $D_x$ ,  $D_z$ ) according to axes X and Z.
- 4 If there is a chamfer or rounding, the cycle runs the first grooving operation until matching the depth of the chamfer or rounding. A second machining operation makes the chamfer or rounding and the rest of the groove up to a diameter  $\Phi$ .
- 5 The cycle begins machining at a feedrate F until reaching the diameter  $\Phi_r$ , from then on, it slows down until the final diameter is reached at a feedrate  $F_r$ .
- 6 The part finishing is done with the set condition; axis feedrate (F), slower feedrate ( $F_r$ ), spindle speed (S), turning direction.
- 7 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance).
- 8 The CNC will not stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

# 6.

CANNED CYCLES OF THE EDITOR. GROOVING.

Cut-off cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

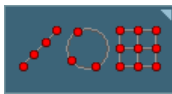
# CANNED CYCLES OF THE EDITOR. PROFILES.

# 7

When pressing the softkey, the editor shows the cycle used last. When pressing the same softkey again, the menu shows all the cycles of the group.



- Point-to-point turning.
- Profile turning.
- Profile in the ZC/YZ plane.
- Rectangular pocket in the ZC/YZ plane.
- Circular pocket in the ZC/YZ plane.
- Circular pre-empted pocket in the ZC/YZ plane.
- 2D profile pocket in the ZC/YZ plane.
- Profile in the XC/XY plane.
- Rectangular pocket in the XC/XY plane.
- Circular pocket in the XC/XY plane.
- Circular pre-empted pocket in the XC/XY plane.
- 2D profile pocket in the XC/XY plane.



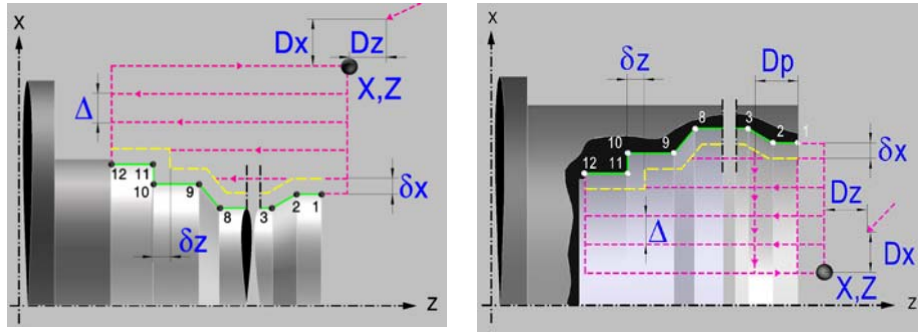
Multiple machining may be added to pocket cycles so the cycle is repeated in several points. The available multiple machining operations are the following. See chapter "[10 Canned cycles of the editor. Multiple positioning.](#)"

- Points in line.
- Points in arc.
- Points in rectangle.
- Points in grid.
- Random points.

To associate multiple machining to a cycle, first select and define a machining cycle from those allowed. Then, without quitting the editing of the cycle, press the softkey associated with multiple machining and select one of them.

## 7.1 Point-to-point turning cycle.

The cycle allows outside and inside machining.



### Geometric parameters.

#### Inside or outside machining.



Outside machining.



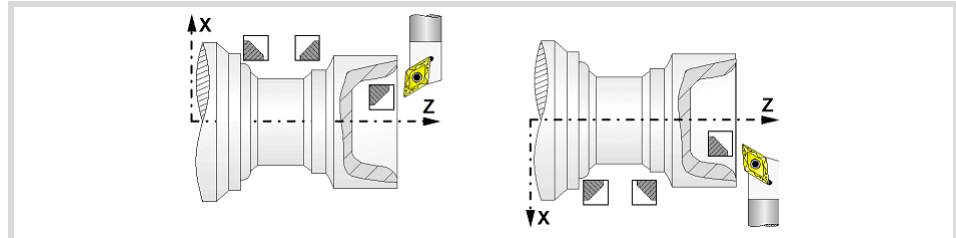
Inside machining.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Work quadrant.



Orientation of the corner to be machined.



#### Coordinates of the starting point

X, Z Coordinates of the starting point

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Safety distance.

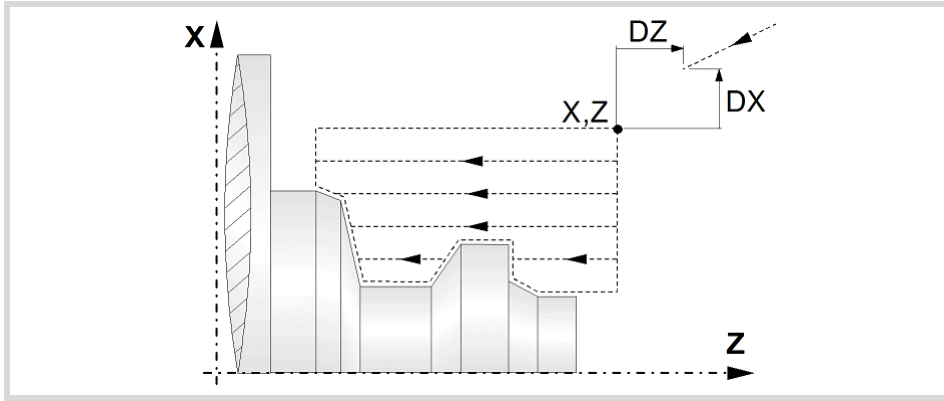
In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the approach point coordinate with respect to the starting point (X, Z).

Dx, Dz Safety distance.

The value of the safety distance on X is always defined in radius.

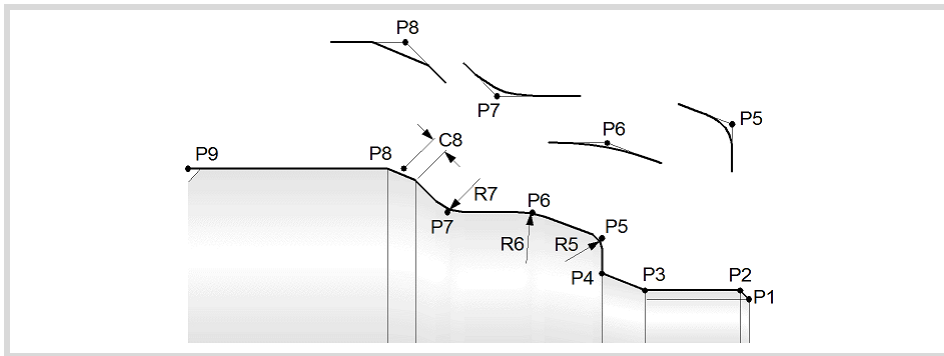
7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
Point-to-point turning cycle.



### Define a profile.

This cycle lets define the profile using the description of its theoretical corners; i.e. their position and type of machining (square corner, rounding or chamfer). The cycle lets use up to 25 points for defining said corners. The point P1 will be the beginning point of the profile and the rest of the points will be sequential.



When not using all 25 points of the table, the following conditions must be met:

- The CNC does not take into account the type of machining of the last profile point.
- The first point not used, leave the box empty or it must be defined with the same coordinates as the last point of the profile. In the example shown in the figure above P10 = P9 must be defined.

### Delete the data table.



Delete all the points of the profile. Select this icon and press [DEL] to delete all the points of the table.

### Data table.

Use the arrow keys to move through the table, the two-color key to toggle the state of the icons and the [ENTER] key to accept the data.

- Each profile point may be defined either in absolute or incremental coordinates. The type of coordinate is selected on the first column of the table.
- To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.
- All intermediate points have an icon to indicate the type of corner; square, rounded or chamfered.



Square corner.



Rounding. The rounding radius must be defined.



Chamfer. The size of the chamfer must be defined.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
Point-to-point turning cycle.

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## Spindle parameters.

### Spindle turning direction

The cycle lets select the spindle turning direction.



Spindle clockwise.



Spindle counterclockwise.

### Coolant status.



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

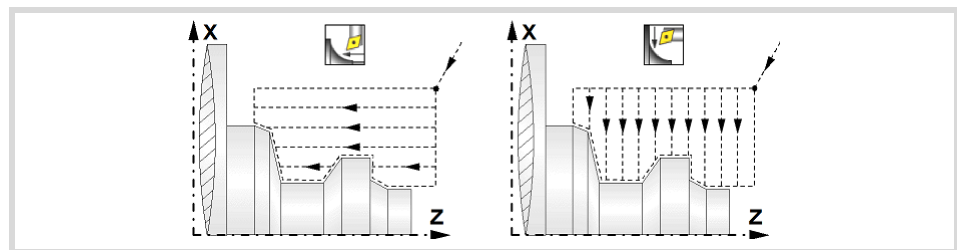
### Machining direction.



Machining in the turning direction.



Machining in the facing direction.



### Spindle work mode; rpm or Constant Surface Speed.



Constant turning speed.



Constant surface speed. Use this icon to select the spindle gear.

### Maximum spindle speed.

**Smax** Maximum spindle turning speed; if not programmed, the cycle will maintain the maximum active speed.

### Spindle range (gear).

**GEAR** Spindle gear; if not programmed, the cycle assumes a value of 0. If defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

### Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the roughing operation.



Do not run the roughing operation.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
Point-to-point turning cycle.

**Tool data.**

- T Tool. If it is set to 0, the cycle does not execute the roughing operation.
- D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining by segments.**

Option available in inside profiles, with machining in the turning direction. This parameter may be used to machine the cycle by segments indicating the maximum length of each segment along the machining axis. This type of roughing is useful on deep profiles; it lets using more robust tools (avoiding runout) and helping chip removal.

- Dp Distance for the segment to machine. When set to 0, the cycle does the roughing in a single segment.

The cycle will adjust the number of steps according to the machining length and the distance by segment "Dp". While machining, the cycle may change the value of parameter Dp to prevent the last segment to be much shorter than the other ones.

**Machining type; paraxial or pattern repeat.**

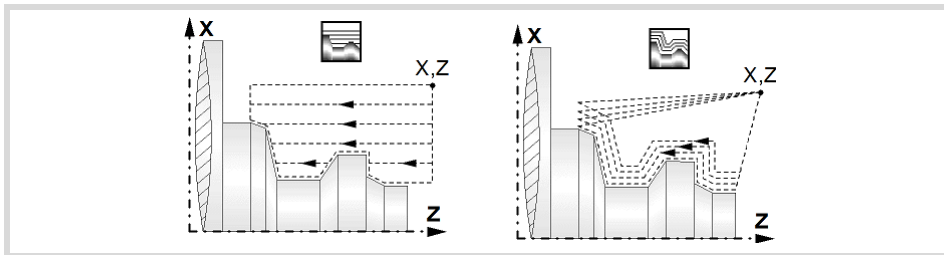


Paraxial machining. The tool penetration feedrate (Fp) must be defined for the roots (valleys).



Machining by pattern repeat (following the profile). The amount of material required to be removed from the origin part ( $\epsilon$ ) must be defined. That value is defined in radius.

- Fp Penetration feedrate in the roots for paraxial machining.
- $\epsilon$  Amount of material to machine by pattern repeat. This value must be programmed in radius and with a positive value. If not programmed, a value of 0 is assumed.



**How to finish the pass in paraxial machining.**

When paraxial roughing, the cycle offers the choice to exit at 45° at the end of each roughing pass; otherwise, each roughing pass ends following the profile.



Tool withdrawal following the profile.



Tool withdrawal with a 45° exit.

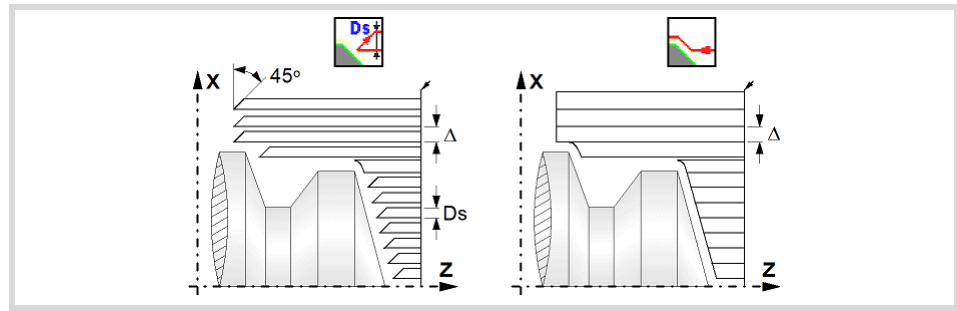
- Ds 45° exit distance after each machining pass. If not programmed, the cycle assumes value of 0.
- Fs Feedrate for the last roughing pass that removes the peaks left by the 45° exits. If not programmed or programmed with a 0 value, the cycle does not run this pass.

**7.**

CANNED CYCLES OF THE EDITOR. PROFILES.

Point-to-point turning cycle.

## 7.

**Maximum machining pass.**

- $\Delta$  Maximum machining pass.
- If  $\Delta$  is positive, the cycle recalculates the step so all the passes are identical with the same value as or smaller than the one programmed.
  - If  $\Delta$  is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

**Machining feedrate.**

F Machining feedrate.

**Spindle speed.**

S Spindle turning speed (rpm) or cutting speed.

**Machining M functions.**

Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**Machining parameters. Finishing.**

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the finishing operation.



Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes ( $\delta$ ,  $\delta x$ ,  $\delta z$ ).

**Finishing stocks.**

The cycle lets define a single finishing stock that will be applied depending on the cutter edge or 2 finishing stocks, one per axis (X, Z).



Define the finishing stock depending on the cutter edge. The finishing stock is measured on the edge of the tool.



Define 2 finishing stocks, one per axis, regardless of the type of tool being used.

$\delta$  Finishing stock depending on the cutter edge.

$\delta x$ ,  $\delta z$  Finishing stock on each axis.

**Machining feedrate.**

F Machining feedrate.

**Spindle speed.**

S Spindle turning speed (rpm) or cutting speed.



**Tool data.**

- T Tool. If it is set to 0, the cycle does not execute the finishing operation.
- D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**



Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

7.

**CANNED CYCLES OF THE EDITOR. PROFILES.**

Point-to-point turning cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 7.1.1 Basic operation.

The machining steps in these cycles are the following:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The spindle starts with the speed selected and in the direction stated.
- 3 The tool approaches in rapid up to the starting point, keeping the safety distance selected according to axes X and Z.
- 4 Roughing operation (if it has been programmed).

The cycle runs successive turning or facing passes (according to the selected option) until reaching the final profile plus the finishing stocks. This operation is done with the conditions set for the roughing operation.

Depending on the programmed conditions (tool withdrawal at the end of the pass with an exit at 45°), the cycle will run one last pass parallel to the profile to remove the peaks left by the exits at 45°.

- 5 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle runs a single pass until reaching the final profile, eliminating the finishing stocks. This operation is done with the conditions set for the finishing operation.

- 6 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
- 7 The CNC will not stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

### Considerations.

If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.

If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the roughing operation, the tool will move to the safety point (starting point plus the safety distance).

# 7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
Point-to-point turning cycle.

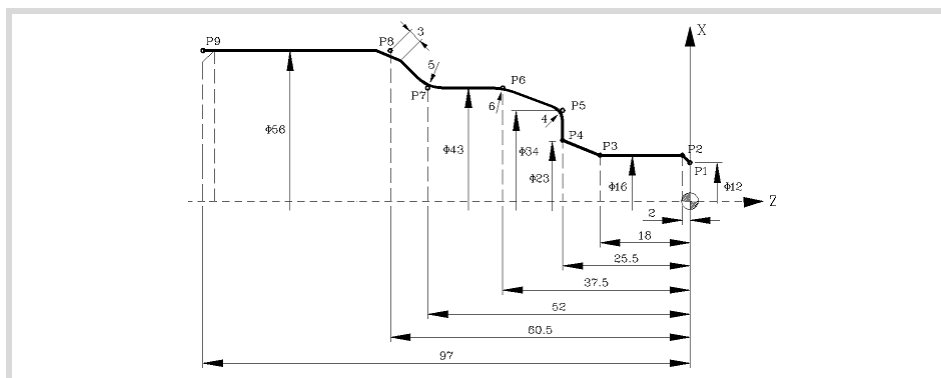


FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

### 7.1.2 Programming example.



#### Geometric parameters.



Outside machining.



Work quadrant.

X 65.0000 Starting point.  
Z 10.0000

Dx 1.0000 Safety distance.  
Dz 1.0000

#### Define the part profile.

P1	ABS	X	12.0000	Z	- 0.0000	
P2	ABS	X	16.0000	Z	- 2.0000	
P3	ABS	X	16.0000	Z	- 18.0000	
P4	ABS	X	23.0000	Z	- 25.5000	
P5	ABS	X	34.0000	Z	- 25.5000	R 4.0000
P6	ABS	X	43.0000	Z	- 37.5000	R 6.0000
P7	ABS	X	43.0000	Z	- 52.0000	R 5.0000
P8	ABS	X	56.0000	Z	- 60.5000	C 3.0000
P9	ABS	X	56.0000	Z	- 97.0000	
P10	ABS	X	56.0000	Z	- 97.0000	

#### Spindle parameters.



Machining in the turning direction.



Constant turning speed.

#### Machining parameters. Roughing.



Δ 2 F 0.8 S 1000 T 3

#### Machining parameters. Finishing.



δ 0.25 F 0.4 S 1000 T 3

# 7.

CANNED CYCLES OF THE EDITOR. PROFILES.

Point-to-point turning cycle.

**FAGOR**

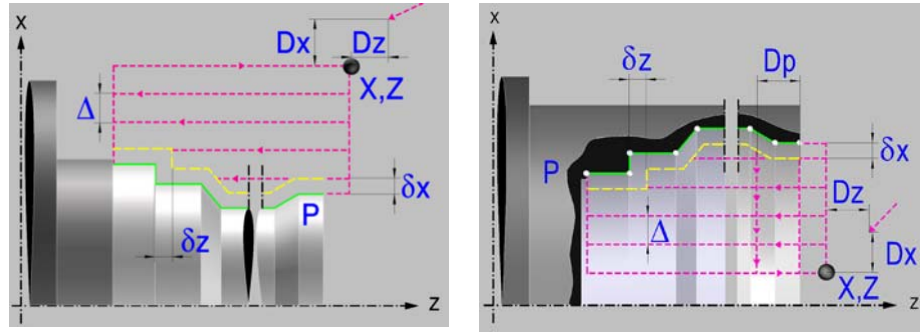
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 7.2 Profile turning cycle.

The cycle allows outside and inside machining.



### Geometric parameters.

#### Inside or outside machining.



Outside machining.



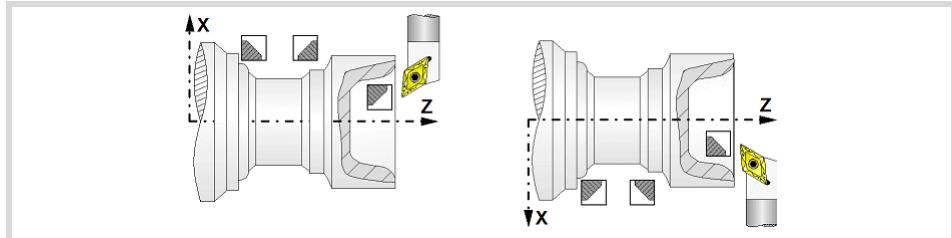
Inside machining.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Work quadrant.



Orientation of the corner to be machined.



#### Coordinates of the starting point

X, Z Coordinates of the starting point

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Safety distance.

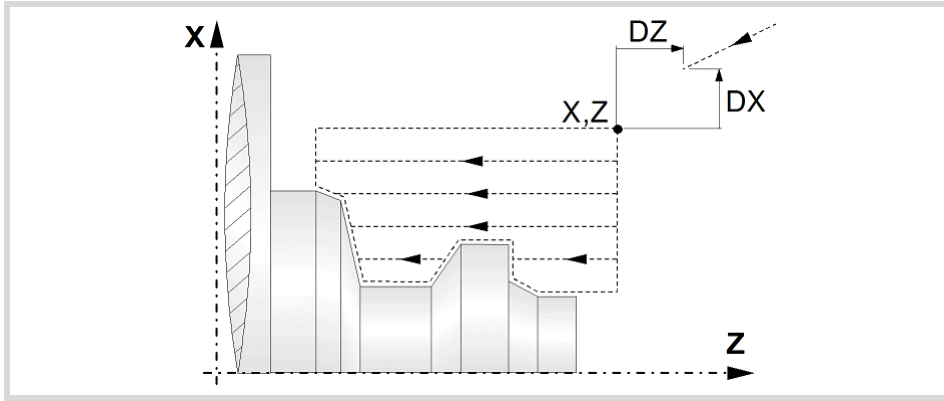
In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the approach point coordinate with respect to the starting point (X, Z).

Dx, Dz Safety distance.

The value of the safety distance on X is always defined in radius.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
Profile turning cycle.



7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
Profile turning cycle.

**Define a profile.**

This icon lets define the profile using the profile editor; it may be an existing profile or a new one.

**Profile definition.**

P Name of the program with the profile (between 0 and 999).

- Edit a new program with the profile. To edit a new program, write the program number (between 0 and 999) and press [RECALL]. The CNC shows the profile editor.
- To select a known program, key in the program number and press [ENTER].
- To select a program from the list, use the following keys.



Expand the program list.



Moves the cursor through the list of programs. To select a program, position the cursor on the desired program and press [ENTER].



Exit the list without selecting any program.

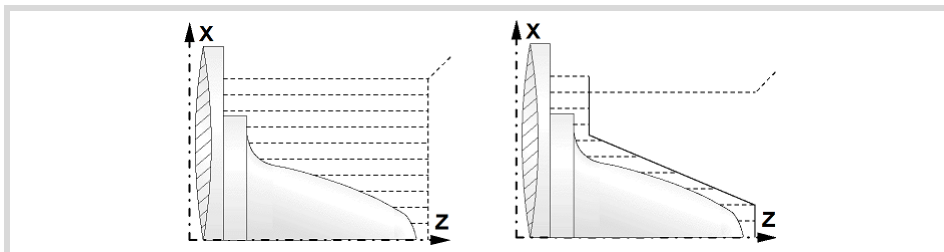
**Modify an existing profile.**

To edit an existing profile program, key in the program number and press [RECALL]. The profile editor of the CNC will show the selected profile, from where the following operations may be carried out.

- Add, delete or modify elements of the profile.
- Add, delete or modify chamfers, roundings, etc.
- Add new profiles to the program.

**Define the profile to optimize the machining operation.**

If only the desired profile is defined, the CNC assumes that the rough part is cylindrical and it machines it as shown in the drawing on the left.



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

- 1 Access the profile editor.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

- 2 Edit the desired final profile.
- 3 Press the "New profile" softkey.
- 4 Edit the rough part profile.
- 5 Exit the profile editor saving the profile.

Remember that the final profile must be defined first and then the rough profile.

# 7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
Profile turning cycle.

## Spindle parameters.

### Spindle turning direction

The cycle lets select the spindle turning direction.



Spindle clockwise.



Spindle counterclockwise.

### Coolant status.



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

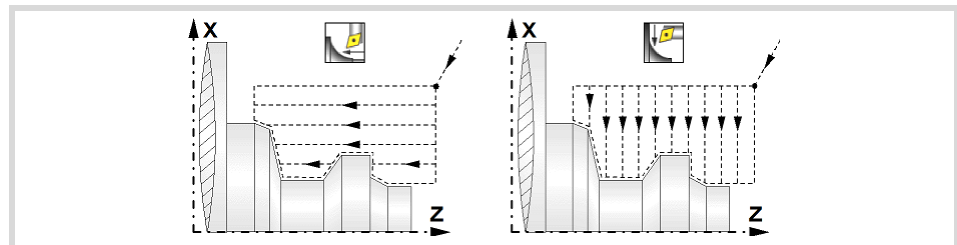
### Machining direction.



Machining in the turning direction.



Machining in the facing direction.



### Spindle work mode; rpm or Constant Surface Speed.



Constant turning speed.



Constant surface speed. Use this icon to select the spindle gear.

### Maximum spindle speed.

**Smax** Maximum spindle turning speed; if not programmed, the cycle will maintain the maximum active speed.

### Spindle range (gear).

**GEAR** Spindle gear; if not programmed, the cycle assumes a value of 0. If defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the roughing operation.



Do not run the roughing operation.

### Tool data.

T Tool. If it is set to 0, the cycle does not execute the roughing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

### Machining by segments.

Option available in inside profiles, with machining in the turning direction. This parameter may be used to machine the cycle by segments indicating the maximum length of each segment along the machining axis. This type of roughing is useful on deep profiles; it lets using more robust tools (avoiding runout) and helping chip removal.

Dp Distance for the segment to machine. When set to 0, the cycle does the roughing in a single segment.

The cycle will adjust the number of steps according to the machining length and the distance by segment "Dp". While machining, the cycle may change the value of parameter Dp to prevent the last segment to be much shorter than the other ones.

### Machining type; paraxial or pattern repeat.



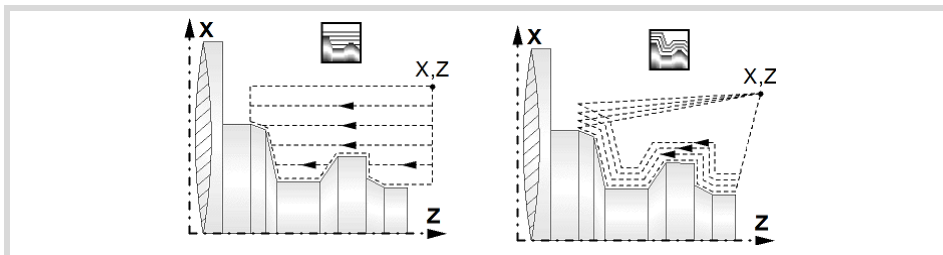
Paraxial machining. The tool penetration feedrate (Fp) must be defined for the roots (valleys).



Machining by pattern repeat (following the profile). The amount of material required to be removed from the origin part ( $\epsilon$ ) must be defined. That value is defined in radius.

Fp Penetration feedrate in the roots for paraxial machining.

$\epsilon$  Amount of material to machine by pattern repeat. This value must be programmed in radius and with a positive value. If not programmed, a value of 0 is assumed.



### How to finish the pass in paraxial machining.

When paraxial roughing, the cycle offers the choice to exit at 45° at the end of each roughing pass; otherwise, each roughing pass ends following the profile.



Tool withdrawal following the profile.



Tool withdrawal with a 45° exit.

Ds 45° exit distance after each machining pass. If not programmed, the cycle assumes value of 0.

Fs Feedrate for the last roughing pass that removes the peaks left by the 45° exits. If not programmed or programmed with a 0 value, the cycle does not run this pass.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.

Profile turning cycle.

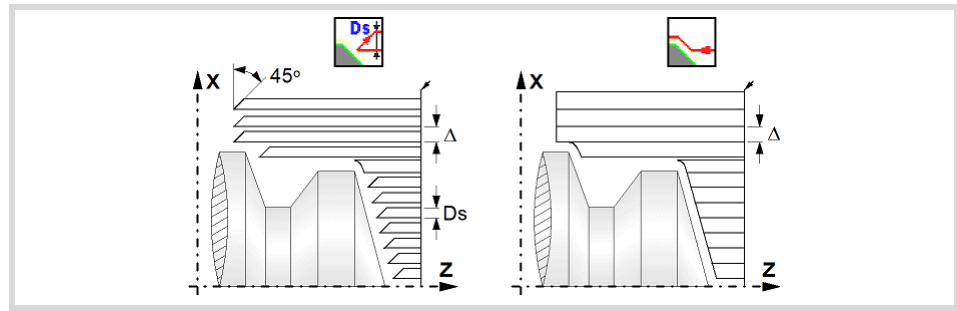
**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 7.

**Maximum machining pass.**

- $\Delta$  Maximum machining pass.
- If  $\Delta$  is positive, the cycle recalculates the step so all the passes are identical with the same value as or smaller than the one programmed.
  - If  $\Delta$  is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

**Machining feedrate.**

F Machining feedrate.

**Spindle speed.**

S Spindle turning speed (rpm) or cutting speed.

**Machining M functions.**

Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**Machining parameters. Finishing.**

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the finishing operation.



Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes ( $\delta$ ,  $\delta x$ ,  $\delta z$ ).

**Finishing stocks.**

The cycle lets define a single finishing stock that will be applied depending on the cutter edge or 2 finishing stocks, one per axis (X, Z).



Define the finishing stock depending on the cutter edge. The finishing stock is measured on the edge of the tool.



Define 2 finishing stocks, one per axis, regardless of the type of tool being used.

$\delta$  Finishing stock depending on the cutter edge.

$\delta x$ ,  $\delta z$  Finishing stock on each axis.

**Machining feedrate.**

F Machining feedrate.

**Spindle speed.**

S Spindle turning speed (rpm) or cutting speed.



**Tool data.**

- T Tool. If it is set to 0, the cycle does not execute the finishing operation.
- D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**



Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.



**CANNED CYCLES OF THE EDITOR. PROFILES.**

Profile turning cycle.



FAGOR AUTOMATION

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

(REF: 1709)

## 7.2.1 Basic operation.

The machining steps in these cycles are the following:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The spindle starts with the speed selected and in the direction stated.
- 3 The tool approaches in rapid up to the starting point, keeping the safety distance selected according to axes X and Z.
- 4 Roughing operation (if it has been programmed).

The cycle runs successive turning or facing passes (according to the selected option) until reaching the final profile plus the finishing stocks. This operation is done with the conditions set for the roughing operation.

Depending on the programmed conditions (tool withdrawal at the end of the pass with an exit at 45°), the cycle will run one last pass parallel to the profile to remove the peaks left by the exits at 45°.

- 5 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle runs a single pass until reaching the final profile, eliminating the finishing stocks. This operation is done with the conditions set for the finishing operation.

- 6 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
- 7 The CNC will not stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

### Considerations.

If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.

If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the roughing operation, the tool will move to the safety point (starting point plus the safety distance).

# 7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
Profile turning cycle.

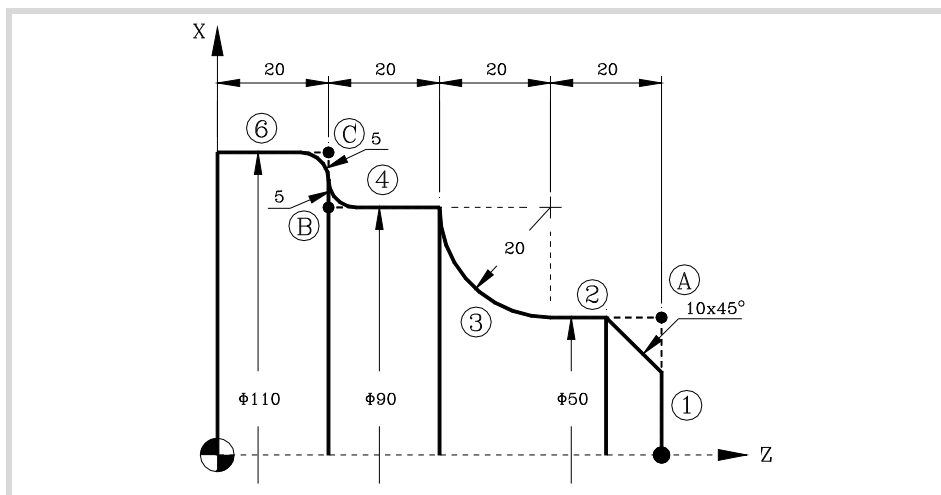


FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 7.2.2 Programming example.



### Geometric parameters.



Outside machining.



Work quadrant.

X 120.0000 Starting point.  
Z 90.0000

Dx 1.0000 Safety distance.  
Dz 1.0000

### Define the part profile.

Definition of a profile without rounding or chamfers.

Section.	Geometry.
Starting point.	Z = 80 X = 0
(1) Straight line.	Z = 80 X = 50
(2) Straight line.	Z = 60 X = 50
(3) Clockwise arc.	Z = 40 X = 90 Zc = 60 Xc = 90 R = 20
(4) Straight line.	Z = 20 X = 90
(5) Straight line.	Z = 20 X = 110
(6) Straight line.	Z = 0 X = 110

Definition of roundings and chamfers.

Corners.	Point.	Radius/Size.
Chamfer.	Point "A".	Size = 10
Rounding.	Point "B".	Radius = 5
Rounding.	Point "C".	Radius = 5

### Spindle parameters.



Machining in the turning direction.



Constant turning speed.

### Machining parameters. Roughing.



$\Delta$  2 F 0.8 S 1000 T 3

### Machining parameters. Finishing.



$\delta$  0.25 F 0.4 S 1000 T 3

# 7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
Profile turning cycle.

**FAGOR** 

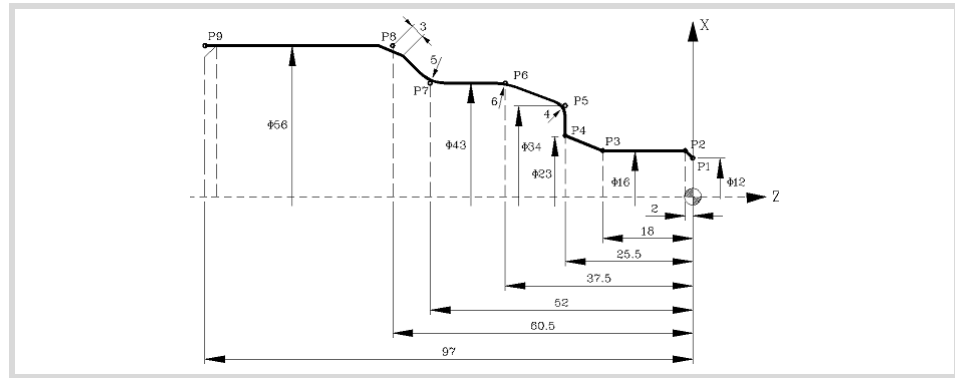
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

# 7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
Profile turning cycle.



### Geometric parameters.



Outside machining.



Work quadrant.

X 65.0000  
Z 10.0000

Starting point.

Dx 1.0000  
Dz 1.0000

Safety distance.

### Define the part profile.

Definition of a profile without rounding or chamfers.

Section.	Geometry.	
Starting point.	Z = 0	X = 0
(1) Straight line.	Z = 0	X = 16
(2) Straight line.	Z = -18	X = 16
(3) Straight line.	Z = -25.53	X = 23
(4) Straight line.	Z = -25.5	X = 34
(5) Straight line.	Z = -37.5	X = 43
(6) Straight line.	Z = -52	X = 43
(7) Straight line.	Z = -60.5	X = 56
(8) Straight line.	Z = -97	X = 56

Definition of roundings and chamfers.

Corners.	Point.	Radius/Size.
Chamfer.	Point "A".	Size = 2
Rounding.	Point "B".	Radius = 4
Rounding.	Point "C".	Radius = 6
Rounding.	Point "D".	Radius = 5
Chamfer.	Point "E".	Size = 3

### Spindle parameters.



Machining in the turning direction.



Constant turning speed.

### Machining parameters. Roughing.

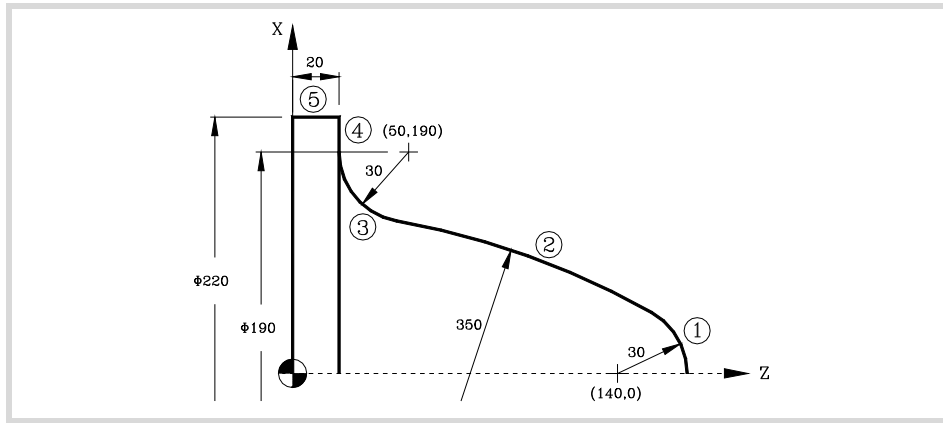


Δ 2      F 0.8      S 1000      T 3

### Machining parameters. Finishing.



δ 0.25      F 0.4      S 1000      T 3



**Geometric parameters.**



Outside machining.



Work quadrant.

X 230.0000 Starting point.  
Z 180.0000

Dx 1.0000 Safety distance.  
Dz 1.0000

**Define the part profile.**

Section.	Geometry.			
Starting point.	Z = 170	X = 0		
(1) Counterclockwise arc.	Zc = 140	Xc = 0	R = 30	
(2) Counterclockwise arc.	R = 350	Tangency = Yes		
(3) Clockwise arc.	Zc = 50	Xc = 190	R = 30	Tangency = Yes
• The CNC shows all the possible options for section 2. Select the correct one.				
(4) Straight line.	Z = 20	X = 220	Tangency = Yes	
• The CNC displays all the possible options between sections 3-4. Select the correct one.				
(5) Straight line.	Z = 0	X = 220		

**Spindle parameters.**



Machining in the turning direction.



Constant turning speed.

**Machining parameters. Roughing.**



Δ 2 F 0.8 S 1000 T 3

**Machining parameters. Finishing.**



δ 0.25 F 0.4 S 1000 T 3

**7.**

**CANNED CYCLES OF THE EDITOR. PROFILES.**  
Profile turning cycle.



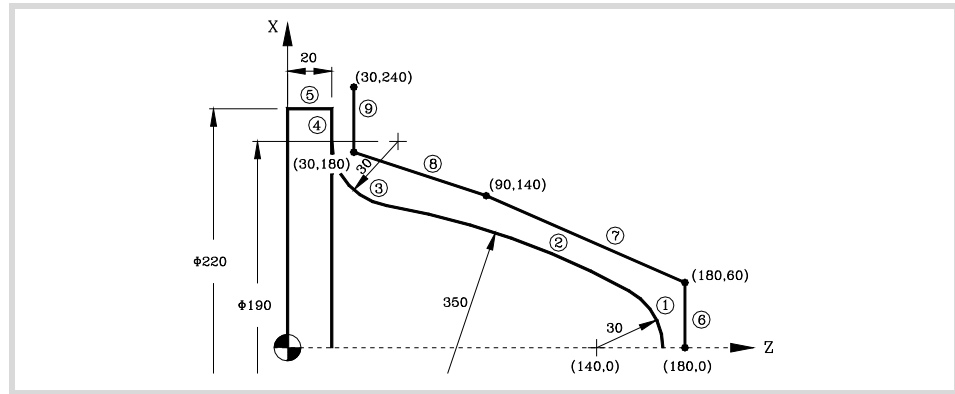
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

# 7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
Profile turning cycle.



### Geometric parameters.



Outside machining.



Work quadrant.

X 230.0000 Starting point.  
Z 180.0000

Dx 1.0000 Safety distance.  
Dz 1.0000

### Define the part profile.

Section.	Geometry.			
Starting point.	Z = 170	X = 0		
(1) Counterclockwise arc.	Zc = 140	Xc = 0	R = 30	
(2) Counterclockwise arc.	R = 350	Tangency = Yes		
(3) Clockwise arc.	Zc = 50	Xc = 190	R = 30	Tangency = Yes
• The CNC shows all the possible options for section 2. Select the correct one.				
(4) Straight line.	Z = 20	X = 220	Tangency = Yes	
• The CNC displays all the possible options between sections 3-4. Select the correct one.				
(5) Straight line.	Z = 0	X = 220		

### Define the raw part profile.

Section.	Geometry.	
Starting point.	Z = 180	X = 0
(6) Straight line.	Z = 180	X = 60
(7) Straight line.	Z = 90	X = 140
(8) Straight line.	Z = 30	X = 180
(9) Straight line.	Z = 30	X = 240

### Spindle parameters.



Machining in the turning direction.



Constant turning speed.

### Machining parameters. Roughing.

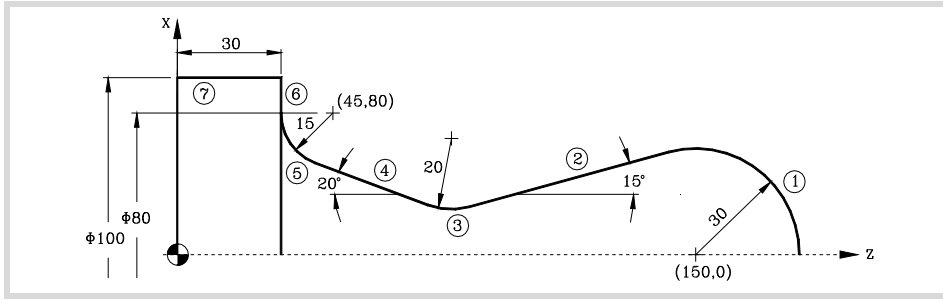


Δ 2 F 0.8 S 1000 T 3

### Machining parameters. Finishing.



δ 0.25 F 0.4 S 1000 T 3



**Geometric parameters.**



Outside machining.



Work quadrant.

X 110.0000 Starting point.  
Z 190.0000

Dx 1.0000 Safety distance.  
Dz 1.0000

**Define the part profile.**

Section.	Geometry.		
Starting point.	Z = 180	X = 0	
(1) Counterclockwise arc.	Zc = 150	Xc = 0	R = 30
(2) Straight line.	Angle = 195	Tangency = Yes	
• The CNC displays all the possible options between sections 1-2. Select the correct one.			
(3) Clockwise arc.	R = 20	Tangency = Yes	
(4) Straight line.	Angle = 160	Tangency = Yes	
(5) Clockwise arc.	Z = 30 X = 80	Zc = 45 Xc = 80	R = 15 Tangency = Yes
• The CNC displays all the possible options between sections 4-5. Select the correct one.			
• The CNC shows all the possible options for section 3. Select the correct one.			
(6) Straight line.	Z = 30	X = 100	Tangency = Yes
(7) Straight line.	Z = 0	X = 100	

**Spindle parameters.**



Machining in the turning direction.



Constant turning speed.

**Machining parameters. Roughing.**



Δ 2 F 0.8 S 1000 T 3

**Machining parameters. Finishing.**



δ 0.25 F 0.4 S 1000 T 3

**7.**

CANNED CYCLES OF THE EDITOR. PROFILES.

Profile turning cycle.



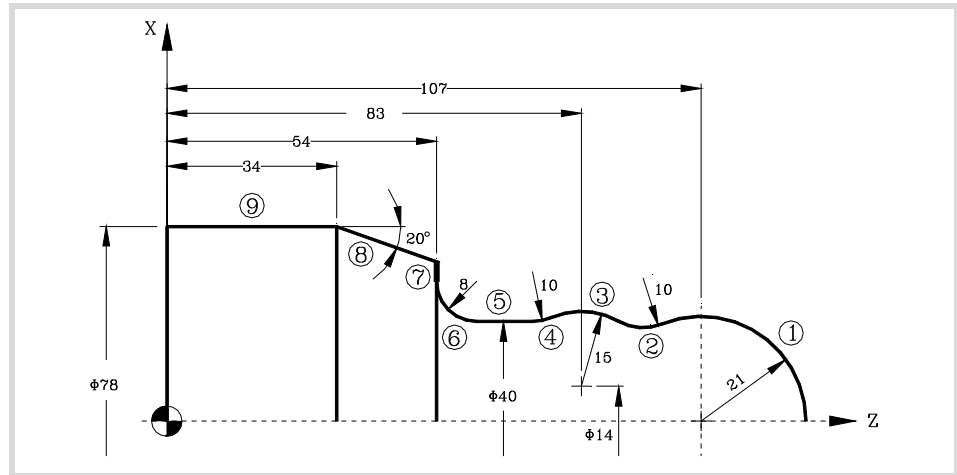
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

# 7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
Profile turning cycle.



### Geometric parameters.



Outside machining.



Work quadrant.

X 85.0000 Starting point.  
Z 135.0000

Dx 1.0000 Safety distance.  
Dz 1.0000

### Define the part profile.

Section.	Geometry.			
Starting point.	Z = 128	X = 0		
(1) Counterclockwise arc.	Zc = 107	Xc = 0	R = 21	
(2) Clockwise arc.	R = 10	Tangency = Yes		
(3) Counterclockwise arc.	Zc = 83	Xc = 14	R = 15	Tangency = Yes
• The CNC shows all the possible options for section 2. Select the correct one.				
(4) Clockwise arc.	R = 10	Tangency = Yes		
(5) Straight line.	X = 40	Angle = 180	Tangency = Yes	
• The CNC shows all the possible options for section 3. Select the correct one.				
(6) Clockwise arc.	Z = 54 X = 56	Zc = 62 Xc = 56	R = 8	Tangency = Yes
(7) Straight line.	Z = 54	Angle = 90	Tangency = Yes	
(8) Straight line.	Z = 34	X = 78	Angle = 160	
(9) Straight line.	Z = 0	X = 78		

### Spindle parameters.



Machining in the turning direction.



Constant turning speed.

### Machining parameters. Roughing.



Δ 2 F 0.8 S 1000 T 3

### Machining parameters. Finishing.



δ 0.25 F 0.4 S 1000 T 3



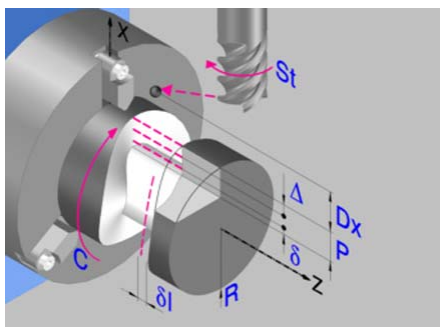
FAGOR AUTOMATION

CNC 8060  
CNC 8065

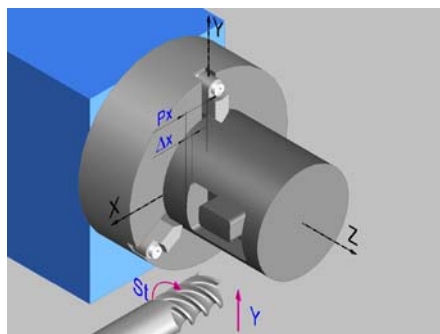
(REF: 1709)



## 7.3 ZC/YZ plane profiling cycle.



Profile in the ZC plane.



Profile in the YZ plane.

### Geometric parameters.

#### Work plane.



ZC plane.



YZ plane.

#### Name of the program with the profile.

P Name of the program with the profile (between 0 and 999).

#### Radius of the cylinder (if ZC plane).

R Outside radius of the part.

#### Coordinates of the plane (if YZ plane).

X X coordinate of the work plane.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Coordinates of the final point (if YZ plane).

Yf Y coordinate for the withdrawal movement after the cycle ends; if not programmed, there is no Y axis withdrawal. If the cycle is machining in multiple mode, the Y withdrawal point is only applied to the last cycle.

This parameter lets withdraw the tool to the right position in order to be able to continue with the turning operations that do not use the Y axis. If the next cycle also uses the Y axis, there is no need to withdraw (retract) the axis.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Total depth.

P Total depth. This parameter must always be defined in radius and with a positive value.

#### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point. The cycle also uses the safety distance for the withdrawal movement between successive roughing passes.

Dx X safety distance referred to the starting point. This parameter is always defined in radius.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
ZC/YZ plane profiling cycle.

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Spindle angular position.**

- Ⓞ Angular position of the spindle, in degrees.

**Define a profile.**

This icon lets define the profile using the profile editor; it may be an existing profile or a new one.

**Profile definition.**

P Name of the program with the profile (between 0 and 999).

- Edit a new program with the profile. To edit a new program, write the program number (between 0 and 999) and press [RECALL]. The CNC shows the profile editor.
- To select a known program, key in the program number and press [ENTER].
- To select a program from the list, use the following keys.



Expand the program list.



Moves the cursor through the list of programs. To select a program, position the cursor on the desired program and press [ENTER].



Exit the list without selecting any program.

**Modify an existing profile.**

To edit an existing profile program, key in the program number and press [RECALL]. The profile editor of the CNC will show the selected profile, from where the following operations may be carried out.

- Add, delete or modify elements of the profile.
- Add, delete or modify chamfers, roundings, etc.
- Add new profiles to the program.

**Spindle parameters.****Live tool turning direction.**

Live tool clockwise.



Live tool counter-clockwise.

**Coolant status.**

Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

**Tool radius compensation.**

Without compensation



Left-hand tool radius compensation.



Right-hand tool radius compensation.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
ZC/YZ plane profiling cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Penetration feedrate.**

Fx Penetration feedrate.

**Machining parameters. Roughing.**

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.

- Run the roughing operation.
- Do not run the roughing operation.

**Maximum penetration step.**

- Δ Maximum penetration step.
  - If Δ is positive, the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
  - If Δ is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

**Machining feedrate.**

F Machining feedrate.

**Live-tool data.**

- Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.
- Live tool data programmed. The cycle shows the data related to the live tool.
- S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Tool data.**

- T Tool. If it is set to 0, the cycle does not execute the roughing operation.
- D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

- Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.
- Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**Machining parameters. Finishing.**

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.

- Run the finishing operation.
- Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes (δ, δl) (δ, δl).

**7.**

CANNED CYCLES OF THE EDITOR. PROFILES.  
ZC/YZ plane profiling cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

Number of penetration passes (steps) for the finishing operation.

N Number of penetration passes (steps) for the finishing operation. This parameter is only available when tool radius compensation is active.

Finishing stocks.

$\delta l$  Side finishing stock. This parameter is only available when tool radius compensation is active.

$\delta$  Bottom finishing stock.

Machining feedrate.

F Machining feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycle assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Tool data.**

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
ZC/YZ plane profiling cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

### 7.3.1 Basic operation.

The machining steps in these cycles are the following:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The live tool starts with the speed selected and in the indicated direction.
- 3 The tool approaches in rapid up to the starting point, keeping the selected safety distance according to the X axis.
- 4 The spindle orients to the indicated "C" position.
- 5 Roughing operation (if it has been programmed).

The cycle runs successive passes, up to a distance from the final profile equal to the finishing stock. This cycle applies withdrawal between passes at the safety distance Dx. This operation is done with the conditions set for the roughing operation.

- 6 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle eliminates the finishing stocks. If tool radius compensation is active, the cycle runs the indicated number of passes until the final profile is reached; if it is not active, the cycle runs a single pass. This operation is done with the conditions set for the finishing operation.

- 7 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
- 8 The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

#### Considerations.

If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.

If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the roughing operation, the tool will move to the safety point (starting point plus the safety distance).

# 7.

CANNED CYCLES OF THE EDITOR. PROFILES.

ZC/YZ plane profiling cycle.

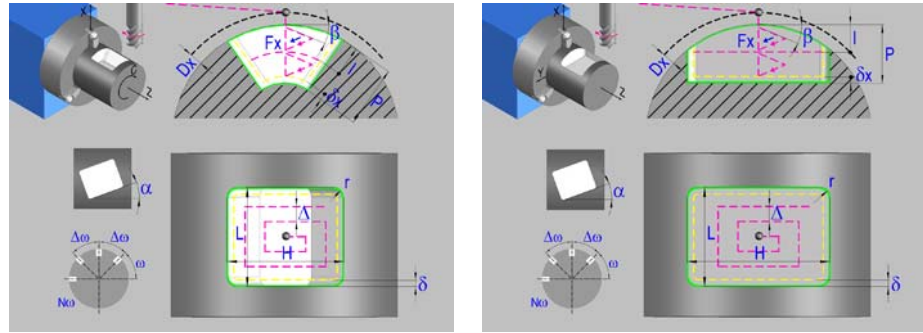
**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 7.4 Rectangular pocket cycle in the ZC/YZ plane.



Rectangular pocket in the ZC plane.

Rectangular pocket in the YZ plane.

### Geometric parameters.

#### Work plane.



ZC plane.



YZ plane.

#### Starting point selection.



Starting point at the corner.



Starting point in the center.

#### Type of pocket corners.



Square (sharp) corner.



Rounded corner. The rounding radius must be defined.



Chamfered corner. The size of the chamfer must be defined.

$r$  Rounding radius of the pocket corners.

$c$  Size of the chamfer of the pocket corners.

#### Coordinates of the starting point

Z, C Coordinates of the starting point (if ZC plane).

Z, Y Coordinates of the starting point (if YZ plane).

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Coordinates of the final point (if YZ plane).

Yf Y coordinate for the withdrawal movement after the cycle ends; if not programmed, there is no Y axis withdrawal. If the cycle is machining in multiple mode, the Y withdrawal point is only applied to the last cycle.

This parameter lets withdraw the tool to the right position in order to be able to continue with the turning operations that do not use the Y axis. If the next cycle also uses the Y axis, there is no need to withdraw (retract) the axis.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
Rectangular pocket cycle in the ZC/YZ plane.

**Pocket dimensions.**

- L Pocket length on the Z axis.
- H Length of the pocket on the C axis (if ZC plane) or on the Y axis (if YZ plane).

**Pocket orientation.**

- $\alpha$  Pocket angle referred to Z axis, in degrees.

**Radius of the cylinder (if ZC plane).**

- R Outside radius of the part.

**Coordinates of the plane (if YZ plane).**

- X X coordinate of the work plane.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

**Pocket depth.**

- P Pocket depth. This parameter is always defined in radius.

**Safety distance.**

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

- Dx X safety distance referred to the starting point. This parameter is always defined in radius.

**Angular position of the machining operations.**

- $\omega$  Angular position of the spindle (in degrees) for the first machining operation.
- $\Delta\omega$  Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.
- $N\omega$  Total number of operations.

**Spindle parameters.**

**Live tool turning direction.**



Live tool clockwise.



Live tool counter-clockwise.

**Coolant status.**



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

**Penetration feedrate.**

- Fx Penetration feedrate.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.

Rectangular pocket cycle in the ZC/YZ plane.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.

- Run the roughing operation.
- Do not run the roughing operation.

### Maximum penetration step.

- I Maximum penetration step.
- If the parameter is positive, the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
  - If the parameter is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

### Sideways penetration angle.

- $\beta$  Sideways penetration angle.

### Milling pass or width.

- $\Delta$  Milling pass or width.

The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

### Machining feedrate.

- F Machining feedrate.



### Live-tool data.

- Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.
  - Live tool data programmed. The cycle shows the data related to the live tool.
- S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

### Machining direction.

-  Clockwise.
-  Counterclockwise.

### Tool data.

- T Tool. If it is set to 0, the cycle does not execute the roughing operation.
- D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
Rectangular pocket cycle in the ZC/YZ plane.



### Machining M functions.



Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

### Machining parameters. Finishing.

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the finishing operation.



Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes ( $\delta$ ,  $\delta l$ ) ( $\delta$ ,  $\delta l$ ).

### Sideways penetration angle.

$\theta$  Sideways penetration angle.

### Finishing stocks.

$\delta$  Side finishing stock.

$\delta x$  Bottom finishing stock. If it is set to 0, the cycle does not execute the finishing operation at the bottom.

### Machining feedrate.

F Machining feedrate.

### Live-tool data.



Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.



Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

### Machining direction.



Clockwise.



Counterclockwise.

### Number of side penetration passes (steps) for the finishing operation.

N Number of side penetration passes (steps) for the finishing operation. If defined with a value other than 0, the cycle executes N finishing passes, even if no side finishing stock has been programmed. If it is set to 0, the cycle does not execute the side finishing operation.

### Tool data.

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.

Rectangular pocket cycle in the ZC/YZ plane.

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

- D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

#### Machining M functions.



Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

# 7.

## CANNED CYCLES OF THE EDITOR. PROFILES.

Rectangular pocket cycle in the ZC/YZ plane.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 7.4.1 Basic operation.

The machining steps in these cycles are the following:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The live tool starts with the speed selected and in the indicated direction.
- 3 The tool approaches in rapid up to the starting point, keeping the selected safety distance according to the X axis.
- 4 The spindle orients to the indicated "C" position.
- 5 Roughing operation (if it has been programmed).

The cycle runs successive passes, up to a distance from the final profile equal to the finishing stock. This operation is done with the conditions set for the roughing operation.

- 6 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle eliminates the finishing stocks. This operation is done with the conditions set for the finishing operation.

The cycle offers two finishing stocks; at the bottom and on the walls. If both stocks are programmed ( $\delta$ ,  $\delta x$ ), the cycle executes the finishing operation at the bottom first and then on the walls. The finishing on the walls is carried out in as many penetration passes as indicated by parameter N. If there are no side finishing stocks, but parameter N is other than 0, the cycle executes N finishing passes.

- 7 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
- 8 The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

### Considerations.

#### Define a tool as T0.

If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.

If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the roughing operation, the tool will move to the safety point (starting point plus the safety distance).

#### Roughing and finishing in multiple pockets.

If the roughing tool and the finishing tool are different, the cycle executes first all the roughing operations and then all the finishing operations.

If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- If no M function has been defined, or they are the same for both operations, the cycle executes the whole pocket (roughing and finishing) before going on to the next one.
- If the M functions assigned to each operation are different, the cycle executes first all the roughing operations and then all the finishing operations.

# 7.

## CANNED CYCLES OF THE EDITOR. PROFILES.

Rectangular pocket cycle in the ZC/YZ plane.

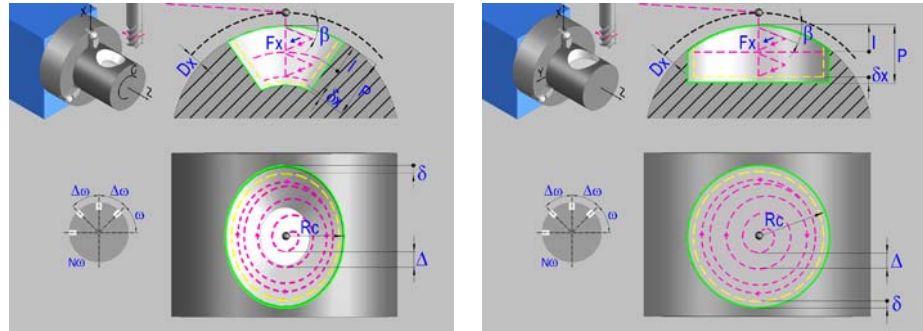


FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 7.5 Circular pocket cycle in the ZC/YZ plane.



Circular pocket in the ZC plane.

Circular pocket in the YZ plane.

### Geometric parameters.

#### Work plane.



ZC plane.



YZ plane.

#### Coordinates of the center of the pocket.

Zc, Cc Coordinates of the center of the pocket (if ZC plane).

Zc, Yc Coordinates of the center of the pocket (if YZ plane).

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Coordinates of the final point (if YZ plane).

Yf Y coordinate for the withdrawal movement after the cycle ends; if not programmed, there is no Y axis withdrawal. If the cycle is machining in multiple mode, the Y withdrawal point is only applied to the last cycle.

This parameter lets withdraw the tool to the right position in order to be able to continue with the turning operations that do not use the Y axis. If the next cycle also uses the Y axis, there is no need to withdraw (retract) the axis.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Pocket radius.

Rc Pocket radius.

#### Radius of the cylinder (if ZC plane).

R Outside radius of the part.

#### Coordinates of the plane (if YZ plane).

X X coordinate of the work plane.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Pocket depth.

P Pocket depth. This parameter is always defined in radius.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
Circular pocket cycle in the ZC/YZ plane.

### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dx X safety distance referred to the starting point. This parameter is always defined in radius.

### Angular position of the machining operations.

$\omega$  Angular position of the spindle (in degrees) for the first machining operation.

$\Delta\omega$  Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

$N\omega$  Total number of operations.

### Spindle parameters.

#### Live tool turning direction.



Live tool clockwise.



Live tool counter-clockwise.

#### Coolant status.



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

#### Penetration feedrate.

Fx Penetration feedrate.

### Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the roughing operation.



Do not run the roughing operation.

#### Maximum penetration step.

I Maximum penetration step.

- If the parameter is positive, the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If the parameter is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

#### Sideways penetration angle.

$\beta$  Sideways penetration angle.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.

Circular pocket cycle in the ZC/NZ plane.

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Milling pass or width.**

Δ Milling pass or width.

The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

**Machining feedrate.**

F Machining feedrate.

**Live-tool data.**

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycle assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Machining direction.**

Clockwise.



Counterclockwise.

**Tool data.**

T Tool. If it is set to 0, the cycle does not execute the roughing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**Machining parameters. Finishing.**

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the finishing operation.



Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes (δ, δl) (δ, δl).

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
Circular pocket cycle in the ZC/MZ plane.

**Sideways penetration angle.**

$\theta$  Sideways penetration angle.

**Finishing stocks.**


$\delta$  Side finishing stock.


$\delta x$  Bottom finishing stock. If it is set to 0, the cycle does not execute the finishing operation at the bottom.

**Machining feedrate.**

F Machining feedrate.

**Live-tool data.**

 Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

 Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Machining direction.**



Clockwise.



Counterclockwise.

**Number of side penetration passes (steps) for the finishing operation.**


N Number of side penetration passes (steps) for the finishing operation. If defined with a value other than 0, the cycle executes N finishing passes, even if no side finishing stock has been programmed. If it is set to 0, the cycle does not execute the side finishing operation.


**Tool data.**

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

 Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

 Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**7.**

**CANNED CYCLES OF THE EDITOR. PROFILES.**  
Circular pocket cycle in the ZC/NZ plane.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 7.5.1 Basic operation.

The machining steps in these cycles are the following:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The live tool starts with the speed selected and in the indicated direction.
- 3 The tool approaches in rapid up to the starting point, keeping the selected safety distance according to the X axis.
- 4 The spindle orients to the indicated "C" position.
- 5 Roughing operation (if it has been programmed).

The cycle runs successive passes, up to a distance from the final profile equal to the finishing stock. This operation is done with the conditions set for the roughing operation.

- 6 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle eliminates the finishing stocks. This operation is done with the conditions set for the finishing operation.

The cycle offers two finishing stocks; at the bottom and on the walls. If both stocks are programmed ( $\delta$ ,  $\delta x$ ), the cycle executes the finishing operation at the bottom first and then on the walls. The finishing on the walls is carried out in as many penetration passes as indicated by parameter N. If there are no side finishing stocks, but parameter N is other than 0, the cycle executes N finishing passes.

- 7 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
- 8 The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

### Considerations.

#### Define a tool as T0.

If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.

If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the roughing operation, the tool will move to the safety point (starting point plus the safety distance).

#### Roughing and finishing in multiple pockets.

If the roughing tool and the finishing tool are different, the cycle executes first all the roughing operations and then all the finishing operations.

If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

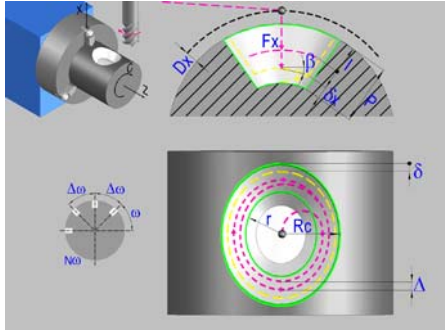
- If no M function has been defined, or they are the same for both operations, the cycle executes the whole pocket (roughing and finishing) before going on to the next one.
- If the M functions assigned to each operation are different, the cycle executes first all the roughing operations and then all the finishing operations.

# 7.

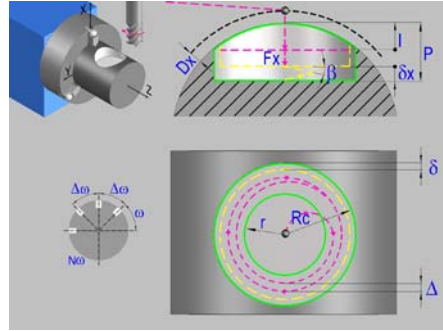
CANNED CYCLES OF THE EDITOR. PROFILES.  
Circular pocket cycle in the ZC/MZ plane.



## 7.6 Circular pre-empted pocket cycle in the ZC/YZ plane.



Circular pre-empted pocket in the ZC plane.



Circular pre-empted pocket in the YZ plane.

### Geometric parameters.

#### Work plane.



ZC plane.



YZ plane.

#### Coordinates of the center of the pocket.

Zc, Cc Coordinates of the center of the pocket (if ZC plane).

Zc, Yc Coordinates of the center of the pocket (if YZ plane).

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Coordinates of the final point (if YZ plane).

Yf Y coordinate for the withdrawal movement after the cycle ends; if not programmed, there is no Y axis withdrawal. If the cycle is machining in multiple mode, the Y withdrawal point is only applied to the last cycle.

This parameter lets withdraw the tool to the right position in order to be able to continue with the turning operations that do not use the Y axis. If the next cycle also uses the Y axis, there is no need to withdraw (retract) the axis.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Pocket radius.

Rc Pocket radius.

r Inside radius of the pocket.

#### Radius of the cylinder (if ZC plane).

R Outside radius of the part.

#### Coordinates of the plane (if YZ plane).

X X coordinate of the work plane.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Pocket depth.

P Pocket depth. This parameter is always defined in radius.

# 7.

CANNED CYCLES OF THE EDITOR. PROFILES.

Circular pre-empted pocket cycle in the ZC/YZ plane.

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Safety distance.**

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dx X safety distance referred to the starting point. This parameter is always defined in radius.

**Angular position of the machining operations.**

$\omega$  Angular position of the spindle (in degrees) for the first machining operation.

$\Delta\omega$  Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

$N\omega$  Total number of operations.

**Spindle parameters.****Live tool turning direction.**

Live tool clockwise.



Live tool counter-clockwise.

**Coolant status.**

Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

**Penetration feedrate.**

Fx Penetration feedrate.

**Machining parameters. Roughing.**

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the roughing operation.



Do not run the roughing operation.

**Maximum penetration step.**

I Maximum penetration step.

- If the parameter is positive, the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If the parameter is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

**Sideways penetration angle.**

$\beta$  Sideways penetration angle.

If the tool fits in the pre-empted area, it goes in straight in G0, ignoring the penetration angle. If the tool does not fit in the pre-empted area, it goes in at the programmed penetration angle.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
Circular pre-empted pocket cycle in the ZC/MZ plane.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

### Milling pass or width.


Δ Milling pass or width.


The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

### Machining feedrate.

F Machining feedrate.

### Live-tool data.

 Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

 Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

### Machining direction.



Clockwise.




Counterclockwise.


### Tool data.

T Tool. If it is set to 0, the cycle does not execute the roughing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

### Machining M functions.


 Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.


 Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

### Machining parameters. Finishing.

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.

 Run the finishing operation.

 Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes (δ, δl) (δ, δl).

### Sideways penetration angle.

θ Sideways penetration angle.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
Circular pre-empted pocket cycle in the ZC/YZ plane.

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)



**Finishing stocks.**

- $\delta$  Side finishing stock.
- $\delta x$  Bottom finishing stock. If it is set to 0, the cycle does not execute the finishing operation at the bottom.

**Machining feedrate.**

- F Machining feedrate.



**Live-tool data.**

-  Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.
-  Live tool data programmed. The cycle shows the data related to the live tool.
- S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Machining direction.**

-  Clockwise.
-  Counterclockwise.



**Number of side penetration passes (steps) for the finishing operation.**

- N Number of side penetration passes (steps) for the finishing operation. If defined with a value other than 0, the cycle executes N finishing passes, even if no side finishing stock has been programmed. If it is set to 0, the cycle does not execute the side finishing operation.

**Tool data.**

- T Tool. If it is set to 0, the cycle does not execute the finishing operation.
- D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

-  Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.
-  Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
Circular pre-empted pocket cycle in the ZC/MZ plane.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 7.6.1 Basic operation.

The machining steps in these cycles are the following:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The live tool starts with the speed selected and in the indicated direction.
- 3 The tool approaches in rapid up to the starting point, keeping the selected safety distance according to the X axis.
- 4 The spindle orients to the indicated "C" position.
- 5 Roughing operation (if it has been programmed).

The cycle runs successive passes, up to a distance from the final profile equal to the finishing stock. This operation is done with the conditions set for the roughing operation.

- 6 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle eliminates the finishing stocks. This operation is done with the conditions set for the finishing operation.

The cycle offers two finishing stocks; at the bottom and on the walls. If both stocks are programmed ( $\delta$ ,  $\delta x$ ), the cycle executes the finishing operation at the bottom first and then on the walls. The finishing on the walls is carried out in as many penetration passes as indicated by parameter N. If there are no side finishing stocks, but parameter N is other than 0, the cycle executes N finishing passes.

- 7 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
- 8 The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

### Considerations.

#### Define a tool as T0.

If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.

If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the roughing operation, the tool will move to the safety point (starting point plus the safety distance).

#### Roughing and finishing in multiple pockets.

If the roughing tool and the finishing tool are different, the cycle executes first all the roughing operations and then all the finishing operations.

If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- If no M function has been defined, or they are the same for both operations, the cycle executes the whole pocket (roughing and finishing) before going on to the next one.
- If the M functions assigned to each operation are different, the cycle executes first all the roughing operations and then all the finishing operations.

# 7.

CANNED CYCLES OF THE EDITOR. PROFILES.

Circular pre-empted pocket cycle in the ZC/NZ plane.

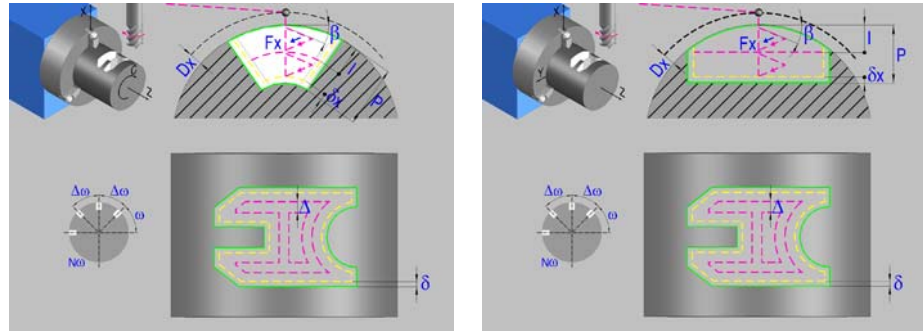
**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 7.7 2D profile pocket cycle in the ZC/YZ plane.



2D profile pocket in the ZC plane.

2D profile pocket in the YZ plane.

### Geometric parameters.

#### Work plane.



ZC plane.



YZ plane.

#### Name of the program with the profile.

P Name of the program with the profile (between 0 and 999).

#### Radius of the cylinder (if ZC plane).

R Outside radius of the part.

#### Coordinates of the plane (if YZ plane).

X X coordinate of the work plane.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Coordinates of the final point (if YZ plane).

Yf Y coordinate for the withdrawal movement after the cycle ends; if not programmed, there is no Y axis withdrawal. If the cycle is machining in multiple mode, the Y withdrawal point is only applied to the last cycle.

This parameter lets withdraw the tool to the right position in order to be able to continue with the turning operations that do not use the Y axis. If the next cycle also uses the Y axis, there is no need to withdraw (retract) the axis.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Total depth.

P Total depth. This parameter must always be defined in radius and with a positive value.

#### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dx X safety distance referred to the starting point. This parameter is always defined in radius.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
2D profile pocket cycle in the ZC/YZ plane.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

### Angular position of the machining operations.

- $\omega$  Angular position of the spindle (in degrees) for the first machining operation.
- $\Delta\omega$  Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.
- $N\omega$  Total number of operations.






### Define a profile.

This icon lets define the profile using the profile editor; it may be an existing profile or a new one.

#### Profile definition.

P Name of the program with the profile (between 0 and 999).

- Edit a new program with the profile. To edit a new program, write the program number (between 0 and 999) and press [RECALL]. The CNC shows the profile editor.
- To select a known program, key in the program number and press [ENTER].
- To select a program from the list, use the following keys.

-  Expand the program list.
-   Moves the cursor through the list of programs. To select a program, position the cursor on the desired program and press [ENTER].
-   Exit the list without selecting any program.

#### Modify an existing profile.

To edit an existing profile program, key in the program number and press [RECALL]. The profile editor of the CNC will show the selected profile, from where the following operations may be carried out.

- Add, delete or modify elements of the profile.
- Add, delete or modify chamfers, roundings, etc.
- Add new profiles to the program.

### Spindle parameters.

#### Live tool turning direction.



Live tool clockwise.



Live tool counter-clockwise.

#### Coolant status.



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
2D profile pocket cycle in the ZC/YZ plane.

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Penetration feedrate.**

Fx Penetration feedrate.

**Machining parameters. Roughing.**

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.

Run the roughing operation.

Do not run the roughing operation.

**Maximum penetration step.**

I Maximum penetration step.

- If the parameter is positive, the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If the parameter is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

**Sideways penetration angle.** $\beta$  Sideways penetration angle.**Milling pass or width.** $\Delta$  Milling pass or width.

The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

**Machining feedrate.**

F Machining feedrate.

**Live-tool data.**

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Tool data.**

T Tool. If it is set to 0, the cycle does not execute the roughing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
2D profile pocket cycle in the ZC/MZ plane.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)



## Machining parameters. Finishing.

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the finishing operation.



Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes ( $\delta$ ,  $\delta l$ ) ( $\delta$ ,  $\delta l$ ).

### Sideways penetration angle.

$\theta$  Sideways penetration angle.

### Finishing stocks.

$\delta$  Side finishing stock.

$\delta x$  Bottom finishing stock. If it is set to 0, the cycle does not execute the finishing operation at the bottom.

### Machining feedrate.

F Machining feedrate.

### Live-tool data.



Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.



Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

### Number of side penetration passes (steps) for the finishing operation.

N Number of side penetration passes (steps) for the finishing operation. If defined with a value other than 0, the cycle executes N finishing passes, even if no side finishing stock has been programmed. If it is set to 0, the cycle does not execute the side finishing operation.

### Tool data.

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

### Machining M functions.



Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
2D profile pocket cycle in the ZC/YZ plane.

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 7.7.1 Basic operation.

The machining steps in these cycles are the following:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The live tool starts with the speed selected and in the indicated direction.
- 3 The tool approaches in rapid up to the starting point, keeping the selected safety distance according to the X axis.
- 4 The spindle orients to the indicated "C" position.
- 5 Roughing operation (if it has been programmed).

The cycle runs successive passes, up to a distance from the final profile equal to the finishing stock. This operation is done with the conditions set for the roughing operation.

- 6 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle eliminates the finishing stocks. This operation is done with the conditions set for the finishing operation.

The cycle offers two finishing stocks; at the bottom and on the walls. If both stocks are programmed ( $\delta$ ,  $\delta x$ ), the cycle executes the finishing operation at the bottom first and then on the walls. The finishing on the walls is carried out in as many penetration passes as indicated by parameter N. If there are no side finishing stocks, but parameter N is other than 0, the cycle executes N finishing passes.

- 7 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
- 8 The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

### Considerations.

#### Define a tool as T0.

If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.

If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the roughing operation, the tool will move to the safety point (starting point plus the safety distance).

#### Roughing and finishing in multiple pockets.

If the roughing tool and the finishing tool are different, the cycle executes first all the roughing operations and then all the finishing operations.

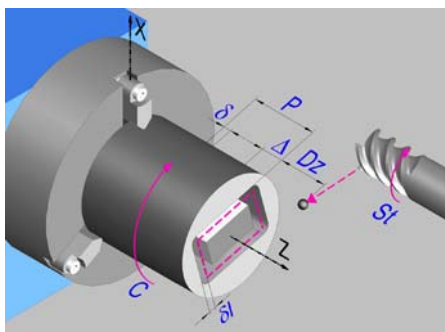
If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- If no M function has been defined, or they are the same for both operations, the cycle executes the whole pocket (roughing and finishing) before going on to the next one.
- If the M functions assigned to each operation are different, the cycle executes first all the roughing operations and then all the finishing operations.

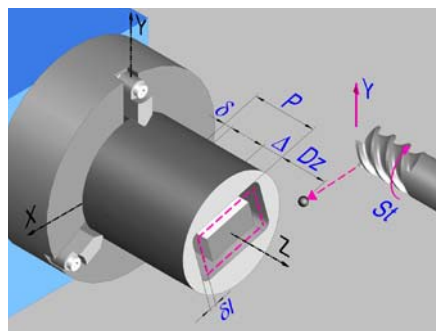
# 7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
2D profile pocket cycle in the ZC/MZ plane.

## 7.8 XC/XY plane profiling cycle.



Profile in the XC plane.



Profile in the XY plane.

### Geometric parameters.

#### Work plane.



XC plane.



XY plane.

#### Name of the program with the profile.

P Name of the program with the profile (between 0 and 999).

#### Coordinates of the final point (if XY plane).

Yf Y coordinate for the withdrawal movement after the cycle ends; if not programmed, there is no Y axis withdrawal. If the cycle is machining in multiple mode, the Y withdrawal point is only applied to the last cycle.

This parameter lets withdraw the tool to the right position in order to be able to continue with the turning operations that do not use the Y axis. If the next cycle also uses the Y axis, there is no need to withdraw (retract) the axis.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Coordinates of the starting point

Z Z coordinate of the penetration point into the part.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Total depth.

P Total depth.

#### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point. The withdrawal movement between successive roughing passes is done at 1 mm of the surface.

Dz X safety distance referred to the starting point.

#### Spindle angular position.

$\omega$  Angular position of the spindle, in degrees.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
XC/XY plane profiling cycle.

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## Define a profile.

This icon lets define the profile using the profile editor; it may be an existing profile or a new one.

### Profile definition.

P Name of the program with the profile (between 0 and 999).

- Edit a new program with the profile. To edit a new program, write the program number (between 0 and 999) and press [RECALL]. The CNC shows the profile editor.
- To select a known program, key in the program number and press [ENTER].
- To select a program from the list, use the following keys.



Expand the program list.



Moves the cursor through the list of programs. To select a program, position the cursor on the desired program and press [ENTER].



Exit the list without selecting any program.

### Modify an existing profile.

To edit an existing profile program, key in the program number and press [RECALL]. The profile editor of the CNC will show the selected profile, from where the following operations may be carried out.

- Add, delete or modify elements of the profile.
- Add, delete or modify chamfers, roundings, etc.
- Add new profiles to the program.

## Spindle parameters.

### Live tool turning direction.



Live tool clockwise.



Live tool counter-clockwise.

### Coolant status.



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

### Tool radius compensation.



Without compensation



Left-hand tool radius compensation.



Right-hand tool radius compensation.

### Penetration feedrate.

Fz Penetration feedrate.

# 7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
XC/XY plane profiling cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.

- Run the roughing operation.
- Do not run the roughing operation.

### Maximum penetration step.

- $\Delta$  Maximum penetration step.
- If  $\Delta$  is positive, the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
  - If  $\Delta$  is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

### Machining feedrate.

- F Machining feedrate.

### Live-tool data.

- Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.
  - Live tool data programmed. The cycle shows the data related to the live tool.
- S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

### Tool data.

- T Tool. If it is set to 0, the cycle does not execute the roughing operation.
- D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

### Machining M functions.

- Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.
- Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

## Machining parameters. Finishing.

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.

- Run the finishing operation.
- Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes ( $\delta$ ,  $\delta l$ ) ( $\delta$ ,  $\delta l$ ).

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
XC/XY plane profiling cycle.

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Number of penetration passes (steps) for the finishing operation.**

N Number of penetration passes (steps) for the finishing operation. This parameter is only available when tool radius compensation is active.

**Finishing stocks.**

$\delta l$  Side finishing stock. This parameter is only available when tool radius compensation is active.

$\delta$  Bottom finishing stock.

**Machining feedrate.**

F Machining feedrate.

**Live-tool data.**

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycle assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Tool data.**

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
XC/XY plane profiling cycle.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 7.8.1 Basic operation.

The machining steps in these cycles are the following:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The live tool starts with the speed selected and in the indicated direction.
- 3 The tool approaches in rapid up to the starting point, keeping the selected safety distance according to the Z axis.
- 4 The spindle orients to the indicated "C" position.
- 5 Roughing operation (if it has been programmed).

The cycle runs successive passes, up to a distance from the final profile equal to the finishing stock. This cycle applies withdrawal between passes at 1 mm from the surface. This operation is done with the conditions set for the roughing operation.

- 6 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle eliminates the finishing stocks. If tool radius compensation is active, the cycle runs the indicated number of passes until the final profile is reached; if it is not active, the cycle runs a single pass. This operation is done with the conditions set for the finishing operation.

- 7 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
- 8 The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

### Considerations.

If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.

If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the roughing operation, the tool will move to the safety point (starting point plus the safety distance).

7.

CANNED CYCLES OF THE EDITOR. PROFILES.

XC/XY plane profiling cycle.

**FAGOR** 

FAGOR AUTOMATION

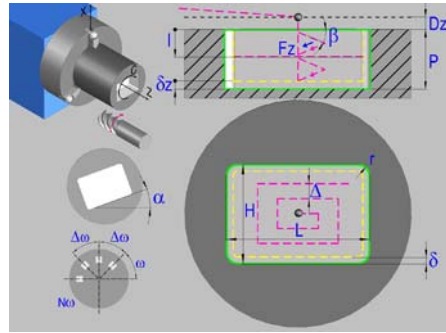
CNC 8060  
CNC 8065

(REF: 1709)

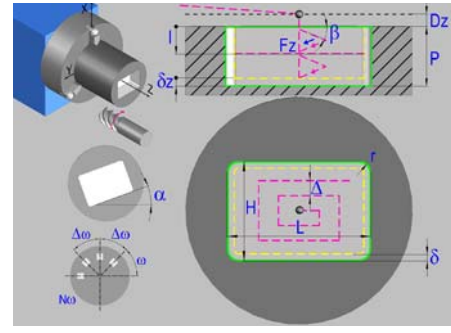
## 7.9 Rectangular pocket cycle in the XC/XY plane.

# 7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
Rectangular pocket cycle in the XC/XY plane.



Rectangular pocket in the XC plane.



Rectangular pocket in the XY plane.

### Geometric parameters.

#### Work plane.



XC plane.



XY plane.

#### Starting point selection.



Starting point at the corner.



Starting point in the center.

#### Type of pocket corners.



Square (sharp) corner.



Rounded corner. The rounding radius must be defined.



Chamfered corner. The size of the chamfer must be defined.

$r$  Rounding radius of the pocket corners.

$c$  Size of the chamfer of the pocket corners.

#### Coordinates of the starting point

X, C Coordinates of the starting point (if XC plane).

X, Y Coordinates of the starting point (if XY plane).

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Coordinates of the final point (if XY plane).

Yf Y coordinate for the withdrawal movement after the cycle ends; if not programmed, there is no Y axis withdrawal. If the cycle is machining in multiple mode, the Y withdrawal point is only applied to the last cycle.

This parameter lets withdraw the tool to the right position in order to be able to continue with the turning operations that do not use the Y axis. If the next cycle also uses the Y axis, there is no need to withdraw (retract) the axis.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.



**Pocket dimensions.**

- L Pocket length on the X axis.
- H Length of the pocket on the C axis (if XC plane) or on the Y axis (if XY plane).

**Pocket orientation.**

- $\alpha$  Pocket angle referred to X axis, in degrees.

**Coordinates of the plane.**

- Z Z coordinate of the work plane.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

**Total depth.**

- P Total depth.

**Safety distance.**

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

- Dz X safety distance referred to the starting point.

**Angular position of the machining operations.**

- $\omega$  Angular position of the spindle (in degrees) for the first machining operation.
- $\Delta\omega$  Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.
- $N\omega$  Total number of operations.

**Spindle parameters.**

**Live tool turning direction.**



Live tool clockwise.



Live tool counter-clockwise.

**Coolant status.**



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

**Penetration feedrate.**

- Fz Penetration feedrate.

**7.**

**CANNED CYCLES OF THE EDITOR. PROFILES.**

Rectangular pocket cycle in the XC/XY plane.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.

- Run the roughing operation.  
 Do not run the roughing operation.

### Maximum penetration step.

- I Maximum penetration step.
- If the parameter is positive, the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
  - If the parameter is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

### Sideways penetration angle.

- $\beta$  Sideways penetration angle.

### Milling pass or width.

- $\Delta$  Milling pass or width.

The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

### Machining feedrate.

- F Machining feedrate.



### Live-tool data.

- Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.  
 Live tool data programmed. The cycle shows the data related to the live tool.  
S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

### Machining direction.

-  Clockwise.  
 Counterclockwise.

### Tool data.

- T Tool. If it is set to 0, the cycle does not execute the roughing operation.  
D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
 Rectangular pocket cycle in the XC/XY plane.

### Machining M functions.



Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

### Machining parameters. Finishing.

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the finishing operation.



Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes ( $\delta$ ,  $\delta l$ ) ( $\delta$ ,  $\delta l$ ).

### Sideways penetration angle.

$\theta$  Sideways penetration angle.

### Finishing stocks.

$\delta$  Side finishing stock.

$\delta x$  Bottom finishing stock. If it is set to 0, the cycle does not execute the finishing operation at the bottom.

### Machining feedrate.

F Machining feedrate.

### Live-tool data.



Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.



Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

### Machining direction.



Clockwise.



Counterclockwise.

### Number of side penetration passes (steps) for the finishing operation.

N Number of side penetration passes (steps) for the finishing operation. If defined with a value other than 0, the cycle executes N finishing passes, even if no side finishing stock has been programmed. If it is set to 0, the cycle does not execute the side finishing operation.

### Tool data.

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.

Rectangular pocket cycle in the Xc/Yc plane.

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

- D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

#### Machining M functions.



Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

# 7.

**CANNED CYCLES OF THE EDITOR. PROFILES.**  
Rectangular pocket cycle in the XC/XY plane.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 7.9.1 Basic operation.

The machining steps in these cycles are the following:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The live tool starts with the speed selected and in the indicated direction.
- 3 The tool approaches in rapid up to the starting point, keeping the selected safety distance according to the Z axis.
- 4 The spindle orients to the indicated "C" position.
- 5 Roughing operation (if it has been programmed).

The cycle runs successive passes, up to a distance from the final profile equal to the finishing stock. This operation is done with the conditions set for the roughing operation.

- 6 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle eliminates the finishing stocks. This operation is done with the conditions set for the finishing operation.

The cycle offers two finishing stocks; at the bottom and on the walls. If both stocks are programmed ( $\delta$ ,  $\delta x$ ), the cycle executes the finishing operation at the bottom first and then on the walls. The finishing on the walls is carried out in as many penetration passes as indicated by parameter N. If there are no side finishing stocks, but parameter N is other than 0, the cycle executes N finishing passes.

- 7 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
- 8 The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

### Considerations.

#### Define a tool as T0.

If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.

If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the roughing operation, the tool will move to the safety point (starting point plus the safety distance).

#### Roughing and finishing in multiple pockets.

If the roughing tool and the finishing tool are different, the cycle executes first all the roughing operations and then all the finishing operations.

If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- If no M function has been defined, or they are the same for both operations, the cycle executes the whole pocket (roughing and finishing) before going on to the next one.
- If the M functions assigned to each operation are different, the cycle executes first all the roughing operations and then all the finishing operations.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.

Rectangular pocket cycle in the Xc/Yc plane.

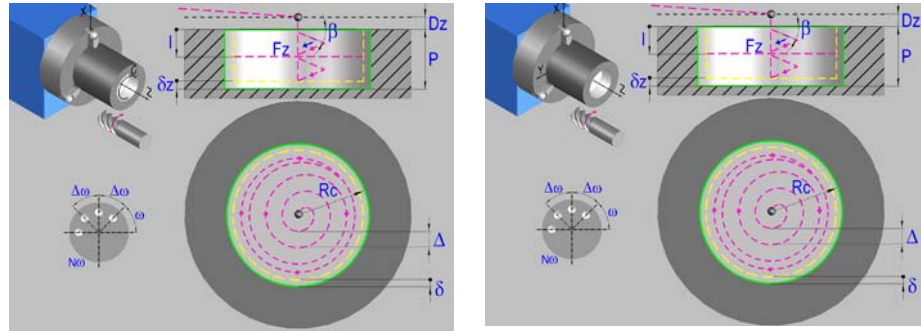
**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 7.10 Circular pocket cycle in the XC/XY plane.



Circular pocket in the XC plane.

Circular pocket in the XY plane.

### Geometric parameters.

#### Work plane.



XC plane.



XY plane.

#### Coordinates of the center of the pocket.

Xc, Cc Coordinates of the center of the pocket (if XC plane).

Xc, Yc Coordinates of the center of the pocket (if XY plane).

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Coordinates of the final point (if XY plane).

Yf Y coordinate for the withdrawal movement after the cycle ends; if not programmed, there is no Y axis withdrawal. If the cycle is machining in multiple mode, the Y withdrawal point is only applied to the last cycle.

This parameter lets withdraw the tool to the right position in order to be able to continue with the turning operations that do not use the Y axis. If the next cycle also uses the Y axis, there is no need to withdraw (retract) the axis.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Pocket radius.

Rc Pocket radius.

#### Coordinates of the plane.

Z Z coordinate of the work plane.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Total depth.

P Total depth.

#### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dz X safety distance referred to the starting point.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
Circular pocket cycle in the XC/XY plane.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

### Angular position of the machining operations.

- $\omega$  Angular position of the spindle (in degrees) for the first machining operation.
- $\Delta\omega$  Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.
- $N\omega$  Total number of operations.

### Spindle parameters.

#### Live tool turning direction.



Live tool clockwise.



Live tool counter-clockwise.

#### Coolant status.



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

#### Penetration feedrate.

- $Fz$  Penetration feedrate.

### Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the roughing operation.



Do not run the roughing operation.

#### Maximum penetration step.

- $I$  Maximum penetration step.
  - If the parameter is positive, the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
  - If the parameter is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

#### Sideways penetration angle.

- $\beta$  Sideways penetration angle.

#### Milling pass or width.

- $\Delta$  Milling pass or width.

The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

#### Machining feedrate.

- $F$  Machining feedrate.

# 7.

CANNED CYCLES OF THE EDITOR. PROFILES.

Circular pocket cycle in the XC/XY plane.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
Circular pocket cycle in the XC/XY plane.

**Live-tool data.**

- Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.
- Live tool data programmed. The cycle shows the data related to the live tool.
- S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Machining direction.**

Clockwise.



Counterclockwise.

**Tool data.**

- T Tool. If it is set to 0, the cycle does not execute the roughing operation.
- D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**Machining parameters. Finishing.**

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the finishing operation.



Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes ( $\delta$ ,  $\delta l$ ) ( $\delta$ ,  $\delta l$ ).

**Sideways penetration angle.**

- $\theta$  Sideways penetration angle.

**Finishing stocks.**

- $\delta$  Side finishing stock.
- $\delta x$  Bottom finishing stock. If it is set to 0, the cycle does not execute the finishing operation at the bottom.

**Machining feedrate.**

- F Machining feedrate.



**Live-tool data.**



Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.



Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Machining direction.**



Clockwise.



Counterclockwise.

**Number of side penetration passes (steps) for the finishing operation.**

N Number of side penetration passes (steps) for the finishing operation. If defined with a value other than 0, the cycle executes N finishing passes, even if no side finishing stock has been programmed. If it is set to 0, the cycle does not execute the side finishing operation.

**Tool data.**

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**



Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**7.**

**CANNED CYCLES OF THE EDITOR. PROFILES.**

Circular pocket cycle in the XC/XY plane.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 7.10.1 Basic operation.

The machining steps in these cycles are the following:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The live tool starts with the speed selected and in the indicated direction.
- 3 The tool approaches in rapid up to the starting point, keeping the selected safety distance according to the Z axis.
- 4 The spindle orients to the indicated "C" position.
- 5 Roughing operation (if it has been programmed).

The cycle runs successive passes, up to a distance from the final profile equal to the finishing stock. This operation is done with the conditions set for the roughing operation.

- 6 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle eliminates the finishing stocks. This operation is done with the conditions set for the finishing operation.

The cycle offers two finishing stocks; at the bottom and on the walls. If both stocks are programmed ( $\delta$ ,  $\delta x$ ), the cycle executes the finishing operation at the bottom first and then on the walls. The finishing on the walls is carried out in as many penetration passes as indicated by parameter N. If there are no side finishing stocks, but parameter N is other than 0, the cycle executes N finishing passes.

- 7 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
- 8 The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

### Considerations.

#### Define a tool as T0.

If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.

If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the roughing operation, the tool will move to the safety point (starting point plus the safety distance).

#### Roughing and finishing in multiple pockets.

If the roughing tool and the finishing tool are different, the cycle executes first all the roughing operations and then all the finishing operations.

If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- If no M function has been defined, or they are the same for both operations, the cycle executes the whole pocket (roughing and finishing) before going on to the next one.
- If the M functions assigned to each operation are different, the cycle executes first all the roughing operations and then all the finishing operations.

# 7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
Circular pocket cycle in the XC/XY plane.



**Safety distance.**

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dz X safety distance referred to the starting point.

**Angular position of the machining operations.**

$\omega$  Angular position of the spindle (in degrees) for the first machining operation.

$\Delta\omega$  Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

$N\omega$  Total number of operations.

**Spindle parameters.****Live tool turning direction.**

Live tool clockwise.



Live tool counter-clockwise.

**Coolant status.**

Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

**Penetration feedrate.**

Fz Penetration feedrate.

**Machining parameters. Roughing.**

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the roughing operation.



Do not run the roughing operation.

**Maximum penetration step.**

I Maximum penetration step.

- If the parameter is positive, the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If the parameter is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

**Sideways penetration angle.**

$\beta$  Sideways penetration angle.

If the tool fits in the pre-empted area, it goes in straight in G0, ignoring the penetration angle. If the tool does not fit in the pre-empted area, it goes in at the programmed penetration angle.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
Circular pre-empted pocket in the XC/XY plane.

### Milling pass or width.


Δ Milling pass or width.


The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

### Machining feedrate.

F Machining feedrate.

### Live-tool data.

 Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

 Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

### Machining direction.



Clockwise.




Counterclockwise.


### Tool data.

T Tool. If it is set to 0, the cycle does not execute the roughing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

### Machining M functions.


 Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.


 Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

### Machining parameters. Finishing.

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.

 Run the finishing operation.

 Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes (δ, δl) (δ, δl).

### Sideways penetration angle.

θ Sideways penetration angle.

7.

**CANNED CYCLES OF THE EDITOR. PROFILES.**  
Circular pre-empted pocket in the XC/XY plane.

**FAGOR** 

FAGOR AUTOMATION

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

(REF: 1709)



**Finishing stocks.**

- $\delta$  Side finishing stock.
- $\delta x$  Bottom finishing stock. If it is set to 0, the cycle does not execute the finishing operation at the bottom.

**Machining feedrate.**

- F Machining feedrate.



**Live-tool data.**

-  Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.
-  Live tool data programmed. The cycle shows the data related to the live tool.
- S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycle assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Machining direction.**

-  Clockwise.
-  Counterclockwise.



**Number of side penetration passes (steps) for the finishing operation.**

- N Number of side penetration passes (steps) for the finishing operation. If defined with a value other than 0, the cycle executes N finishing passes, even if no side finishing stock has been programmed. If it is set to 0, the cycle does not execute the side finishing operation.

**Tool data.**

- T Tool. If it is set to 0, the cycle does not execute the finishing operation.
- D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

-  Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.
-  Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

7.

**CANNED CYCLES OF THE EDITOR. PROFILES.**  
Circular pre-empted pocket in the XC/XY plane.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

### 7.11.1 Basic operation.

The machining steps in these cycles are the following:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The live tool starts with the speed selected and in the indicated direction.
- 3 The tool approaches in rapid up to the starting point, keeping the selected safety distance according to the Z axis.
- 4 The spindle orients to the indicated "C" position.
- 5 Roughing operation (if it has been programmed).

The cycle runs successive passes, up to a distance from the final profile equal to the finishing stock. This operation is done with the conditions set for the roughing operation.

- 6 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle eliminates the finishing stocks. This operation is done with the conditions set for the finishing operation.

The cycle offers two finishing stocks; at the bottom and on the walls. If both stocks are programmed ( $\delta$ ,  $\delta x$ ), the cycle executes the finishing operation at the bottom first and then on the walls. The finishing on the walls is carried out in as many penetration passes as indicated by parameter N. If there are no side finishing stocks, but parameter N is other than 0, the cycle executes N finishing passes.

- 7 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
- 8 The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

### Considerations.

#### Define a tool as T0.

If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.

If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the roughing operation, the tool will move to the safety point (starting point plus the safety distance).

#### Roughing and finishing in multiple pockets.

If the roughing tool and the finishing tool are different, the cycle executes first all the roughing operations and then all the finishing operations.

If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- If no M function has been defined, or they are the same for both operations, the cycle executes the whole pocket (roughing and finishing) before going on to the next one.
- If the M functions assigned to each operation are different, the cycle executes first all the roughing operations and then all the finishing operations.

# 7.

## CANNED CYCLES OF THE EDITOR. PROFILES.

Circular pre-empted pocket in the XC/XY plane.

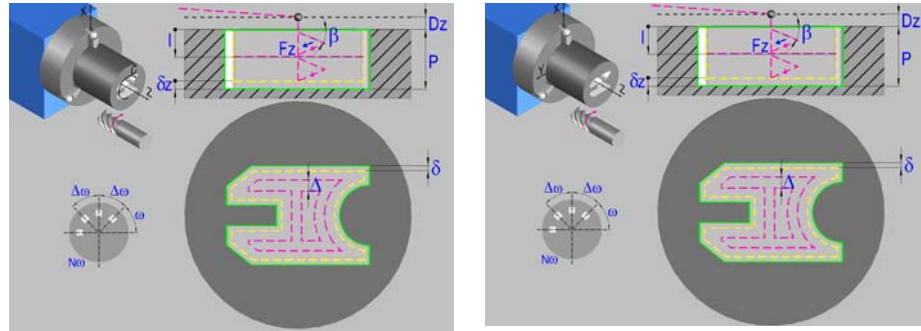


FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 7.12 2D profile pocket cycle in the XC/XY plane.



2D profile pocket in the XC plane.

2D profile pocket in the XY plane.

**Geometric parameters.****Work plane.**

XC plane.



XY plane.

**Name of the program with the profile.**

P Name of the program with the profile (between 0 and 999).

**Coordinates of the final point (if XY plane).**

Yf Y coordinate for the withdrawal movement after the cycle ends; if not programmed, there is no Y axis withdrawal. If the cycle is machining in multiple mode, the Y withdrawal point is only applied to the last cycle.

This parameter lets withdraw the tool to the right position in order to be able to continue with the turning operations that do not use the Y axis. If the next cycle also uses the Y axis, there is no need to withdraw (retract) the axis.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

**Coordinates of the plane.**

Z Z coordinate of the work plane.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

**Total depth.**

P Total depth.

**Safety distance.**

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dz X safety distance referred to the starting point.

**Angular position of the machining operations.** $\omega$  Angular position of the spindle (in degrees) for the first machining operation. $\Delta\omega$  Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.N $\omega$  Total number of operations.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
2D profile pocket cycle in the XC/XY plane.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)



## Define a profile.

This icon lets define the profile using the profile editor; it may be an existing profile or a new one.

### Profile definition.

P Name of the program with the profile (between 0 and 999).

- Edit a new program with the profile. To edit a new program, write the program number (between 0 and 999) and press [RECALL]. The CNC shows the profile editor.
- To select a known program, key in the program number and press [ENTER].
- To select a program from the list, use the following keys.



Expand the program list.



Moves the cursor through the list of programs. To select a program, position the cursor on the desired program and press [ENTER].



Exit the list without selecting any program.

### Modify an existing profile.

To edit an existing profile program, key in the program number and press [RECALL]. The profile editor of the CNC will show the selected profile, from where the following operations may be carried out.

- Add, delete or modify elements of the profile.
- Add, delete or modify chamfers, roundings, etc.
- Add new profiles to the program.

## Spindle parameters.

### Live tool turning direction.



Live tool clockwise.



Live tool counter-clockwise.

### Coolant status.



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

### Penetration feedrate.

Fz Penetration feedrate.

## Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the roughing operation.



Do not run the roughing operation.

7.

CANNED CYCLES OF THE EDITOR. PROFILES.

2D profile pocket cycle in the XC/XY plane.

**FAGOR**

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Maximum penetration step.**

I Maximum penetration step.

- If the parameter is positive, the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If the parameter is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

**Sideways penetration angle.** $\beta$  Sideways penetration angle.**Milling pass or width.** $\Delta$  Milling pass or width.

The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

**Machining feedrate.**

F Machining feedrate.

**Live-tool data.**

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.



Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Tool data.**

T Tool. If it is set to 0, the cycle does not execute the roughing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**Machining parameters. Finishing.**

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.



Run the finishing operation.



Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes ( $\delta$ ,  $\delta l$ ) ( $\delta$ ,  $\delta l$ ).

7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
2D profile pocket cycle in the XC/XY plane.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Sideways penetration angle.**

$\theta$  Sideways penetration angle.

**Finishing stocks.**


$\delta$  Side finishing stock.


$\delta x$  Bottom finishing stock. If it is set to 0, the cycle does not execute the finishing operation at the bottom.

**Machining feedrate.**

F Machining feedrate.

**Live-tool data.**

 Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

 Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Number of side penetration passes (steps) for the finishing operation.**


N Number of side penetration passes (steps) for the finishing operation. If defined with a value other than 0, the cycle executes N finishing passes, even if no side finishing stock has been programmed. If it is set to 0, the cycle does not execute the side finishing operation.


**Tool data.**

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

 Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

 Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**7.**

**CANNED CYCLES OF THE EDITOR. PROFILES.**  
2D profile pocket cycle in the XC/XY plane.

## 7.12.1 Basic operation.

The machining steps in these cycles are the following:

- 1 If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
- 2 The live tool starts with the speed selected and in the indicated direction.
- 3 The tool approaches in rapid up to the starting point, keeping the selected safety distance according to the Z axis.
- 4 The spindle orients to the indicated "C" position.
- 5 Roughing operation (if it has been programmed).

The cycle runs successive passes, up to a distance from the final profile equal to the finishing stock. This operation is done with the conditions set for the roughing operation.

- 6 Finishing operation (if it has been programmed).

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

The cycle eliminates the finishing stocks. This operation is done with the conditions set for the finishing operation.

The cycle offers two finishing stocks; at the bottom and on the walls. If both stocks are programmed ( $\delta$ ,  $\delta x$ ), the cycle executes the finishing operation at the bottom first and then on the walls. The finishing on the walls is carried out in as many penetration passes as indicated by parameter N. If there are no side finishing stocks, but parameter N is other than 0, the cycle executes N finishing passes.

- 7 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
- 8 The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

### Considerations.

#### Define a tool as T0.

If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.

If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the roughing operation, the tool will move to the safety point (starting point plus the safety distance).

#### Roughing and finishing in multiple pockets.

If the roughing tool and the finishing tool are different, the cycle executes first all the roughing operations and then all the finishing operations.

If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- If no M function has been defined, or they are the same for both operations, the cycle executes the whole pocket (roughing and finishing) before going on to the next one.
- If the M functions assigned to each operation are different, the cycle executes first all the roughing operations and then all the finishing operations.

# 7.

CANNED CYCLES OF THE EDITOR. PROFILES.  
2D profile pocket cycle in the XC/XY plane.

# CANNED CYCLES OF THE EDITOR. LONGITUDINAL MACHINING.

# 8

When pressing the softkey, the editor shows the cycle used last. When pressing the same softkey again, the menu shows all the cycles of the group.



- Center punching.
- Drilling.
- Tapping.
- Multiple center-punching.
- Multiple drilling.
- Multiple tapping.
- Multiple reaming.
- Multiple boring.
- Multiple bore milling.
- Multiple thread milling.
- Multiple slot milling.



When these cycles are executed in planes with C axis or Y axis, multiple machining may be added to them so the cycle is repeated in several points. The available multiple machining operations are the following. See chapter "[10 Canned cycles of the editor. Multiple positioning.](#)".

- Points in line.
- Points in arc.
- Points in rectangle.
- Points in grid.
- Random points.

To associate multiple machining to a cycle, first select and define a machining cycle from those allowed. Then, without quitting the editing of the cycle, press the softkey associated with multiple machining and select one of them.

## Selecting the positioning rotary axis for multiple machining.

The cycle uses the following criteria for selecting the positioning rotary axis for multiple machining.

- 1 The C axis that was active before executing the cycle, programmed with #CAX if it is a spindle or #FACE or #CYL if it is an axis.
- 2 The master spindle before the cycle, if its machine parameter CAXIS=YES.
- 3 The first non-master spindle whose machine parameter CAXIS=YES.
- 4 The first axis whose machine parameter CAXIS=YES.
- 5 M19 with master spindle.



FAGOR AUTOMATION

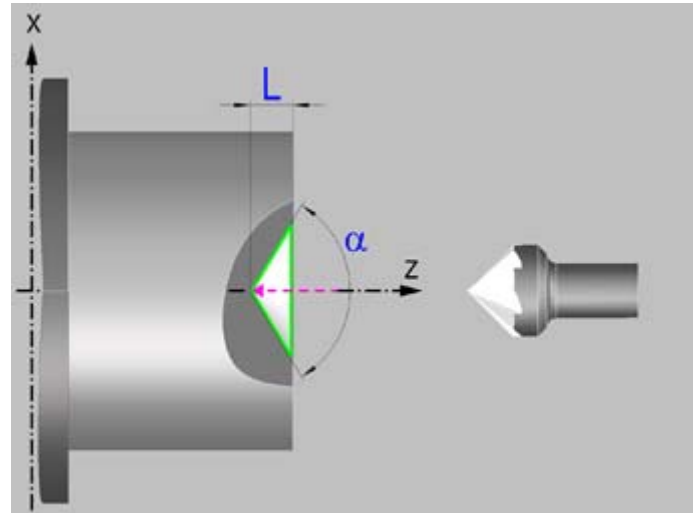
CNC 8060  
CNC 8065

(REF: 1709)

## 8.1 Center punching.

8.

CANNED CYCLES OF THE EDITOR: LONGITUDINAL  
Center punching.



### Geometric parameters.

#### Coordinates of the starting point

**X** X coordinate of the starting point; if not programmed, the cycle will assume a 0 value. The coordinate must be given in the active units, radius or diameter.

**Z** Z coordinate of the starting point.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

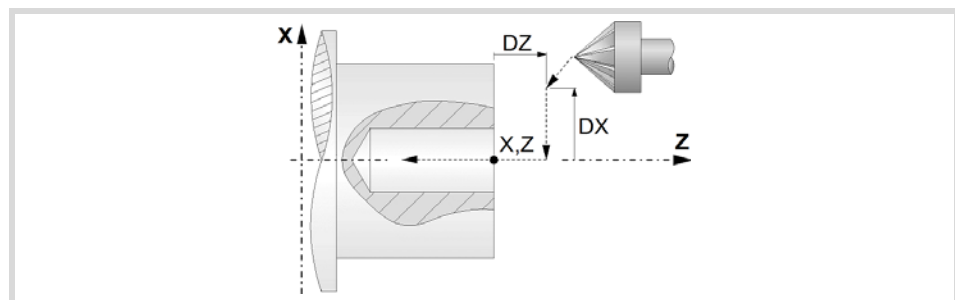
Although the center punching is normally done in the rotation center, the CNC enables X to be defined with a value other than 0 and make grooves in the front face of the part. For center punching, X must be 0.

#### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

**Dx, Dz** Safety distance referred to the starting point.

The value of the safety distance on X is always defined in radius. The value of the safety distance in Z is always assumed with positive values.



#### Programming the center punching operation.



Programming the depth of the center punching operation.



Programming the angle and the diameter of the center punching operation.

(REF: 1709)

**FAGOR**   
FAGOR AUTOMATION

CNC 8060  
CNC 8065

- L Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.
- $\alpha$  Center-punching angle.
- $\phi$  Center-punching diameter.

## Spindle parameters.

### Spindle turning direction

The cycle lets select the spindle turning direction.



Spindle clockwise.



Spindle counterclockwise.

### Coolant status.



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

### Maximum spindle speed.

Smax Maximum spindle turning speed; if not programmed, the cycle will maintain the maximum active speed.

### Spindle range (gear).

GEAR Spindle gear; if not programmed, the cycle assumes a value of 0. If defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

## Machining parameters.

### Dwell at the bottom.

t The waiting time in seconds, after center punching, until the withdrawal movement starts.

### Penetration feedrate.

F Penetration feedrate.

### Spindle speed.

S Spindle speed.

### Tool data.

T Tool.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

8.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL  
Center punching.

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Machining M functions.**

Activating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivating the execution of M functions before machining.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**8.**

**CANNED CYCLES OF THE EDITOR: LONGITUDINAL**  
Center punching.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)



### 8.1.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 If the operation was programmed with another tool, the CNC will make a tool change moving to the change point if the machine so requires.
- 2 The spindle starts with the speed selected and in the direction stated.
- 3 The tool approaches in rapid up to the starting point, keeping the safety distance selected according to axes X and Z (safety point).
- 4 Fast movement to the center-punching position in X (approach point).
- 5 Center punching of the part in work feedrate, until the programmed depth is reached.
- 6 Dwell at the bottom, if it has been programmed.
- 7 Fast backwards to the approach point.
- 8 Once the operation or cycle has ended, the tool will return to the safety point.
- 9 The CNC will not stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

8.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL  
Center punching.

**FAGOR** 

FAGOR AUTOMATION

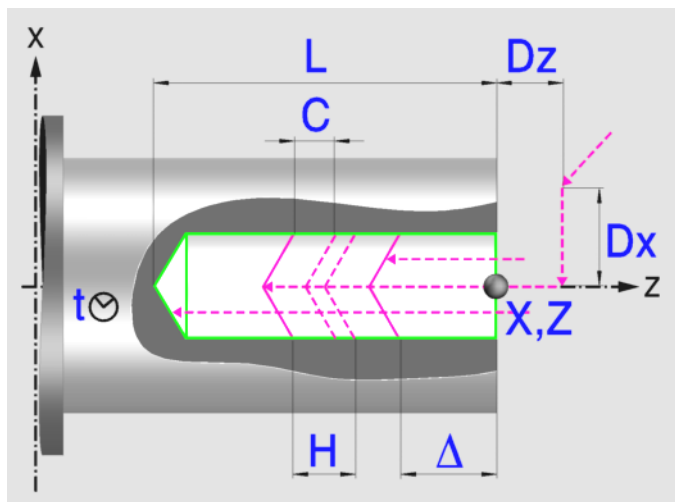
CNC 8060  
CNC 8065

(REF: 1709)

## 8.2 Drilling.

8.

CANNED CYCLES OF THE EDITOR: LONGITUDINAL Drilling.



### Geometric parameters.

#### Coordinates of the starting point

X X coordinate of the starting point; if not programmed, the cycle will assume a 0 value. The coordinate must be given in the active units, radius or diameter.

Z Z coordinate of the starting point.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

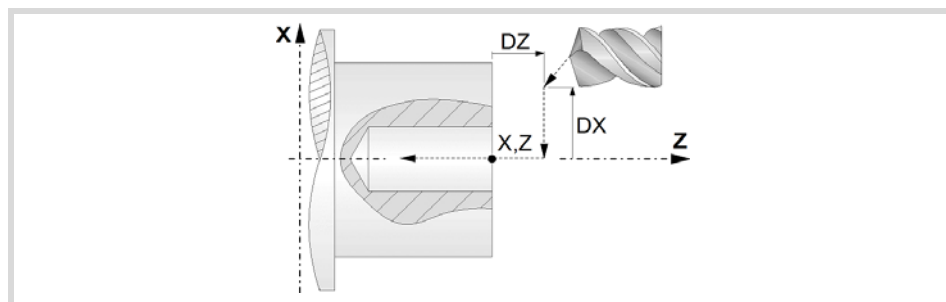
Although the drilling is normally done in the rotation center, the CNC enables X to be defined with a value other than 0 and make grooves in the front face of the part. For drilling, X must be 0.

#### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dx, Dz Safety distance referred to the starting point.

The value of the safety distance on X is always defined in radius. The value of the safety distance in Z is always assumed with positive values.



#### Drilling depth.

L Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.

## Spindle parameters.

### Spindle turning direction

The cycle lets select the spindle turning direction.



Spindle clockwise.



Spindle counterclockwise.

### Coolant status.



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

### Maximum spindle speed.

**Smax** Maximum spindle turning speed; if not programmed, the cycle will maintain the maximum active speed.

### Spindle range (gear).

**GEAR** Spindle gear; if not programmed, the cycle assumes a value of 0. If defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

## Machining parameters.

### Drilling step (peck); constant-peck or variable-peck drilling.

**Δ** Maximum drilling peck (step).

**KΔ** Reduction factor of the drilling peck; if not programmed, the cycle will assume a factor of 1.

**δ** Minimum drilling step (peck); constant-peck or variable-peck drilling.

The first drilling peck will be "Δ", the second one "KΔ D", the third one "KΔ (KΔ D)" and so on until reaching the minimum peck; in other words, from the second peck on, the new peck will be the result of multiplying the previous peck by KΔ.

If the minimum drilling peck is greater than the maximum peck ( $\delta > \Delta$ ), the cycle carries out a drilling at a constant peck equal to the minimum peck "δ". If the minimum drilling peck is smaller than the maximum peck ( $\delta < \Delta$ ), the cycle carries out a drilling at a variable peck.

### Withdrawal and approach distance in each penetration.

**H** Withdrawal distance after each peck; if not programmed, the cycle will assume a value of 1 mm. When programming H=0, it will withdraw up to the Z coordinate of the safety point.

**C** Approach distance in each penetration. If parameter C is not assigned a value or 0 is assigned, it assumes 1 mm by default.

### Dwell at the bottom.

**t** The waiting time in seconds, after drilling, until the withdrawal movement starts.

### Penetration feedrate.

**F** Penetration feedrate.

8.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL Drilling.

**FAGOR** 

FAGOR AUTOMATION

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

(REF: 1709)

## 8.

## CANNED CYCLES OF THE EDITOR: LONGITUDINAL Drilling.

**Spindle speed.**

S Spindle speed.

**Tool data.**

T Tool.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

Activating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivating the execution of M functions before machining.

The editor will only show this option if the user has configured the editor to allow programming M functions.

## 8.2.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 If the operation was programmed with another tool, the CNC will make a tool change moving to the change point if the machine so requires.
- 2 The spindle starts with the speed selected and in the direction stated.
- 3 The tool approaches in rapid up to the starting point, keeping the safety distance selected according to axes X and Z (safety point).
- 4 Fast movement to the drilling position in X (approach point).
- 5 Drilling loop. Depending on the minimum and maximum pecks defined, the cycle will carry out a drilling operation at a constant peck or at a variable peck.

If the minimum drilling peck is greater than the maximum peck ( $\delta > \Delta$ ), the cycle carries out a drilling at a constant peck equal to the minimum peck " $\delta$ ". If the minimum drilling peck is smaller than the maximum peck ( $\delta < \Delta$ ), the cycle carries out a drilling at a variable peck.

In a drilling operation at variable peck, parameter  $\Delta$  defines the drilling peck and  $K\Delta$  the reduction factor for this peck. The first drilling peck will be " $\Delta$ ", the second one " $K\Delta$  D", the third one " $K\Delta (K\Delta D)$ " and so on until reaching the minimum peck; in other words, from the second peck on, the new peck will be the result of multiplying the previous peck by  $K\Delta$ .

In either case, the cycle repeats the following steps until reaching the programmed depth.

First, rapid approach to the distance indicated by parameter C with respect to the previous penetration. Then, drilling down to the next penetration. And last, rapid withdrawal to the distance indicated by parameter H. When programming  $H=0$ , it will withdraw up to the Z coordinate of the safety point.

- 6 Dwell at the bottom, if it has been programmed.
- 7 Fast backwards to the approach point.
- 8 Once the operation or cycle has ended, the tool will return to the safety point.
- 9 The CNC will not stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

8.

Drilling.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL

**FAGOR** 

FAGOR AUTOMATION

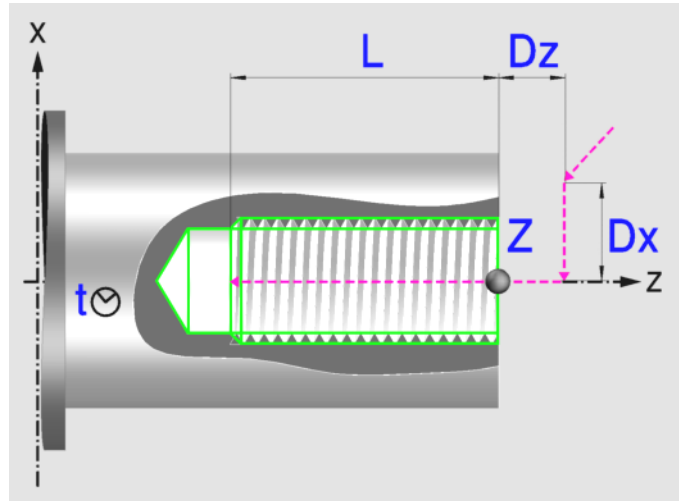
CNC 8060  
CNC 8065

(REF: 1709)

### 8.3 Tapping.

# 8.

CANNED CYCLES OF THE EDITOR: LONGITUDINAL Tapping.



#### Geometric parameters.

##### Coordinates of the starting point

Z Z coordinate of the starting point.

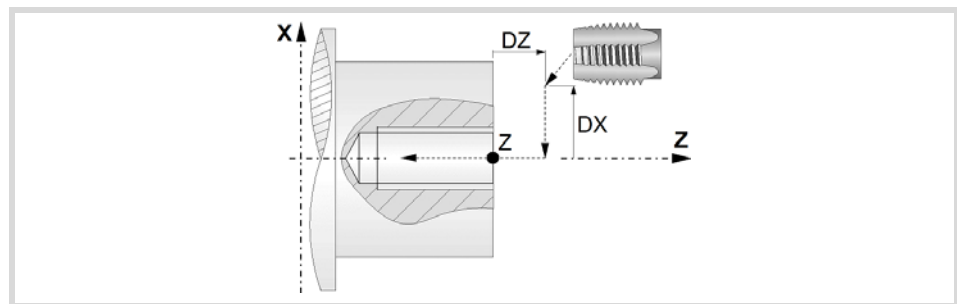
To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

##### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dx, Dz Safety distance referred to the starting point.

The value of the safety distance on X is always defined in radius. The value of the safety distance in Z is always assumed with positive values.



##### Tapping depth.

L Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.

##### Tapping type.



Tapping with a clutch.



Rigid tapping. The spindle must have a servo-motor-drive system and an encoder.

## Spindle parameters.

### Spindle turning direction

The cycle lets select the spindle turning direction.



Spindle clockwise.



Spindle counterclockwise.

### Coolant status.



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

### Spindle range (gear).

**GEAR** Spindle gear; if not programmed, the cycle assumes a value of 0. If defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

## Machining parameters.

### Define the thread.



Define the thread with the feedrate and the turning direction.



Define the thread with the pitch and the turning direction.

**F** Penetration feedrate.

**P** Thread pitch.

### Dwell at the bottom.

**t** The waiting time in seconds, after tapping, until the withdrawal movement starts.

### Spindle speed.

**S** Spindle speed.

### Tool data.

**T** Tool.

**D** Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

### Machining M functions.



Activating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivating the execution of M functions before machining.

The editor will only show this option if the user has configured the editor to allow programming M functions.

8.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL Tapping.

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

### 8.3.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 If the operation was programmed with another tool, the CNC will make a tool change moving to the change point if the machine so requires.
- 2 The spindle starts with the speed selected and in the direction stated.
- 3 The tool approaches in rapid up to the starting point, keeping the safety distance selected according to axes X and Z (safety point).
- 4 Fast movement to the tapping position in X (approach point).
- 5 Tapping of the part in work feedrate, until the programmed depth is reached.
- 6 The spindle reverses turning direction.

If a dwell time at the bottom has been defined, the spindle stops and, after this time, the spindle starts in the other direction and withdraws at working feedrate at the approach point.

- 7 Once the operation or cycle has ended, the tool will return to the safety point.
- 8 If it is rigid tapping, the CNC will stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

When tapping with a clutch, the CNC will not stop the spindle.

When tapping with a clutch and rigid tapping, general logic output "TAPPING" stays active while executing the cycle. General logic output "RIGID" stays active while executing the rigid tapping.

# 8.

CANNED CYCLES OF THE EDITOR: LONGITUDINAL Tapping.



FAGOR AUTOMATION

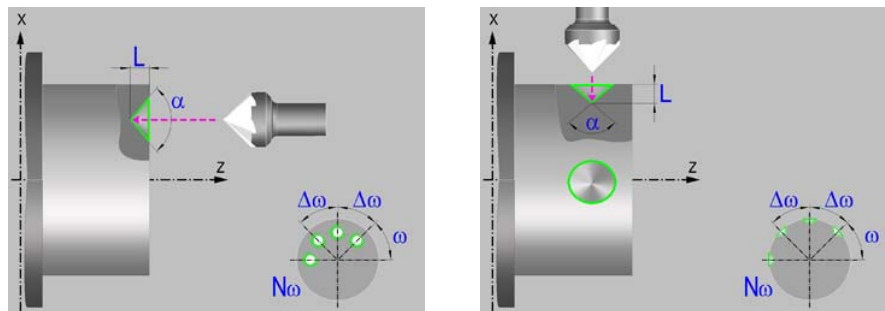
CNC 8060  
CNC 8065

(REF: 1709)



## 8.4 Multiple center-punching.

The cycle allows multiple center punching on the side of the part or on its face.



### Geometric parameters.

#### Machining on the face or side of the part.



Machining on the face of the part.



Machining on the side of the part.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Work plane.



Positioning on the XZ axes.



Positioning on the C axis.



Positioning on the Y axis.

#### Coordinates of the profile's starting and final points.

**X** X coordinate of the initial point of each machining operation. The coordinate must be given in the active units, radius or diameter.

**Z** Z coordinate of the initial point of each machining operation.

**Y** Y coordinate of the first positioning movement.

**Yf** Y coordinate for the withdrawal movement after the cycle ends; if not programmed, there is no Y axis withdrawal. If the cycle is machining in multiple mode, the Y withdrawal point is only applied to the last cycle.

This parameter lets withdraw the tool to the right position in order to be able to continue with the turning operations that do not use the Y axis. If the next cycle also uses the Y axis, there is no need to withdraw (retract) the axis.

**C** C coordinate of the first positioning movement.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

**Dx, Dz** Safety distance referred to the starting point.

8.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL  
Multiple center-punching.

**FAGOR** 

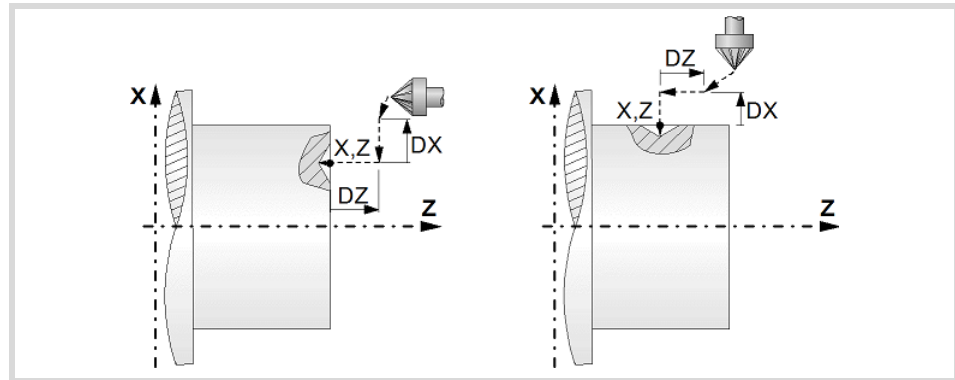
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

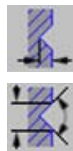
# 8.

**CANNED CYCLES OF THE EDITOR: LONGITUDINAL**  
Multiple center-punching.



- Machining on the face of the part. The value of the safety distance on X is always defined in radius and it may be either positive or negative. The value of the safety distance in Z is always assumed with positive values.
- Machining on the side of the part. The value of the safety distance on X is always defined in radius and it is always assumed with positive values. The value of the safety distance in Z may be either negative or positive.

**Programming the center punching operation.**



Programming the depth of the center punching operation.

Programming the angle and the diameter of the center punching operation.

L Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.

$\alpha$  Center-punching angle.

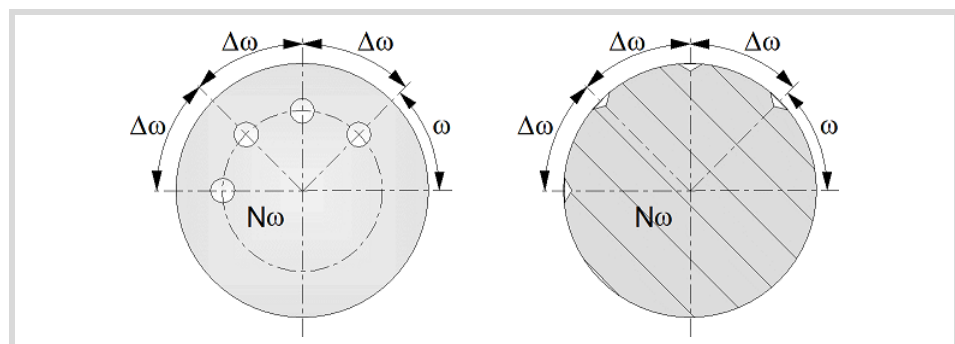
$\phi$  Center-punching diameter.

**Angular position of the machining operations.**

$\omega$  Angular position of the spindle (in degrees) for the first machining operation.

$\Delta\omega$  Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

$N\omega$  Total number of operations.



**Live-tool parameters.**

**Live tool turning direction.**



Live tool clockwise.

Live tool counter-clockwise.

**Coolant status.**



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

**Machining parameters.**

**Dwell at the bottom.**

t The waiting time in seconds, after center punching, until the withdrawal movement starts.

**Penetration feedrate.**

F Penetration feedrate.

**Live-tool data.**



Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.



Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Tool data.**

T Tool.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**



Activating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivating the execution of M functions before machining.

The editor will only show this option if the user has configured the editor to allow programming M functions.

### 8.4.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 If the operation was programmed with another tool, the CNC will make a tool change moving to the change point if the machine so requires.
- 2 The cycle turns the live tool at the indicated rpm.
- 3 The cycle orients the spindle to the angular position corresponding to the initial machining operation.
- 4 The tool approaches in rapid up to the starting point, keeping the safety distance selected according to axes X and Z (safety point).
- 5 Rapid movement to the machining position, in X for the face and in Z for the side of the part (approach point).
- 6 Center punching of the part in work feedrate, until the programmed depth is reached.
- 7 Dwell at the bottom, if it has been programmed.
- 8 Fast backwards to the approach point.
- 9 Depending on the number of machining operations programmed, the spindle orients to the second point and performs a new machining operation as described in point 6.
- 10 Once the operation or cycle has ended, the tool will return to the safety point.
- 11 The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

# 8.

**CANNED CYCLES OF THE EDITOR: LONGITUDINAL**  
Multiple center-punching.



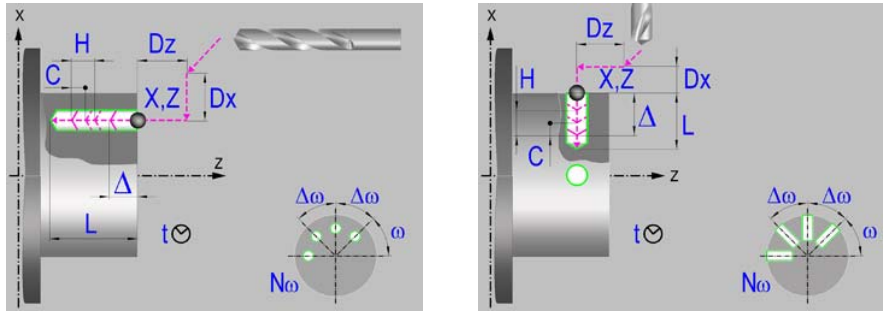
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 8.5 Multiple drilling.

The cycle allows multiple drilling on the side of the part or on its face.



### Geometric parameters.

#### Machining on the face or side of the part.



Machining on the face of the part.



Machining on the side of the part.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Work plane.



Positioning on the XZ axes.



Positioning on the C axis.



Positioning on the Y axis.

#### Coordinates of the profile's starting and final points.

**X** X coordinate of the initial point of each machining operation. The coordinate must be given in the active units, radius or diameter.

**Z** Z coordinate of the initial point of each machining operation.

**Y** Y coordinate of the first positioning movement.

**Yf** Y coordinate for the withdrawal movement after the cycle ends; if not programmed, there is no Y axis withdrawal. If the cycle is machining in multiple mode, the Y withdrawal point is only applied to the last cycle.

This parameter lets withdraw the tool to the right position in order to be able to continue with the turning operations that do not use the Y axis. If the next cycle also uses the Y axis, there is no need to withdraw (retract) the axis.

**C** C coordinate of the first positioning movement.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

**Dx, Dz** Safety distance referred to the starting point.



CANNED CYCLES OF THE EDITOR. LONGITUDINAL Multiple drilling.

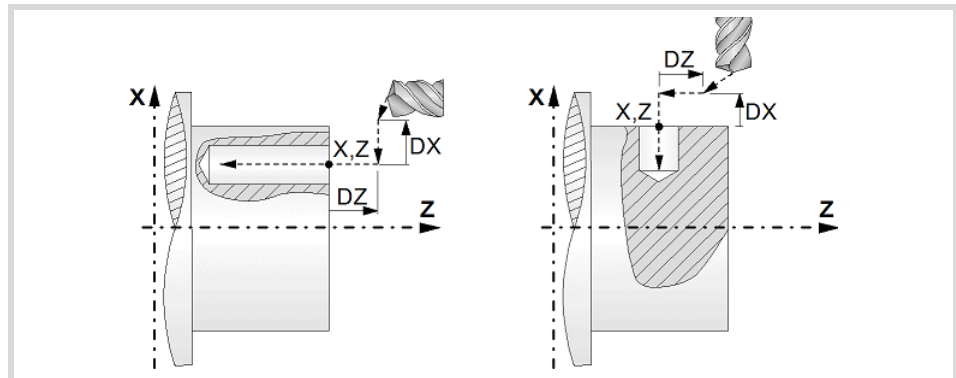


FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

- Machining on the face of the part. The value of the safety distance on X is always defined in radius and it may be either positive or negative. The value of the safety distance in Z is always assumed with positive values.
- Machining on the side of the part. The value of the safety distance on X is always defined in radius and it is always assumed with positive values. The value of the safety distance in Z may be either negative or positive.

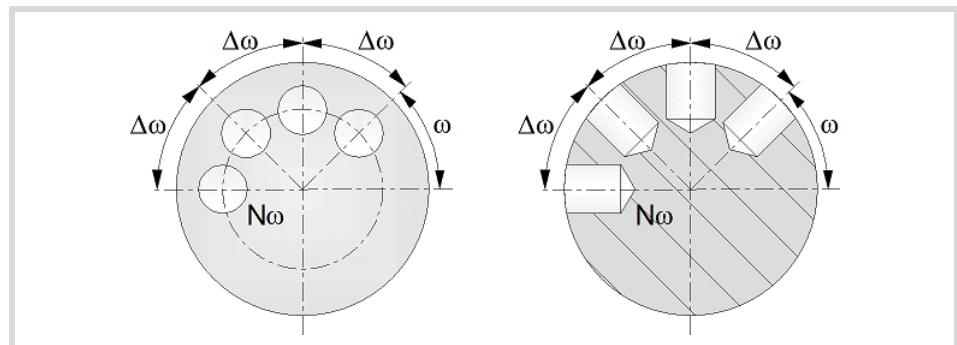


**Programming of the drilling operation.**

L Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.

**Angular position of the machining operations.**

- $\omega$  Angular position of the spindle (in degrees) for the first machining operation.
- $\Delta\omega$  Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.
- $N\omega$  Total number of operations.



**Live-tool parameters.**

**Live tool turning direction.**



Live tool clockwise.



Live tool counter-clockwise.

**8.**

CANNED CYCLES OF THE EDITOR: LONGITUDINAL Multiple drilling.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Coolant status.**



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

**Machining parameters.**

**Drilling step (peck); constant-peck or variable-peck drilling.**

- $\Delta$  Maximum drilling peck (step).
- $K\Delta$  Reduction factor of the drilling peck; if not programmed, the cycle will assume a factor of 1.
- $\delta$  Minimum drilling step (peck); constant-peck or variable-peck drilling.

The first drilling peck will be " $\Delta$ ", the second one " $K\Delta$  D", the third one " $K\Delta (K\Delta D)$ " and so on until reaching the minimum peck; in other words, from the second peck on, the new peck will be the result of multiplying the previous peck by  $K\Delta$ .

If the minimum drilling peck is greater than the maximum peck ( $\delta > \Delta$ ), the cycle carries out a drilling at a constant peck equal to the minimum peck " $\delta$ ". If the minimum drilling peck is smaller than the maximum peck ( $\delta < \Delta$ ), the cycle carries out a drilling at a variable peck.

**Withdrawal and approach distance in each penetration.**

- H Withdrawal distance after each peck; if not programmed, the cycle will assume a value of 1 mm. When programming  $H=0$ , it will withdraw up to the Z coordinate of the safety point.
- C Approach distance in each penetration. If parameter C is not assigned a value or 0 is assigned, it assumes 1 mm by default.



**Dwell at the bottom.**

- t The waiting time in seconds, after drilling, until the withdrawal movement starts.

**Penetration feedrate.**

- F Penetration feedrate.

**Live-tool data.**

-  Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.
-  Live tool data programmed. The cycle shows the data related to the live tool.
- S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Tool data.**

- T Tool.
- D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

8.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL Multiple drilling.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Machining M functions.**

Activating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivating the execution of M functions before machining.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**8.**

**CANNED CYCLES OF THE EDITOR: LONGITUDINAL**  
Multiple drilling.



## 8.5.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 If the operation was programmed with another tool, the CNC will make a tool change moving to the change point if the machine so requires.
- 2 The cycle turns the live tool at the indicated rpm.
- 3 The cycle orients the spindle to the angular position corresponding to the initial machining operation.
- 4 The tool approaches in rapid up to the starting point, keeping the safety distance selected according to axes X and Z (safety point).
- 5 Rapid movement to the machining position, in X for the face and in Z for the side of the part (approach point).
- 6 Drilling loop. Depending on the minimum and maximum pecks defined, the cycle will carry out a drilling operation at a constant peck or at a variable peck.

If the minimum drilling peck is greater than the maximum peck ( $\delta > \Delta$ ), the cycle carries out a drilling at a constant peck equal to the minimum peck " $\delta$ ". If the minimum drilling peck is smaller than the maximum peck ( $\delta < \Delta$ ), the cycle carries out a drilling at a variable peck.

In a drilling operation at variable peck, parameter  $\Delta$  defines the drilling peck and  $K\Delta$  the reduction factor for this peck. The first drilling peck will be " $\Delta$ ", the second one " $K\Delta$  D", the third one " $K\Delta (K\Delta D)$ " and so on until reaching the minimum peck; in other words, from the second peck on, the new peck will be the result of multiplying the previous peck by  $K\Delta$ .

In either case, the cycle repeats the following steps until reaching the programmed depth.

First, rapid approach to the distance indicated by parameter C with respect to the previous penetration. Then, drilling down to the next penetration. And last, rapid withdrawal to the distance indicated by parameter H. When programming  $H=0$ , it will withdraw up to the Z coordinate of the safety point.

- 7 Depending on the number of machining operations programmed, the spindle orients to the second point and performs a new machining operation as described in point 6.
- 8 Once the operation or cycle has ended, the tool will return to the safety point.
- 9 The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

8.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL  
Multiple drilling.

**FAGOR** 

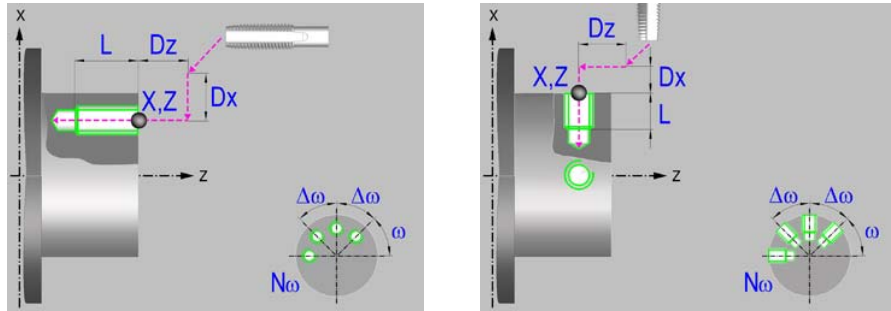
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 8.6 Multiple tapping.

The cycle allows multiple tapping on the side of the part or on its face.



### Geometric parameters.

#### Machining on the face or side of the part.



Machining on the face of the part.



Machining on the side of the part.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Work plane.



Positioning on the XZ axes.



Positioning on the C axis.



Positioning on the Y axis.

#### Coordinates of the profile's starting and final points.

**X** X coordinate of the initial point of each machining operation. The coordinate must be given in the active units, radius or diameter.

**Z** Z coordinate of the initial point of each machining operation.

**Y** Y coordinate of the first positioning movement.

**Yf** Y coordinate for the withdrawal movement after the cycle ends; if not programmed, there is no Y axis withdrawal. If the cycle is machining in multiple mode, the Y withdrawal point is only applied to the last cycle.

This parameter lets withdraw the tool to the right position in order to be able to continue with the turning operations that do not use the Y axis. If the next cycle also uses the Y axis, there is no need to withdraw (retract) the axis.

**C** C coordinate of the first positioning movement.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Safety distance.

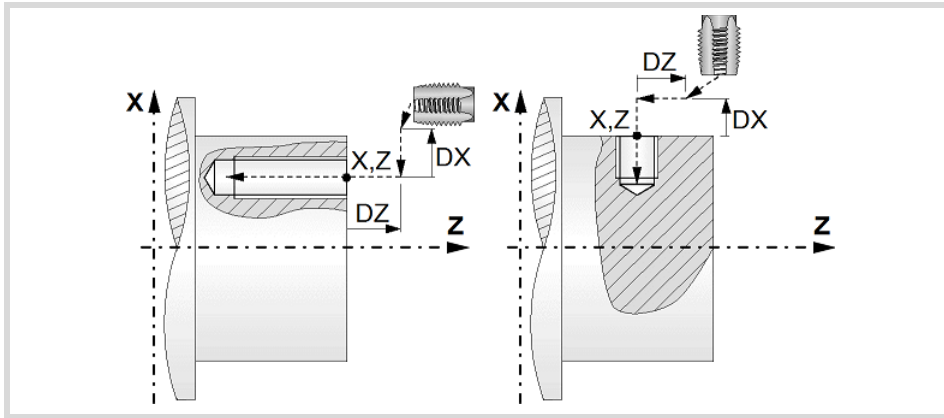
In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

**Dx, Dz** Safety distance referred to the starting point.

# 8.

CANNED CYCLES OF THE EDITOR: LONGITUDINAL  
Multiple tapping.

- Machining on the face of the part. The value of the safety distance on X is always defined in radius and it may be either positive or negative. The value of the safety distance in Z is always assumed with positive values.
- Machining on the side of the part. The value of the safety distance on X is always defined in radius and it is always assumed with positive values. The value of the safety distance in Z may be either negative or positive.



**Tapping type.**



Tapping with a clutch.



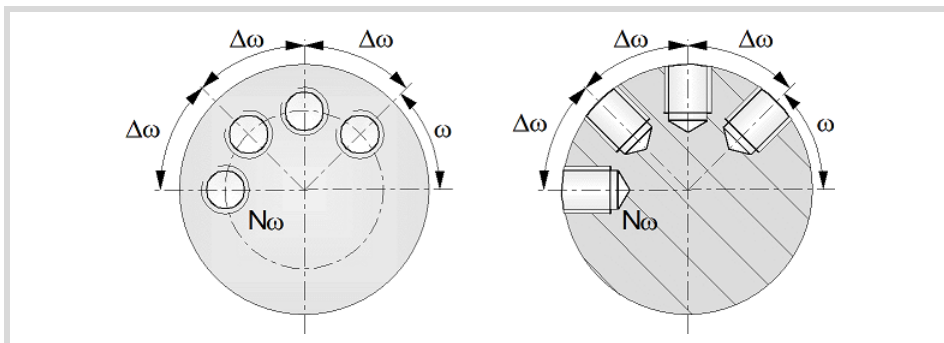
Rigid tapping. The spindle must have a servo-motor-drive system and an encoder.

**Programming of the tapping operation.**

L Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.

**Angular position of the machining operations.**

- $\omega$  Angular position of the spindle (in degrees) for the first machining operation.
- $\Delta\omega$  Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.
- $N\omega$  Total number of operations.



**Live-tool parameters.**

**Live tool turning direction.**



Live tool clockwise.



Live tool counter-clockwise.



CANNED CYCLES OF THE EDITOR. LONGITUDINAL Multiple tapping.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Coolant status.**

Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

8.

CANNED CYCLES OF THE EDITOR: LONGITUDINAL  
Multiple tapping.

**Machining parameters.****Define the thread.**

Define the thread with the feedrate.



Define the thread with the pitch.

F Penetration feedrate.

P Thread pitch.

**Dwell at the bottom.**

t The waiting time in seconds, after tapping, until the withdrawal movement starts.

**Live-tool data.**

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Tool data.**

T Tool.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

Activating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivating the execution of M functions before machining.

The editor will only show this option if the user has configured the editor to allow programming M functions.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

### 8.6.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 If the spindle is working in open loop (RPM or CSS mode) the CNC stops the spindle and performs a home search on the spindle (I0).
- 2 If the operation was programmed with another tool, the CNC will make a tool change moving to the change point if the machine so requires.
- 3 The cycle turns the live tool at the indicated rpm.
- 4 The cycle orients the spindle to the angular position corresponding to the initial machining operation.
- 5 The tool approaches in rapid up to the starting point, keeping the safety distance selected according to axes X and Z (safety point).
- 6 Rapid movement to the machining position, in X for the face and in Z for the side of the part (approach point).
- 7 Tapping of the part in work feedrate, until the programmed depth is reached.
- 8 Reversal of the turning direction of the live tool.
- 9 Fast backwards to the approach point.
- 10 Depending on the number of machining operations programmed, the spindle orients to the second point and performs a new machining operation as described in points 7, 8 and 9.
- 11 Once the operation or cycle has ended, the tool will return to the safety point.
- 12 The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

8.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL

Multiple tapping.

**FAGOR** 

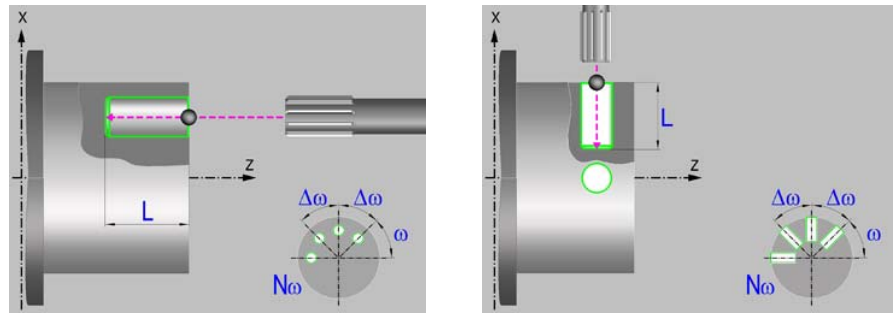
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 8.7 Multiple reaming.

The cycle allows multiple reaming on the side of the part or on its face.



### Geometric parameters.

#### Machining on the face or side of the part.



Machining on the face of the part.



Machining on the side of the part.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Work plane.



Positioning on the XZ axes.



Positioning on the C axis.



Positioning on the Y axis.

#### Coordinates of the profile's starting and final points.

**X** X coordinate of the initial point of each machining operation. The coordinate must be given in the active units, radius or diameter.

**Z** Z coordinate of the initial point of each machining operation.

**Y** Y coordinate of the first positioning movement.

**Yf** Y coordinate for the withdrawal movement after the cycle ends; if not programmed, there is no Y axis withdrawal. If the cycle is machining in multiple mode, the Y withdrawal point is only applied to the last cycle.

This parameter lets withdraw the tool to the right position in order to be able to continue with the turning operations that do not use the Y axis. If the next cycle also uses the Y axis, there is no need to withdraw (retract) the axis.

**C** C coordinate of the first positioning movement.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Safety distance.

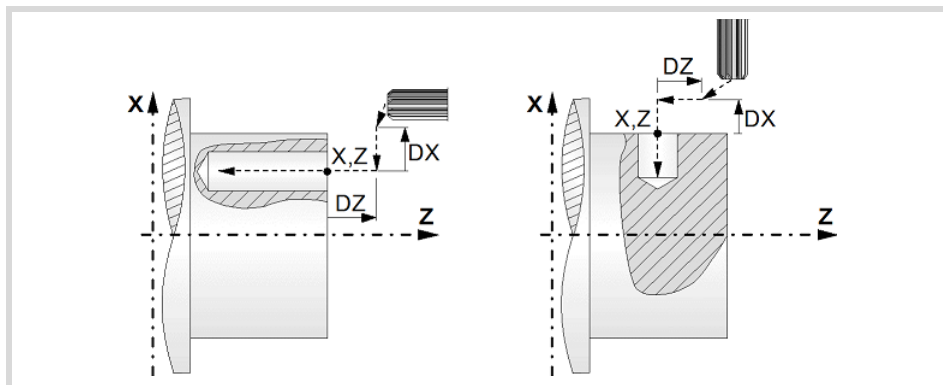
In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

**Dx, Dz** Safety distance referred to the starting point.

# 8.

CANNED CYCLES OF THE EDITOR: LONGITUDINAL  
Multiple reaming.

- Machining on the face of the part. The value of the safety distance on X is always defined in radius and it may be either positive or negative. The value of the safety distance in Z is always assumed with positive values.
- Machining on the side of the part. The value of the safety distance on X is always defined in radius and it is always assumed with positive values. The value of the safety distance in Z may be either negative or positive.

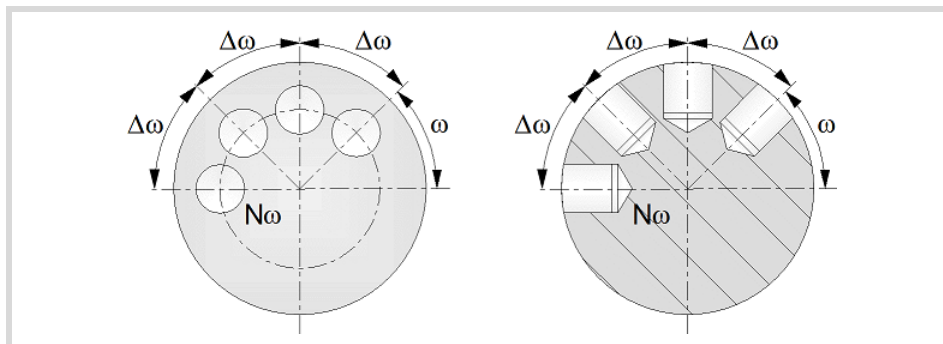


#### Programming the reaming operation.

L Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.

#### Angular position of the machining operations.

- $\omega$  Angular position of the spindle (in degrees) for the first machining operation.
- $\Delta\omega$  Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.
- $N\omega$  Total number of operations.



#### Live-tool parameters.

##### Live tool turning direction.



Live tool clockwise.



Live tool counter-clockwise.



CANNED CYCLES OF THE EDITOR. LONGITUDINAL Multiple reaming.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Coolant status.**

Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

8.

CANNED CYCLES OF THE EDITOR: LONGITUDINAL  
Multiple reaming.

**Machining parameters.****Dwell at the bottom.**

t The waiting time in seconds, after reaming, until the withdrawal movement starts.

**Penetration feedrate.**

F Penetration feedrate.

**Live-tool data.**

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.



Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Tool data.**

T Tool.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

Activating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivating the execution of M functions before machining.

The editor will only show this option if the user has configured the editor to allow programming M functions.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)



### 8.7.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 If the operation was programmed with another tool, the CNC will make a tool change moving to the change point if the machine so requires.
- 2 The cycle turns the live tool at the indicated rpm.
- 3 The cycle orients the spindle to the angular position corresponding to the initial machining operation.
- 4 The tool approaches in rapid up to the starting point, keeping the safety distance selected according to axes X and Z (safety point).
- 5 Rapid movement to the machining position, in X for the face and in Z for the side of the part (approach point).
- 6 Reaming of the part in work feedrate, until the programmed depth is reached.
- 7 Dwell at the bottom, if it has been programmed.
- 8 Fast backwards to the approach point.
- 9 Depending on the number of machining operations programmed, the spindle orients to the second point and performs a new machining operation as described in point 6.
- 10 Once the operation or cycle has ended, the tool will return to the safety point.
- 11 The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

8.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL

Multiple reaming.

**FAGOR** 

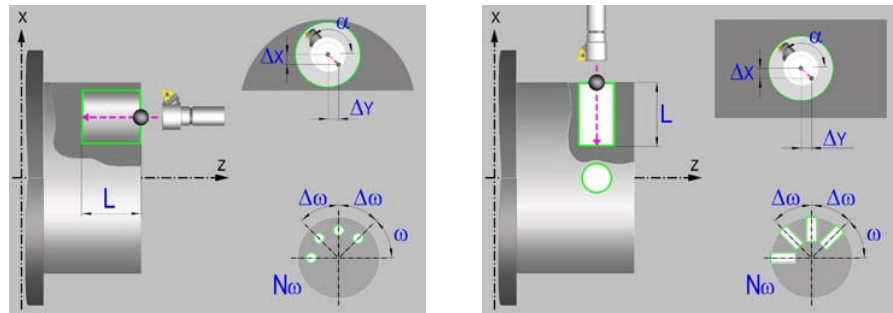
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 8.8 Multiple boring.

The cycle allows multiple boring on the side of the part or on its face.



### Geometric parameters.

#### Machining on the face or side of the part.



Machining on the face of the part.



Machining on the side of the part.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Work plane.



Positioning on the XZ axes.



Positioning on the C axis.



Positioning on the Y axis.

#### Coordinates of the profile's starting and final points.

**X** X coordinate of the initial point of each machining operation. The coordinate must be given in the active units, radius or diameter.

**Z** Z coordinate of the initial point of each machining operation.

**Y** Y coordinate of the first positioning movement.

**Yf** Y coordinate for the withdrawal movement after the cycle ends; if not programmed, there is no Y axis withdrawal. If the cycle is machining in multiple mode, the Y withdrawal point is only applied to the last cycle.

This parameter lets withdraw the tool to the right position in order to be able to continue with the turning operations that do not use the Y axis. If the next cycle also uses the Y axis, there is no need to withdraw (retract) the axis.

**C** C coordinate of the first positioning movement.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

**Dx, Dz** Safety distance referred to the starting point.

# 8.

CANNED CYCLES OF THE EDITOR: LONGITUDINAL  
Multiple boring.

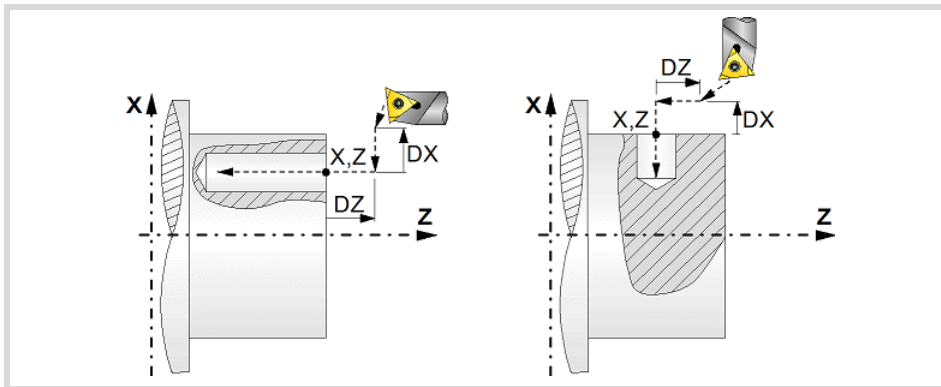


FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

- Machining on the face of the part. The value of the safety distance on X is always defined in radius and it may be either positive or negative. The value of the safety distance in Z is always assumed with positive values.
- Machining on the side of the part. The value of the safety distance on X is always defined in radius and it is always assumed with positive values. The value of the safety distance in Z may be either negative or positive.

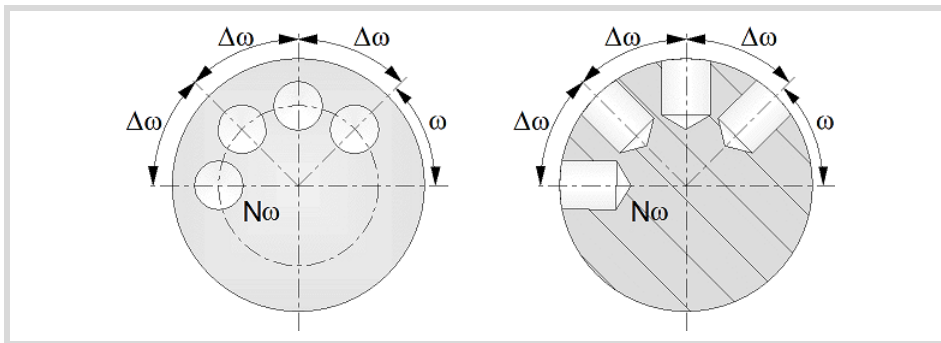


**Programming the boring operation.**

L Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.

**Angular position of the machining operations.**

- $\omega$  Angular position of the spindle (in degrees) for the first machining operation.
- $\Delta\omega$  Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.
- $N\omega$  Total number of operations.



**Live-tool parameters.**

**Live tool turning direction.**



Live tool clockwise.



Live tool counter-clockwise.



CANNED CYCLES OF THE EDITOR. LONGITUDINAL Multiple boring.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Coolant status.**



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

**8.**

CANNED CYCLES OF THE EDITOR: LONGITUDINAL Multiple boring.

**Machining parameters.**

**Type of withdrawal.**



Withdrawal at work feedrate while the spindle is turning.



Rapid withdrawal while the spindle is stopped and the tool is touching the surface.

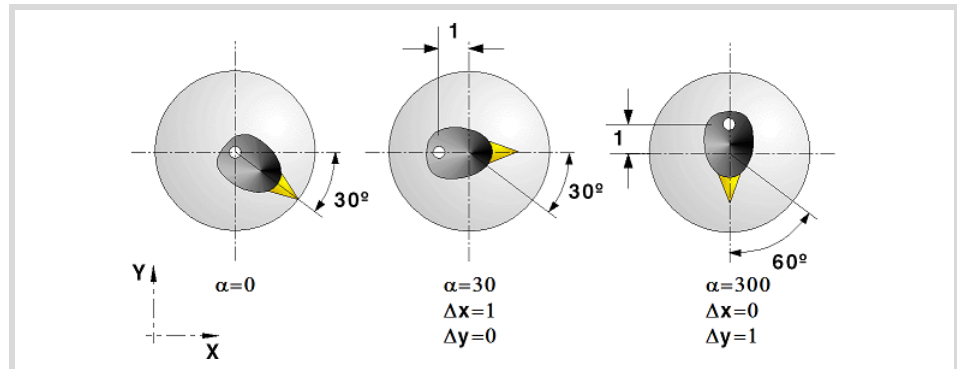


Rapid withdrawal with spindle orientation and the tool is not touching the surface. This type of withdrawal is not available for XZ positioning, nor for C axis positioning on the cylindrical side of the part.

$\alpha$  Angular position, in degrees, of the spindle of the live tool for the withdrawal.

$\Delta X$  X distance away from (off) the wall of the hole; if not programmed, the cycle will assume a 0 value.

$\Delta Y$  Y distance away from (off) the wall of the hole; if not programmed, the cycle will assume a 0 value.



**Dwell at the bottom.**

t The waiting time in seconds, after boring, until the withdrawal movement starts.

**Penetration feedrate.**

F Penetration feedrate.

**Live-tool data.**



Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.



Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Tool data.**

- T Tool.
- D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**



Activating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivating the execution of M functions before machining.

The editor will only show this option if the user has configured the editor to allow programming M functions.

8.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL.  
Multiple boring.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 8.8.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 If the operation was programmed with another tool, the CNC will make a tool change moving to the change point if the machine so requires.
- 2 The cycle turns the live tool at the indicated rpm.
- 3 The cycle orients the spindle to the angular position corresponding to the initial machining operation.
- 4 The tool approaches in rapid up to the starting point, keeping the safety distance selected according to axes X and Z (safety point).
- 5 Rapid movement to the machining position, in X for the face and in Z for the side of the part (approach point).
- 6 Boring of the part in work feedrate, until the programmed depth is reached.
- 7 Dwell at the bottom, if it has been programmed.
- 8 Tool withdrawal up to the approach point, according to the selected mode.



Withdrawal at work feedrate while the spindle is turning.



Rapid withdrawal while the spindle is stopped and the tool is touching the surface.



Rapid withdrawal with spindle orientation and the tool has been retracted away from the surface.

- 9 Depending on the number of machining operations programmed, the spindle orients to the second point and performs a new machining operation as described in points 6, 7 and 8.
- 10 Once the operation or cycle has ended, the tool will return to the safety point.
- 11 The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

# 8.

CANNED CYCLES OF THE EDITOR: LONGITUDINAL  
Multiple boring.



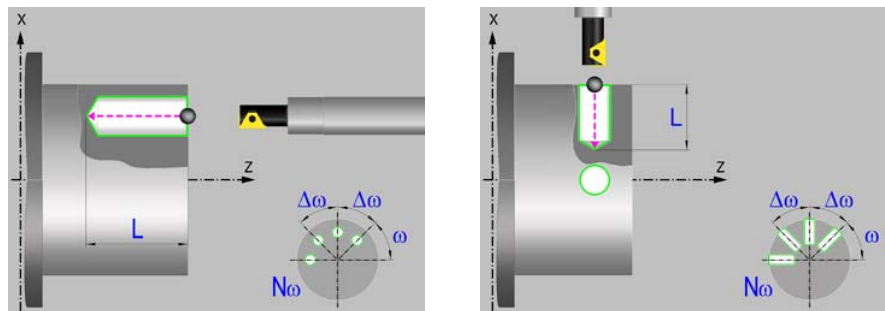
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 8.9 Multiple bore milling.

The cycle allows multiple bore milling on the side of the part or on its face.



### Geometric parameters.

#### Machining on the face or side of the part.



Machining on the face of the part.



Machining on the side of the part.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Coordinates of the starting point

- X X coordinate of the hole center of each machining operation. The coordinate must be given in the active units, radius or diameter.
- Z Z coordinate of the hole center of each machining operation.
- Y Y coordinate of the first positioning movement.
- Yf Y coordinate for the withdrawal movement after the cycle ends; if not programmed, there is no Y axis withdrawal. If the cycle is machining in multiple mode, the Y withdrawal point is only applied to the last cycle.

This parameter lets withdraw the tool to the right position in order to be able to continue with the turning operations that do not use the Y axis. If the next cycle also uses the Y axis, there is no need to withdraw (retract) the axis.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dx, Dz Safety distance referred to the starting point.

- Machining on the face of the part. The value of the safety distance on X is always defined in radius and it may be either positive or negative. The value of the safety distance in Z is always assumed with positive values.
- Machining on the side of the part. The value of the safety distance on X is always defined in radius and it is always assumed with positive values. The value of the safety distance in Z may be either negative or positive.

8.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL.  
Multiple bore milling.

**FAGOR** 

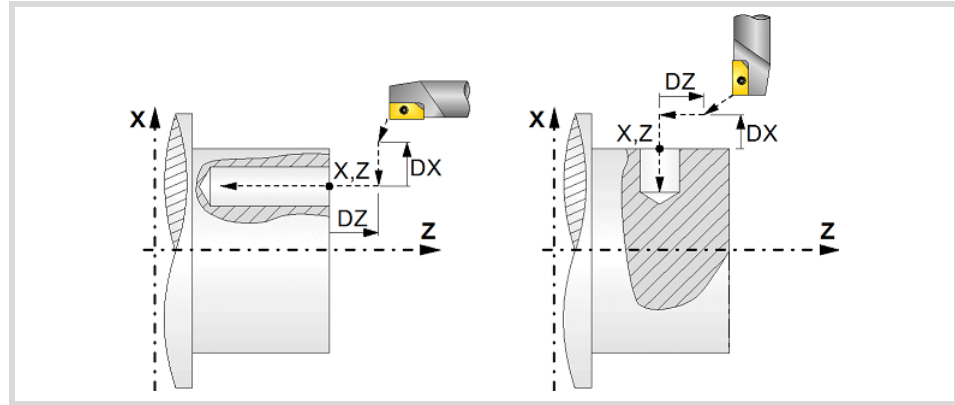
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

# 8.

**CANNED CYCLES OF THE EDITOR: LONGITUDINAL**  
Multiple bore milling.



**Programming the machining operation.**

L Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.

Ø Hole diameter.

Øk Pre-hole diameter.

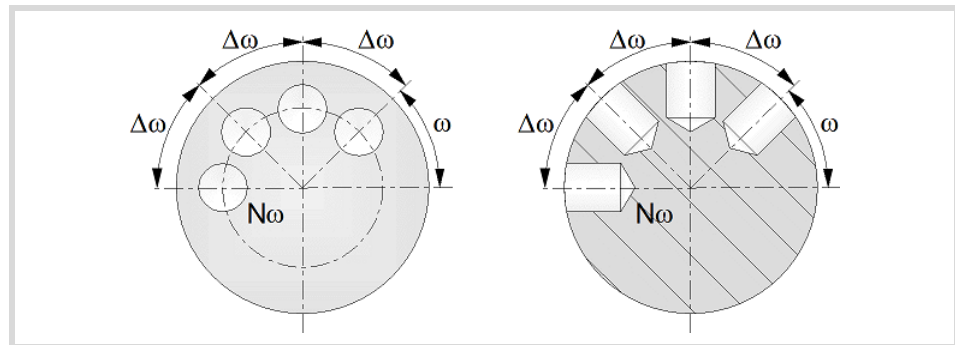
When starting from a hole that was machined earlier, parameter Øk defines the diameter of that hole; if not programmed or programmed with a 0 value, the cycle assumes that there is no previous hole.

**Angular position of the machining operations.**

ω Angular position of the spindle (in degrees) for the first machining operation.


Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.


Nω Total number of operations.



**Live-tool parameters.**

**Live tool turning direction.**

 Live tool clockwise.

 Live tool counter-clockwise.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)



**Coolant status.**



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

**Machining parameters.**

**Helical machining.**



Counterclockwise machining.



Clockwise machining.

B Helical drilling step.

**Bottom milling.**



Straight finishing. The cycle redoes the bottom of the hole (blind hole).



Helical finishing. The cycle does not redo the bottom of the hole (thru hole).

**Penetration feedrate.**

F Penetration feedrate.

**Live-tool data.**



Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.



Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Tool data.**

T Tool.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**



Activating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivating the execution of M functions before machining.

The editor will only show this option if the user has configured the editor to allow programming M functions.

8.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL Multiple bore milling.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 8.9.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 If the operation was programmed with another tool, the CNC will make a tool change moving to the change point if the machine so requires.
- 2 The cycle turns the live tool at the indicated rpm.
- 3 The cycle orients the spindle to the angular position corresponding to the initial machining operation.
- 4 The tool approaches in rapid up to the starting point, keeping the safety distance selected according to axes X and Z (safety point).
- 5 Rapid movement to the tangential entry point (approach point).
- 6 Tangential entry to the helical path of the drilling.
- 7 Helical movement, with the given pitch and in the given direction, down to the bottom of the hole.
- 8 Redo the bottom of the hole, if it has been programmed.
- 9 Tangential exit movement to the helical path of the drilling to the center of the hole.
- 10 Fast backwards to the approach point.
- 11 Depending on the number of machining operations programmed, the spindle orients to the second point and performs a new machining operation as described in points 6 to 10.
- 12 Once the operation or cycle has ended, the tool will return to the safety point.
- 13 The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

# 8.

**CANNED CYCLES OF THE EDITOR. LONGITUDINAL**  
Multiple bore milling.



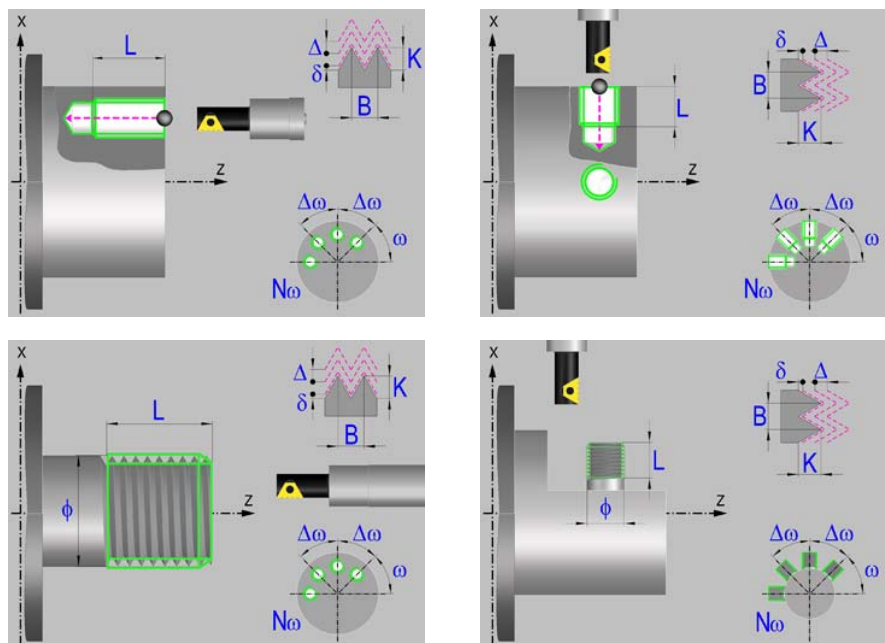
FAGOR AUTOMATION

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

(REF: 1709)

## 8.10 Multiple thread milling.

The cycle allows multiple thread milling on the side of the part or on its face.



### Geometric parameters.

#### Machining on the face or side of the part.



Machining on the face of the part.



Machining on the side of the part.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Inside or outside machining.



Inside machining.



Outside machining.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Coordinates of the profile's starting and final points.

- X X coordinate of the center of the hole or of the boss. The coordinate must be given in the active units, radius or diameter.
- Z Z coordinate of the center of the hole or of the boss.
- Y Y coordinate of the first positioning movement.
- Yf Y coordinate for the withdrawal movement after the cycle ends; if not programmed, there is no Y axis withdrawal. If the cycle is machining in multiple mode, the Y withdrawal point is only applied to the last cycle.

This parameter lets withdraw the tool to the right position in order to be able to continue with the turning operations that do not use the Y axis. If the next cycle also uses the Y axis, there is no need to withdraw (retract) the axis.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.



CANNED CYCLES OF THE EDITOR. LONGITUDINAL Multiple thread milling.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

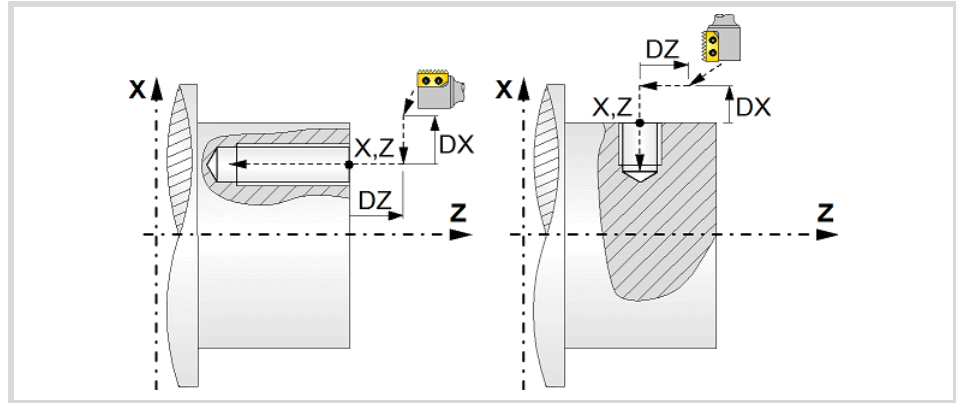
(REF: 1709)

**Safety distance.**

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

$Dx, Dz$  Safety distance referred to the starting point.

- Machining on the face of the part. The value of the safety distance on X is always defined in radius and it may be either positive or negative. The value of the safety distance in Z is always assumed with positive values.
- Machining on the side of the part. The value of the safety distance on X is always defined in radius and it is always assumed with positive values. The value of the safety distance in Z may be either negative or positive.



**Programming the machining operation.**

$L$  Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.

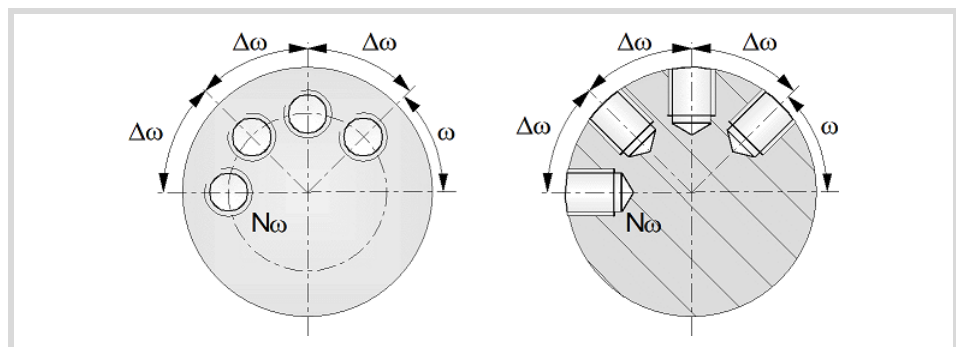
$\emptyset$  Nominal thread diameter.

**Angular position of the machining operations.**

$\omega$  Angular position of the spindle (in degrees) for the first machining operation.


$\Delta\omega$  Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.


$N\omega$  Total number of operations.



**Live-tool parameters.**

**Live tool turning direction.**

 Live tool clockwise.

 Live tool counter-clockwise.

8.

CANNED CYCLES OF THE EDITOR: LONGITUDINAL Multiple thread milling.

**Coolant status.**



Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

**Machining parameters.**

**Helical machining.**



Counterclockwise machining.



Clockwise machining.

**Machining direction.**



Machining from the part surface towards the bottom of the thread.



Machining from the bottom of the thread towards the part surface.

**Tool type.**



Tool for threading in a single pass.



Tool with a single edge cutter.



Tool with a multi-edge cutter. The number of edges of the tool must be defined.

N Number of edges. If it is set to 0, the cycle execute a threading of a single pitch.

**Thread data.**

- B Thread pitch.
- K Depth of the thread (distance between the crest and the root of the thread).
- $\Delta$  Maximum penetration step of the thread. If this data is not programmed or programmed with a 0 value, the cycle will run a single pass down to the finishing stock.
- $\delta$  Finishing stock; if not programmed, the cycle assumes a value of 0.
- $\alpha$  Thread entry (start) angle. Angle (in degrees) of the segment formed by the center of the hole or of the boss and the thread entry point with respect to the abscissa axis. If not programmed, the cycle will take the value of 0.
- Ds Safety distance to the wall of the thread for the threading entry (start). If not programmed or programmed with a 0 value, the cycle will enter the thread from the center of the hole if it is an inside threading or it will issue an error message if it is an outside threading.

**Penetration feedrate.**

- F Penetration feedrate.

## 8.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL  
Multiple thread milling.**Live-tool data.**

- Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.
  - Live tool data programmed. The cycle shows the data related to the live tool.
- S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Tool data.**

- T Tool.
- D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

- Activating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.
- Deactivating the execution of M functions before machining.

The editor will only show this option if the user has configured the editor to allow programming M functions.

### 8.10.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 If the operation was programmed with another tool, the CNC will make a tool change moving to the change point if the machine so requires.
- 2 The cycle turns the live tool at the indicated rpm.
- 3 The tool approaches in rapid up to the starting point, keeping the safety distance selected according to axes X and Z (safety point).
- 4 The cycle orients the spindle to the angular position corresponding to the initial machining operation.
- 5 Rapid movement to the machining point, in X for the face and in Z for the side of the part (approach point).
- 6 Rapid movement to the thread entry point, calculated by the cycle.
- 7 Machining depending on the selected tool.



- Tool for threading in a single pass.

Helical movement, with the indicated direction and pitch, to the bottom of the thread (the movement will only be one revolution).

Helical thread exiting movement, tangent to the previous helical path.

It must be borne in mind that in the exit tangent to the helical path, the exit point will exceed the coordinate of the bottom of the thread along the longitudinal axis.



- Tool with a single edge cutter.

Helical movement, with the indicated pitch and direction, to the bottom of the thread.

Helical thread exiting movement, tangent to the previous helical path.

It must be borne in mind that in the exit tangent to the helical path, the exit point will exceed the coordinate of the bottom of the thread along the longitudinal axis.



- Tool with a multi-edge cutter.

Helical movement, with the indicated pitch and direction (the movement will be one revolution).

Helical thread exiting movement, tangent to the previous helical path.

Rapid movement to the thread entry point of the next threading path.

Repetition of the previous 2 steps until reaching the bottom of the thread. It must be borne in mind that in the last helical exit, the exit point will exceed the coordinate of the bottom of the thread along the longitudinal axis.

- 8 Fast backwards to the approach point.
- 9 Depending on the number of machining operations programmed, the spindle orients to the second point and performs a new machining operation as described in points 6, 7 and 8.
- 10 Once the operation or cycle has ended, the tool will return to the safety point.
- 11 The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

8.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL Multiple thread milling.

**FAGOR** 

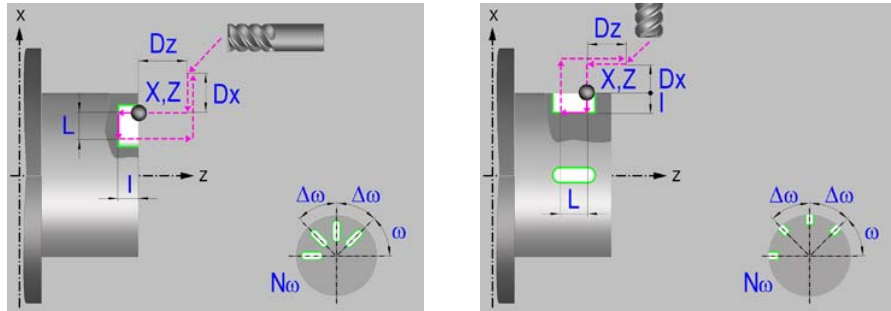
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 8.11 Multiple slot milling.

Multiple slots may be milled out on the side of the part or on its face.



### Machining on the face or side of the part.



Machining on the face of the part.



Machining on the side of the part.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

### Work plane.



Positioning on the XZ axes.



Positioning on the C axis.



Positioning on the Y axis.

### Coordinates of the profile's starting and final points.

**X** X coordinate of the initial point of each machining operation. The coordinate must be given in the active units, radius or diameter.

**Z** Z coordinate of the initial point of each machining operation.

**Y** Y coordinate of the first positioning movement.

**Yf** Y coordinate for the withdrawal movement after the cycle ends; if not programmed, there is no Y axis withdrawal. If the cycle is machining in multiple mode, the Y withdrawal point is only applied to the last cycle.

This parameter lets withdraw the tool to the right position in order to be able to continue with the turning operations that do not use the Y axis. If the next cycle also uses the Y axis, there is no need to withdraw (retract) the axis.

**C** C coordinate of the first positioning movement.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

### Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

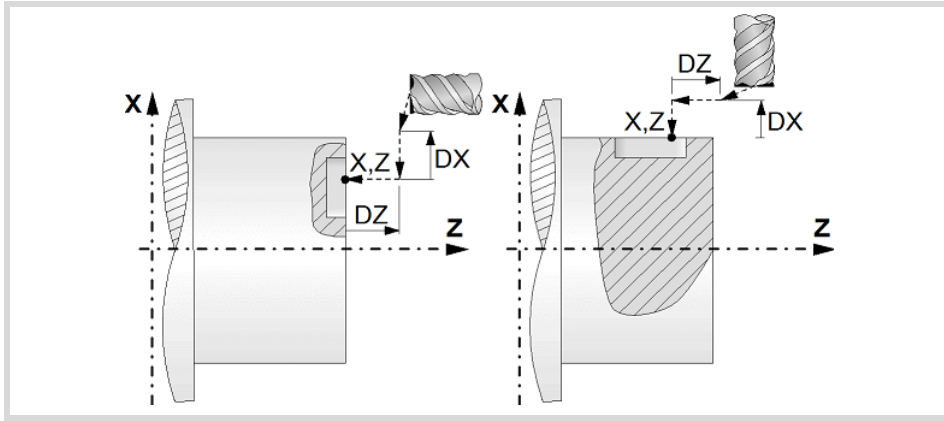
**Dx, Dz** Safety distance referred to the starting point.

- Machining on the face of the part. The value of the safety distance on X is always defined in radius and it may be either positive or negative. The value of the safety distance in Z is always assumed with positive values.
- Machining on the side of the part. The value of the safety distance on X is always defined in radius and it is always assumed with positive values. The value of the safety distance in Z may be either negative or positive.

# 8.

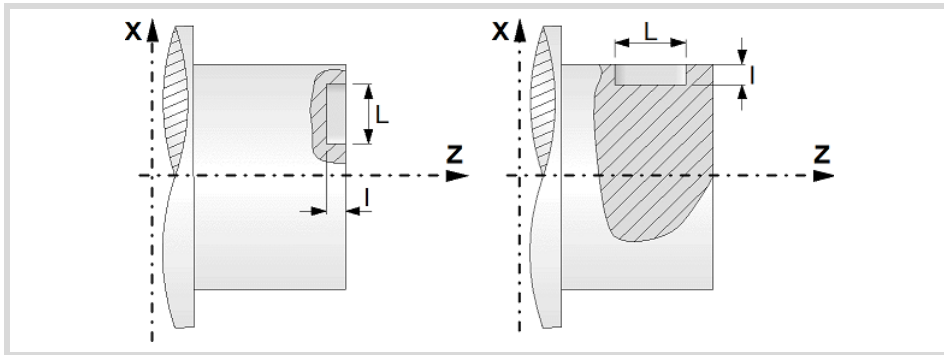
CANNED CYCLES OF THE EDITOR: LONGITUDINAL  
Multiple slot milling.





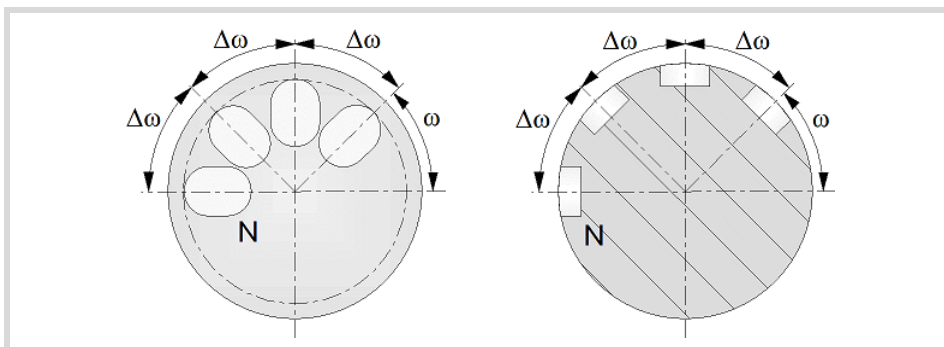
**Programming of the drilling operation.**

- I Depth of the slot.
- L Length of the slot.





**Angular position of the machining operations.**

- $\omega$  Angular position of the spindle (in degrees) for the first machining operation.
- $\Delta\omega$  Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.
- $N\omega$  Total number of operations.



**Live-tool parameters.**

**Live tool turning direction.**

-  Live tool clockwise.
-  Live tool counter-clockwise.



**CANNED CYCLES OF THE EDITOR. LONGITUDINAL**  
Multiple slot milling.



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

(REF: 1709)

**Coolant status.**

Coolant ON. The CNC outputs the M8 function to the PLC.



Coolant OFF. The CNC outputs the M9 function to the PLC.

**Machining parameters. Roughing.****Penetration step.**

$\Delta$  Maximum penetration step. If not programmed or programmed with a 0 value, it will run a single pass up to the finishing stock.

The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed.

**Feedrate.**

F Roughing feedrate.

Fp Penetration feedrate.

**Live-tool data.**

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.



Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycle assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

**Tool data.**

T Tool.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

**Machining M functions.**

Activating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



Deactivating the execution of M functions before machining.

The editor will only show this option if the user has configured the editor to allow programming M functions.

**Machining parameters. Finishing.****Finishing stock at the bottom.**

$\delta$  Finishing stock at the bottom.

**Feedrate.**

F Finishing feedrate.

8.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL  
Multiple slot milling.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

**Live-tool data.**

S Live tool turning speed.

8.

**CANNED CYCLES OF THE EDITOR. LONGITUDINAL**  
Multiple slot milling.



FAGOR AUTOMATION

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

(REF: 1709)

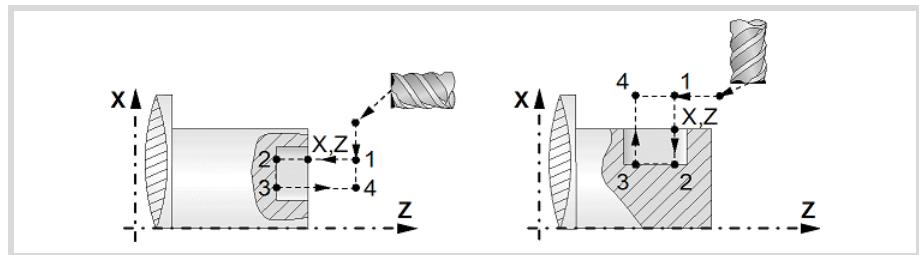
### 8.11.1 Basic operation.

The machining steps in this cycle are as follows:

- 1 If the operation was programmed with another tool the CNC will make a tool change, moving to the safety point.
- 2 The cycle turns the live tool at the indicated rpm.
- 3 The tool approaches in rapid up to the starting point, keeping the safety distance selected according to axes X and Z (safety point).
- 4 The cycle orients the spindle to the angular position corresponding to the initial slot.
- 5 Rapid movement to the slot milling position, in X for the face and in Z for the side of the part (approach point).
- 6 Milling of the slot by following these steps:

The cycle runs successive roughing passes until reaching the bottom of the slot respecting the finishing stocks. It then executes a single finishing pass eliminating the finishing stock.

Each pass is executed as follows. First, penetration at programmed feedrate (segment 1-2). Then, mill the slot by moving the X or Z axis accordingly (segment 2-3). And last, withdrawal to the approach point (sections 3-4 and 4-1).



- 7 Depending on the number of machining operations programmed, the spindle orients to the second point and mills a new slot as described in point 6.
- 8 Once the operation or cycle has ended, the tool will return to the safety point.
- 9 The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

# 8.

CANNED CYCLES OF THE EDITOR: LONGITUDINAL  
Multiple slot milling.



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

# CANNED CYCLES OF THE EDITOR. POSITIONING.

# 9

When pressing the softkey, the editor shows the cycle used last. When pressing the same softkey again, the menu shows all the cycles of the group.

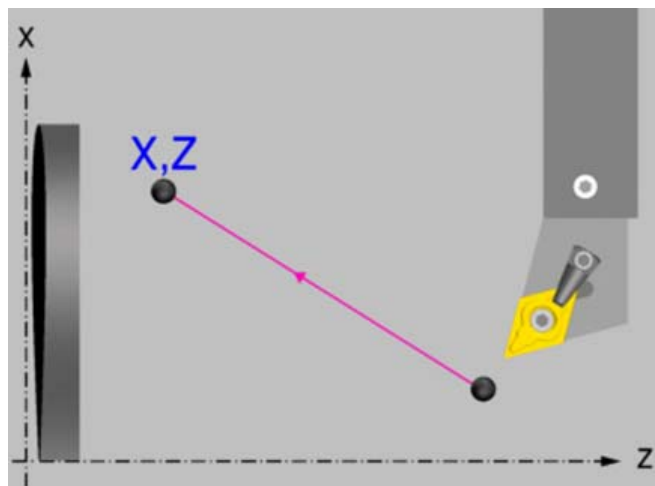


- Positioning cycle.
- Positioning cycle with M functions.

## 9.1 Positioning cycle.

9.

CANNED CYCLES OF THE EDITOR. POSITIONING. Positioning cycle.



### Geometric parameters.

#### Type of movement.



Interpolated movement.



First the X axis (cross) and then the Z axis (longitudinal).



First the Z axis (longitudinal) and then the X axis (cross).

Each time the type of movement is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Coordinates of the target point.

X, Z      Coordinates of the starting point

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

### Spindle parameters.

#### Spindle turning direction

The cycle lets select the spindle turning direction or leave the spindle stopped.



Spindle stopped.



Spindle clockwise.



Spindle counterclockwise.

#### Spindle work mode; rpm or Constant Surface Speed.



Constant turning speed.



Constant surface speed. Use this icon to select the spindle gear.

**Maximum spindle speed.**

Smax Maximum spindle turning speed; if not programmed, the cycle will maintain the maximum active speed.

**Spindle range (gear).**

GEAR Spindle gear; if not programmed, the cycle assumes a value of 0. If it is set to 0, the cycle does not change gears.

**Movement parameters.**

**Machining feedrate.**



Programmed feedrate F. The feedrate (F) must be defined with this icon.



Rapid feed.

F Machining feedrate.

**Spindle speed.**

S Spindle turning speed (rpm) or cutting speed.

**Tool data.**

T Tool.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

9.

CANNED CYCLES OF THE EDITOR. POSITIONING.  
Positioning cycle.



FAGOR AUTOMATION

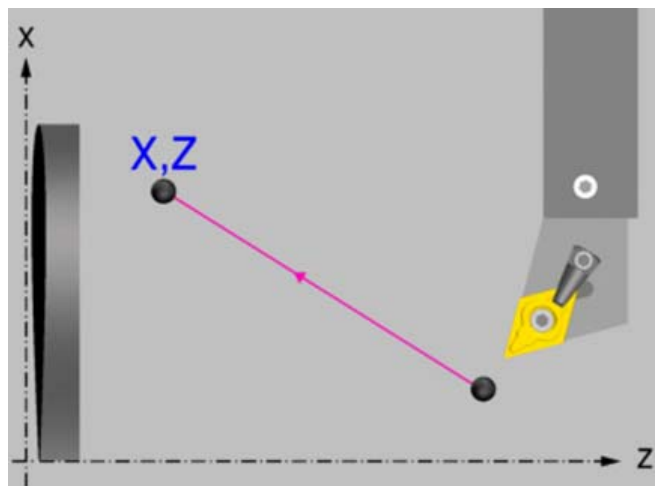
CNC 8060  
CNC 8065

(REF: 1709)

## 9.2 Positioning cycle with M functions.

9.

CANNED CYCLES OF THE EDITOR. POSITIONING.  
Positioning cycle with M functions.



### Geometric parameters.

#### Type of movement.



Interpolated movement.



First the X axis (cross) and then the Z axis (longitudinal).



First the Z axis (longitudinal) and then the X axis (cross).

Each time the type of movement is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

#### Coordinates of the target point.

X, Z      Coordinates of the starting point

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

#### The auxiliary functions “M” which will be executed before and after positioning.

Auxiliary function “M” is the name given to the functions determined by the manufacturer which allow the different machine devices to be governed. Up to 12 auxiliary functions may be defined, 6 before executing the movement and 6 afterwards.

The functions will be executed in the same order as these are arranged on the list.

### Spindle parameters.

#### Spindle turning direction

The cycle lets select the spindle turning direction or leave the spindle stopped.



Spindle stopped.



Spindle clockwise.



Spindle counterclockwise.



### Spindle work mode; rpm or Constant Surface Speed.



Constant turning speed.



Constant surface speed. Use this icon to select the spindle gear.

### Maximum spindle speed.

Smax Maximum spindle turning speed; if not programmed, the cycle will maintain the maximum active speed.

### Spindle range (gear).

GEAR Spindle gear; if not programmed, the cycle assumes a value of 0. If it is set to 0, the cycle does not change gears.

## Movement parameters.

### Machining feedrate.



Programmed feedrate F. The feedrate (F) must be defined with this icon.



Rapid feed.

F Machining feedrate.

### Spindle speed.

S Spindle turning speed (rpm) or cutting speed.

### Tool data.

T Tool.

D Tool offset; if not programmed, the cycle will assume the offset associated with the tool, set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.

9.

**CANNED CYCLES OF THE EDITOR. POSITIONING.**  
Positioning cycle with M functions.

**FAGOR** 

FAGOR AUTOMATION

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

(REF: 1709)

# 9.

## **CANNED CYCLES OF THE EDITOR. POSITIONING.**

Positioning cycle with M functions.



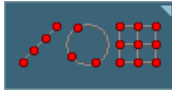
**CNC 8060**  
**CNC 8065**

(REF: 1709)

# CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.

# 10

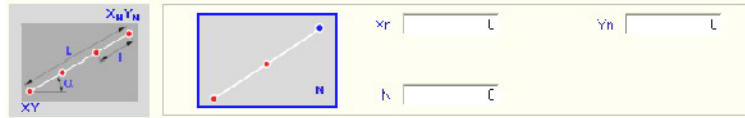
When pressing the softkey, the editor shows the cycle used last. When pressing the same softkey again, the menu shows all the cycles of the group.



- Multiple machining in straight line
- Multiple machining in arc pattern
- Multiple machining in rectangular pattern
- Multiple machining in grid pattern
- Random multiple machining (several points defined by the user).
- Without multiple machining.

## 10.1 Multiple machining in straight line.

There are 5 different ways to define the machining operation. Use the icon to select the desired option.



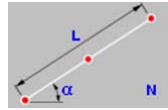
10.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.  
Multiple machining in straight line.

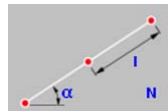
### Positioning options.



End point and number of machining operations.



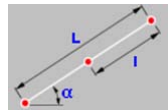
Length, angle and number of machining operations.



Distance between machining operations, angle and number of machining operations.



End point and distance between two machining operations.



Length, angle and distance between two machining operations.

$X_n, Y_n$  Coordinates of the end point of the positioning movement.

$N$  Number of machining operations including that of the cycle definition point.

$\alpha$  Angle of the line with the abscissa axis.

$L$  Total length of the line.

$I$  Distance between two consecutive machining operations.



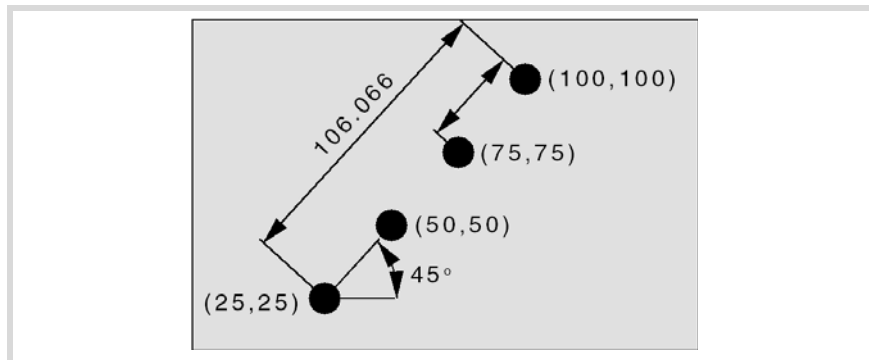
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

### 10.1.1 Programming example.

The canned cycle defined at point X25, Y25 is to be repeated at the rest of the points.



We now show the 5 possible ways to define it.

- |    |  |                |
|----|--|----------------|
| 1) | Coordinates of the end point.          | Xn 100, Yn 100 |
|    | Total number of machining operations.  | N 4            |
| 2) | Angle of the path.                     | $\alpha$ 45    |
|    | Length of the line.                    | L 106.066      |
|    | Total number of machining operations.  | N 4            |
| 3) | Angle of the path.                     | $\alpha$ 45    |
|    | Total number of machining operations.  | N 4            |
|    | Distance between machining operations. | I 35.3553      |
| 4) | Coordinates of the end point.          | Xn 100, Yn 100 |
|    | Distance between machining operations. | I 35.3553      |
| 5) | Angle of the path.                     | $\alpha$ 45    |
|    | Length of the line.                    | L 106.066      |
|    | Distance between machining operations. | I 35.3553      |

# 10.

**CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.**  
Multiple machining in straight line.

**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 10.2 Multiple machining in arc pattern.

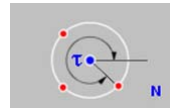
There are 9 ways to define the machining operation. Use the icon to select the desired option.



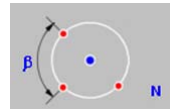
# 10.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.  
Multiple machining in arc pattern.

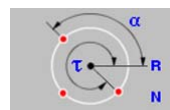
### Positioning options.



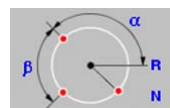
Arc center coordinates, number of machining operations and final angle of the arc.



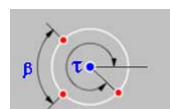
Arc center coordinates, number of machining operations and angular distance between two machining operations.



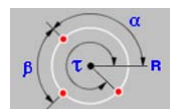
Arc radius, number of machining operations and starting and final angles of the arc.



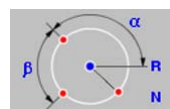
Arc radius, number of machining operations, starting angle of the arc and angular distance between two machining operations.



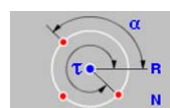
Arc center coordinates, final angle of the arc and angular distance between two machining operations.



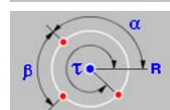
Arc radius, angular distance between two machining operations and starting and final angles of the arc.



Arc center coordinates, arc radius, number of machining operations, starting angle of the arc and angular distance between two machining operations.



Arc center coordinates, arc radius, number of machining operations and starting and final angles of the arc.



Arc center coordinates, arc radius, angular distance between two machining operations and starting and final angles of the arc.

Xa, Ya Arc center coordinates.

R Radius of the arc.

N Number of machining operations including that of the cycle definition point.

$\alpha$  Starting angle of the arc.

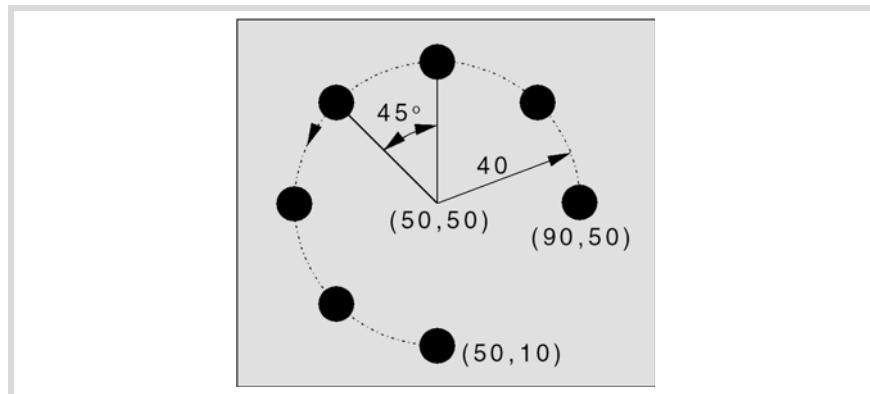
$\tau$  Final angle of the arc.

$\beta$  Angular distance between two machining operations.

The cycle moves counter-clockwise; to move clockwise, define the angular distance between machining operations with a negative sign.

### 10.2.1 Programming example.

The canned cycle defined at point X90, Y50 is to be repeated at the rest of the points.



We now show the 9 possible ways to define it.

- |    |  |              |
|----|--|--------------|
| 1) | Arc center coordinates.                        | Xa 50, Ya 50 |
|    | Total number of machining operations.          | N 7          |
|    | Angle of the end point.                        | $\tau$ 270   |
| 2) | Arc center coordinates.                        | Xa 50, Ya 50 |
|    | Total number of machining operations.          | N 7          |
|    | Angular distance between machining operations. | $\beta$ 45   |
| 3) | Radius of the arc.                             | R 40         |
|    | Total number of machining operations.          | N 7          |
|    | Angle of the starting point.                   | $\alpha$ 0   |
|    | Angle of the end point.                        | $\tau$ 270   |
| 4) | Radius of the arc.                             | R 40         |
|    | Total number of machining operations.          | N 7          |
|    | Angle of the starting point.                   | $\alpha$ 0   |
|    | Angular distance between machining operations. | $\beta$ 45   |
| 5) | Arc center coordinates.                        | Xa 50, Ya 50 |
|    | Angle of the end point.                        | $\tau$ 270   |
|    | Angular distance between machining operations. | $\beta$ 45   |
| 6) | Radius of the arc.                             | R 40         |
|    | Angle of the starting point.                   | $\alpha$ 0   |
|    | Angle of the end point.                        | $\tau$ 270   |
|    | Angular distance between machining operations. | $\beta$ 45   |
| 7) | Arc center coordinates.                        | Xa 50, Ya 50 |
|    | Radius of the arc.                             | R 40         |
|    | Total number of machining operations.          | N 7          |
|    | Angle of the starting point.                   | $\alpha$ 0   |
|    | Angular distance between machining operations. | $\beta$ 45   |
| 8) | Arc center coordinates.                        | Xa 50, Ya 50 |
|    | Radius of the arc.                             | R 40         |
|    | Total number of machining operations.          | N 7          |
|    | Angle of the starting point.                   | $\alpha$ 0   |
|    | Angle of the end point.                        | $\tau$ 270   |
| 9) | Arc center coordinates.                        | Xa 50, Ya 50 |
|    | Radius of the arc.                             | R 40         |
|    | Angle of the starting point.                   | $\alpha$ 0   |
|    | Angle of the end point.                        | $\tau$ 270   |
|    | Angular distance between machining operations. | $\beta$ 45   |

# 10.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.  
Multiple machining in arc pattern.

**FAGOR** 

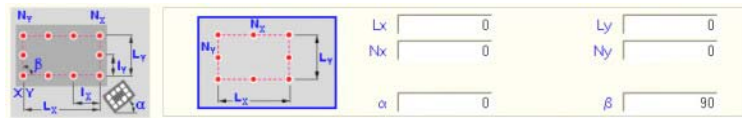
FAGOR AUTOMATION

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

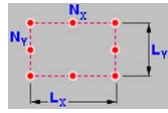
(REF: 1709)

## 10.3 Multiple machining in rectangular pattern.

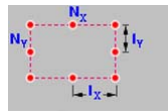
There are 3 ways to define the machining operation. Use the icon to select the desired option.



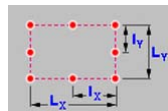
### Positioning options.



Length of each side and number of machining operations.



Distance between two machining operations and number of machining operations.



Length of each side and distance between two machining operations.

$L_x, L_y$  Length of the rectangle, on each axis.

$N_x, N_y$  Number of machining operations (including that of the cycle definition point) on each axis.

$l_x, l_y$  Distance between two machining operations, on each axis.

$\alpha$  Angle of the rectangle with the abscissa axis.

$\beta$  Angle between both sides.

The cycle assumes as starting point the lower left one; if it is not, define the machining length ( $L_x, L_y$ ) with the proper sign or the distance between machining operations ( $l_x, l_y$ ).

# 10.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.  
Multiple machining in rectangular pattern.



FAGOR AUTOMATION

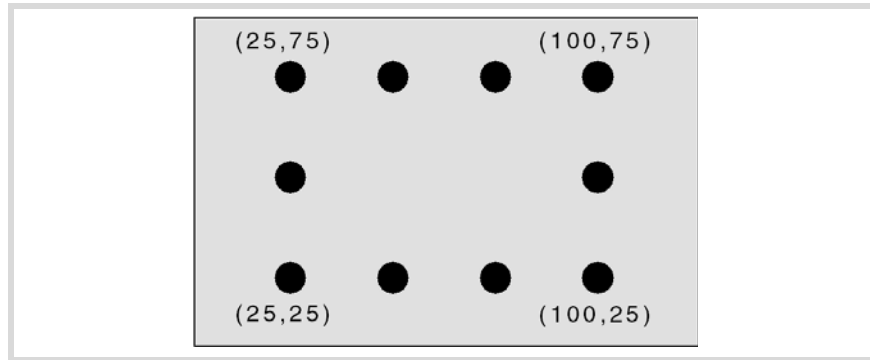
CNC 8060  
CNC 8065

(REF: 1709)



### 10.3.1 Programming example.

The canned cycle defined at point X25, Y25 is to be repeated at the rest of the points.



We now show the 3 possible ways to define it.

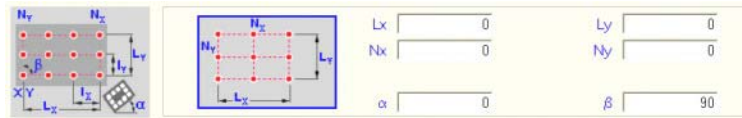
- |    |  |              |
|----|--|--------------|
| 1) | Length of each side of the rectangle.    | Lx 75, Ly 50 |
|    | Number of machining operations.          | Nx 4, Ny 3   |
|    | Angle with respect to the abscissa axis. | $\alpha$ 0   |
|    | Angle between paths.                     | $\beta$ 90   |
| 2) | Number of machining operations.          | Nx 4, Ny 3   |
|    | Distance between machining operations.   | lx 25, ly 25 |
|    | Angle with respect to the abscissa axis. | $\alpha$ 0   |
|    | Angle between paths.                     | $\beta$ 90   |
| 3) | Length of each side of the rectangle.    | Lx 75, Ly 50 |
|    | Distance between machining operations.   | lx 25, ly 25 |
|    | Angle with respect to the abscissa axis. | $\alpha$ 0   |
|    | Angle between paths.                     | $\beta$ 90   |

# 10.

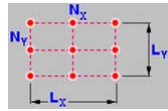
**CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.**  
Multiple machining in rectangular pattern.

## 10.4 Multiple machining in grid pattern.

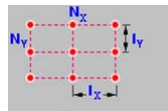
There are 3 ways to define the machining operation. Use the icon to select the desired option.



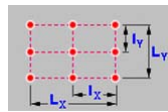
### Positioning options.



Length of each side and number of machining operations.



Distance between two machining operations and number of machining operations.



Length of each side and distance between two machining operations.

$L_x, L_y$  Length of the grid, on each axis.

$N_x, N_y$  Number of machining operations (including that of the cycle definition point) on each axis.

$I_x, I_y$  Distance between two machining operations, on each axis.

$\alpha$  Angle of the grid with the abscissa axis.

$\beta$  Angle between both sides.

The cycle assumes as starting point the lower left one; if it is not, define the machining length ( $L_x, L_y$ ) with the proper sign or the distance between machining operations ( $I_x, I_y$ ).

# 10.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.  
Multiple machining in grid pattern.



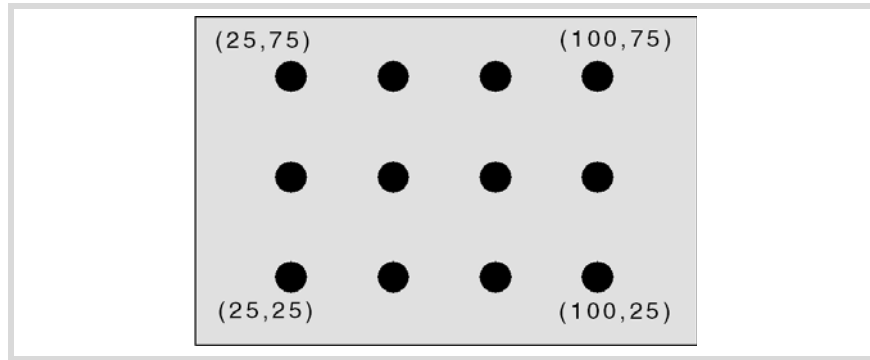
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

### 10.4.1 Programming example.

The canned cycle defined at point X25, Y25 is to be repeated at the rest of the points.



We now show the 3 possible ways to define it.

- |    |  |              |
|----|--|--------------|
| 1) | Length of each side of the grid.         | Lx 75, Ly 50 |
|    | Number of machining operations.          | Nx 4, Ny 3   |
|    | Angle with respect to the abscissa axis. | $\alpha$ 0   |
|    | Angle between paths.                     | $\beta$ 90   |
| 2) | Number of machining operations.          | Nx 4, Ny 3   |
|    | Distance between machining operations.   | lx 25, ly 25 |
|    | Angle with respect to the abscissa axis. | $\alpha$ 0   |
|    | Angle between paths.                     | $\beta$ 90   |
| 3) | Length of each side of the grid.         | Lx 75, Ly 50 |
|    | Distance between machining operations.   | lx 25, ly 25 |
|    | Angle with respect to the abscissa axis. | $\alpha$ 0   |
|    | Angle between paths.                     | $\beta$ 90   |

# 10.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.

Multiple machining in grid pattern.

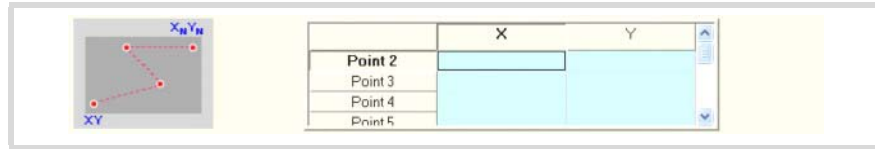
**FAGOR** 

FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 10.5 Random multiple machining.



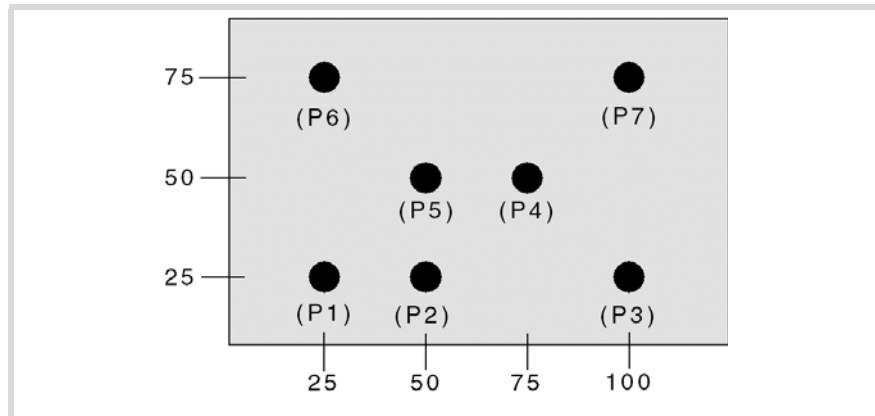
The starting point is the cycle defining point. The rest of the points (P2) to (P12) must be defined in the area for multiple machining.

# 10.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.  
Random multiple machining.

### 10.5.1 Programming example.

The canned cycle defined at point X25, Y25 is to be repeated at the rest of the points.



The canned cycle is defined at point (P1) X25, Y25. The rest of the points (P2) to (P7) must be defined in the area for multiple machining.

```
(P2)   X 50   Y 25
(P3)   X 100  Y 25
(P4)   X 75   Y 50
(P5)   X 50   Y 50
(P6)   X 25   Y 75
(P7)   X 100  Y 75
```



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

At all the levels except in face threading, the diameter value may be entered so the CNC calculates the relevant pitch and depths.

A new field (window) may be used to select the type of standard thread. If P.H (free-pitch thread) is selected, the pitch and the depth of the thread are selected directly by the user. The standard threads are single entry cylindrical threads.

It is possible to select a taper thread and also a standard thread, in this case, it will calculate the pitch and depth that would correspond to the standard cylindrical thread and it will be assigned to that taper thread.

The available thread types are:

M (S.I.)	Metric thread with normal pitch (international system).
M (S.I.F.)	Metric thread with fine pitch (international system).
B.S.W. (W)	Whitworth thread with normal pitch.
B.S.F.	Whitworth thread with fine pitch.
U.N.C.	Unified American thread with normal pitch.
U.N.F.	Unified American thread with fine pitch.
B.S.P.	Whitworth gas thread.
A.P.I.	Threads for pipes of the oil sector.

## 11.1 Regular pitch metric thread — M (S.I.)

11.

## STANDARD THREADS

Regular pitch metric thread — M (S.I.)

Diameter		Pitch		Depth (mm)	
(mm)	(inches)	(mm)	(inches)	Internal	External
0,3000	0,0118	0,0750	0,0030	0,0406	0,0460
0,4000	0,0157	0,1000	0,0039	0,0541	0,0613
0,5000	0,0197	0,1250	0,0049	0,0677	0,0767
0,6000	0,0236	0,1500	0,0059	0,0812	0,0920
0,8000	0,0315	0,2000	0,0079	0,1083	0,1227
1,0000	0,0394	0,2500	0,0098	0,1353	0,1534
1,2000	0,0472	0,2500	0,0098	0,1353	0,1534
1,4000	0,0551	0,3000	0,0118	0,1624	0,1840
1,6000	0,0630	0,3500	0,0138	0,1895	0,2147
1,7000	0,0669	0,3500	0,0138	0,1895	0,2147
1,8000	0,0709	0,3500	0,0138	0,1895	0,2147
2,0000	0,0787	0,4000	0,0157	0,2165	0,2454
2,2000	0,0866	0,4500	0,0177	0,2436	0,2760
2,3000	0,0906	0,4000	0,0157	0,2165	0,2454
2,5000	0,0984	0,4500	0,0177	0,2436	0,2760
2,6000	0,1024	0,4500	0,0177	0,2436	0,2760
3,0000	0,1181	0,5000	0,0197	0,2707	0,3067
3,5000	0,1378	0,6000	0,0236	0,3248	0,3680
4,0000	0,1575	0,7000	0,0276	0,3789	0,4294
4,5000	0,1772	0,7500	0,0295	0,4060	0,4601
5,0000	0,1969	0,8000	0,0315	0,4330	0,4907
5,5000	0,2165	0,9000	0,0354	0,4872	0,5521
6,0000	0,2362	1,0000	0,0394	0,5413	0,6134
7,0000	0,2756	1,0000	0,0394	0,5413	0,6134
8,0000	0,3150	1,2500	0,0492	0,6766	0,7668
9,0000	0,3543	1,2500	0,0492	0,6766	0,7668
10,0000	0,3937	1,5000	0,0591	0,8120	0,9201
11,0000	0,4331	1,5000	0,0591	0,8120	0,9201
12,0000	0,4724	1,7500	0,0689	0,9473	1,0735
14,0000	0,5512	2,0000	0,0787	1,0826	1,2268
16,0000	0,6299	2,0000	0,0787	1,0826	1,2268
18,0000	0,7087	2,5000	0,0984	1,3533	1,5335
20,0000	0,7874	2,5000	0,0984	1,3533	1,5335
22,0000	0,8661	2,5000	0,0984	1,3533	1,5335
24,0000	0,9449	3,0000	0,1181	1,6239	1,8402
27,0000	1,0630	3,0000	0,1181	1,6239	1,8402
30,0000	1,1811	3,5000	0,1378	1,8946	2,1469
33,0000	1,2992	3,5000	0,1378	1,8946	2,1469
36,0000	1,4173	4,0000	0,1575	2,1652	2,4536
39,0000	1,5354	4,0000	0,1575	2,1652	2,4536
42,0000	1,6535	4,5000	0,1772	2,4359	2,7603
45,0000	1,7717	4,5000	0,1772	2,4359	2,7603
48,0000	1,8898	5,0000	0,1969	2,7065	3,0670
52,0000	2,0472	5,0000	0,1969	2,7065	3,0670
56,0000	2,2047	5,5000	0,2165	2,9772	3,3737
60,0000	2,3622	5,5000	0,2165	2,9772	3,3737
64,0000	2,5197	6,0000	0,2362	3,2478	3,6804
68,0000	2,6772	6,0000	0,2362	3,2478	3,6804
72,0000	2,8346	6,0000	0,2362	3,2478	3,6804
76,0000	2,9921	6,0000	0,2362	3,2478	3,6804
80,0000	3,1496	6,0000	0,2362	3,2478	3,6804

## 11.2 Fine pitch metric thread — M (S.I.F.)

The CNC calculates the depth with the formulae:

- Depth in internal threads = 0.5413 x Pitch
- Depth in external threads = 0.6134 x Pitch

Diameter		Pitch		Depth (mm)	
(mm)	(inches)	(mm)	(inches)	Internal	External
1,000	0,0394	0,2000	0,0079	0,1083	0,1227
1,200	0,0472	0,2000	0,0079	0,1083	0,1227
1,400	0,0551	0,2000	0,0079	0,1083	0,1227
1,700	0,0669	0,2000	0,0079	0,1083	0,1227
2,000	0,0787	0,2500	0,0098	0,1353	0,1534
2,300	0,0906	0,2500	0,0098	0,1353	0,1534
2,500	0,0984	0,3500	0,0138	0,1895	0,2147
2,600	0,1024	0,3500	0,0138	0,1895	0,2147
3,000	0,1181	0,3500	0,0138	0,1895	0,2147
3,500	0,1378	0,3500	0,0138	0,1895	0,2147
4,000	0,1575	0,5000	0,0197	0,2707	0,3067
4,500	0,1772	0,5000	0,0197	0,2707	0,3067
5,000	0,1969	0,5000	0,0197	0,2707	0,3067
6,000	0,2362	0,7500	0,0295	0,4060	0,4601
7,000	0,2756	0,7500	0,0295	0,4060	0,4601
8,000	0,3150	1,0000	0,0394	0,5413	0,6134
9,000	0,3543	1,0000	0,0394	0,5413	0,6134
10,000	0,3937	1,0000	0,0394	0,5413	0,6134
12,000	0,4724	1,2500	0,0492	0,6766	0,7668
13,000	0,5118	1,5000	0,0591	0,8120	0,9201
14,000	0,5512	1,5000	0,0591	0,8120	0,9201
16,000	0,6299	1,5000	0,0591	0,8120	0,9201
18,000	0,7087	1,5000	0,0591	0,8120	0,9201
20,000	0,7874	1,5000	0,0591	0,8120	0,9201
22,000	0,8661	1,5000	0,0591	0,8120	0,9201
24,000	0,9449	2,0000	0,0787	1,0826	1,2268
27,000	1,0630	2,0000	0,0787	1,0826	1,2268
30,000	1,1811	2,0000	0,0787	1,0826	1,2268
33,000	1,2992	2,0000	0,0787	1,0826	1,2268
36,000	1,4173	3,0000	0,1181	1,6239	1,8402
39,000	1,5354	3,0000	0,1181	1,6239	1,8402
42,000	1,6535	3,0000	0,1181	1,6239	1,8402
45,000	1,7717	3,0000	0,1181	1,6239	1,8402
48,000	1,8898	3,0000	0,1181	1,6239	1,8402
52,000	2,0472	3,0000	0,1181	1,6239	1,8402
56,000	2,2047	4,0000	0,1575	2,1652	2,4536
60,000	2,3622	4,0000	0,1575	2,1652	2,4536
64,000	2,5197	4,0000	0,1575	2,1652	2,4536
68,000	2,6772	4,0000	0,1575	2,1652	2,4536
72,000	2,8346	4,0000	0,1575	2,1652	2,4536
76,000	2,9921	4,0000	0,1575	2,1652	2,4536
80,000	3,1496	4,0000	0,1575	2,1652	2,4536

# 11.

STANDARD THREADS

Fine pitch metric thread — M (S.I.F.)



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 11.3 Regular pitch Whitworth thread — BSW (W.)

The threads must be defined in millimeters or inches. For example, to define a Whitworth thread with a 1/16 pitch, enter 1.5875 mm or 0.0625 inches.

The CNC calculates the pitch and the depth with the formulae:

- Pitch in millimeters = 25.4 / number of edges
- Pitch in inches = 1 / number of edges
- Depth in internal threads = 0.6403 x Pitch
- Depth in external threads = 0.6403 x Pitch

# 11.

STANDARD THREADS  
Regular pitch Whitworth thread — BSW (W.)

Thread	Pitch		Edges	Pitch		Depth (mm)	
	(mm)	(inches)		(mm)	(inches)	Internal	External
1/16	1,5875	0,0625	60	0,4233	0,0167	0,2710	0,2710
3/32	2,3812	0,0937	48	0,5292	0,0208	0,3388	0,3388
1/8	3,1750	0,1250	40	0,6350	0,0250	0,4066	0,4066
5/32	3,9687	0,1562	32	0,7938	0,0313	0,5083	0,5083
3/16	4,7625	0,1875	24	1,0583	0,0417	0,6776	0,6776
7/32	5,5562	0,2187	24	1,0583	0,0417	0,6776	0,6776
1/4	6,3500	0,2500	20	1,2700	0,0500	0,8132	0,8132
5/16	7,9375	0,3125	18	1,4111	0,0556	0,9035	0,9035
3/8	9,5250	0,3750	16	1,5875	0,0625	1,0165	1,0165
7/16	11,1125	0,4375	14	1,8143	0,0714	1,1617	1,1617
1/2	12,7000	0,5000	12	2,1167	0,0833	1,3553	1,3553
9/16	14,2875	0,5625	12	2,1167	0,0833	1,3553	1,3553
5/8	15,8750	0,6250	11	2,3091	0,0909	1,4785	1,4785
3/4	19,0500	0,7500	10	2,5400	0,1000	1,6264	1,6264
7/8	22,2250	0,8750	9	2,8222	0,1111	1,8071	1,8071
1	25,4000	1,0000	8	3,1750	0,1250	2,0330	2,0330
1 1/8	28,5750	1,1250	7	3,6286	0,1429	2,3234	2,3234
1 1/4	31,7500	1,2500	7	3,6286	0,1429	2,3234	2,3234
1 3/8	34,9250	1,3750	6	4,2333	0,1667	2,7106	2,7106
1 1/2	38,1000	1,5000	6	4,2333	0,1667	2,7106	2,7106
1 5/8	41,2750	1,6250	5	5,0800	0,2000	3,2527	3,2527
1 3/4	44,4500	1,7500	5	5,0800	0,2000	3,2527	3,2527
1 7/8	47,6250	1,8750	5	5,6444	0,2222	3,6141	3,6141
2	50,8000	2,0000	5	5,6444	0,2222	3,6141	3,6141
2 1/8	53,9750	2,1250	5	5,6444	0,2222	3,6141	3,6141
2 1/4	57,1500	2,2500	4	6,3500	0,2500	4,0659	4,0659
2 3/8	60,3250	2,3750	4	6,3500	0,2500	4,0659	4,0659
2 1/2	63,5000	2,5000	4	6,3500	0,2500	4,0659	4,0659
2 5/8	66,6750	2,6250	4	6,3500	0,2500	4,0659	4,0659
2 3/4	69,8500	2,7500	4	7,2571	0,2857	4,6467	4,6467
2 7/8	73,0250	2,8750	4	7,2571	0,2857	4,6467	4,6467
3	76,2000	3,0000	4	7,2571	0,2857	4,6467	4,6467
3 1/4	82,5500	3,2500	3	7,8154	0,3077	5,0042	5,0042
3 1/2	88,9000	3,5000	3	7,8154	0,3077	5,0042	5,0042
3 3/4	95,2500	3,7500	3	8,4667	0,3333	5,4212	5,4212
4	101,6000	4,0000	3	8,4667	0,3333	5,4212	5,4212
4 1/4	107,9500	4,2500	3	8,8348	0,3478	5,6569	5,6569
4 1/2	114,3000	4,5000	3	8,8348	0,3478	5,6569	5,6569
4 3/4	120,6500	4,7500	3	9,2364	0,3636	5,9141	5,9141
5	127,0000	5,0000	3	9,2364	0,3636	5,9141	5,9141



## 11.4 Fin pitch Whitworth thread — BSF

The threads must be defined in millimeters or inches. For example, to define a Whitworth thread with a 3/16 pitch, enter 4.7625 mm or 0.1875 inches.

The CNC calculates the pitch and the depth with the formulae:

- Pitch in millimeters = 25.4 / number of edges
- Pitch in inches = 1 / number of edges
- Depth in internal threads = 0.6403 x Pitch
- Depth in external threads = 0.6403 x Pitch

Thread	Pitch		Pitch Edges	Depth (mm)			
	(mm)	(inches)		Internal	External		
3/16	4,7625	0,1875	32	0,7937	0,0312	0,5082	0,5082
7/32	5,5562	0,2187	28	0,9071	0,0357	0,5808	0,5808
1/4	6,3500	0,2500	26	0,9769	0,0385	0,6255	0,6255
9/32	7,1437	0,2812	26	0,9769	0,0385	0,6255	0,6255
5/16	7,9375	0,3125	22	1,1545	0,0455	0,7392	0,7392
3/8	9,5250	0,3750	20	1,2700	0,0500	0,8132	0,8132
7/16	11,1125	0,4375	18	1,4111	0,0556	0,9035	0,9035
1/2	12,7000	0,5000	16	1,5875	0,0625	1,0165	1,0165
9/16	14,2875	0,5625	16	1,5875	0,0625	1,0165	1,0165
5/8	15,8750	0,6250	14	1,8143	0,0714	1,1617	1,1617
11/16	17,4625	0,6875	14	1,8143	0,0714	1,1617	1,1617
3/4	19,0500	0,7500	12	2,1167	0,0833	1,3553	1,3553
13/16	20,6375	0,8125	12	2,1167	0,0833	1,3553	1,3553
7/8	22,2250	0,8750	11	2,3091	0,0909	1,4785	1,4785
1	25,4000	1,0000	10	2,5400	0,1000	1,6264	1,6264
1 1/8	28,5750	1,1250	9	2,8222	0,1111	1,8071	1,8071
1 1/4	31,7500	1,2500	9	2,8222	0,1111	1,8071	1,8071
1 3/8	34,9250	1,3750	8	3,1750	0,1250	2,0330	2,0330
1 1/2	38,1000	1,5000	8	3,1750	0,1250	2,0330	2,0330
1 5/8	41,2750	1,6250	8	3,1750	0,1250	2,0330	2,0330
1 3/4	44,4500	1,7500	7	3,6286	0,1429	2,3234	2,3234
2	50,8000	2,0000	7	3,6286	0,1429	2,3234	2,3234
2 1/4	57,1500	2,2500	6	4,2333	0,1667	2,7106	2,7106
2 1/2	63,5000	2,5000	6	4,2333	0,1667	2,7106	2,7106
2 3/4	69,8500	2,7500	6	4,2333	0,1667	2,7106	2,7106
3	76,2000	3,0000	5	5,0800	0,2000	3,2527	3,2527

# 11.

## STANDARD THREADS

Fin pitch Whitworth thread — BSF



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 11.5 Regular pitch Unified American thread — UNC (NC, USS)

The threads must be defined in millimeters or inches. For example, to define an American thread with a 1/4 pitch, enter 6.3500 mm or 0.2500 inches.

The CNC calculates the pitch and the depth with the formulae:

- Pitch in millimeters = 25.4 / number of edges
- Pitch in inches = 1 / number of edges
- Depth in internal threads = 0.5413 x Pitch
- Depth in external threads = 0.6134 x Pitch

# 11.

STANDARD THREADS  
Regular pitch Unified American thread — UNC (NC, USS)

Thread	Pitch		Edges	Pitch		Depth (mm)	
	(mm)	(inches)		(mm)	(inches)	Internal	External
0,0730	1,8542	0,0730	64	0,3969	0,0156	0,2148	0,2435
0,0860	2,1844	0,0860	56	0,4536	0,0179	0,2455	0,2782
0,0990	2,5146	0,0990	48	0,5292	0,0208	0,2865	0,3246
0,1120	2,8448	0,1120	40	0,6350	0,0250	0,3437	0,3895
0,1250	3,1750	0,1250	40	0,6350	0,0250	0,3437	0,3895
0,1380	3,5052	0,1380	32	0,7938	0,0313	0,4297	0,4869
0,1640	4,1656	0,1640	32	0,7938	0,0313	0,4297	0,4869
0,1900	4,8260	0,1900	24	1,0583	0,0417	0,5729	0,6492
0,2160	5,4864	0,2160	24	1,0583	0,0417	0,5729	0,6492
1/4	6,3500	0,2500	20	1,2700	0,0500	0,6875	0,7790
5/16	7,9375	0,3125	18	1,4111	0,0556	0,7638	0,8656
3/8	9,5250	0,3750	16	1,5875	0,0625	0,8593	0,9738
7/16	11,1125	0,4375	14	1,8143	0,0714	0,9821	1,1129
1/2	12,7000	0,5000	13	1,9538	0,0769	1,0576	1,1985
9/16	14,2875	0,5625	12	2,1167	0,0833	1,1458	1,2984
5/8	15,8750	0,6250	11	2,3091	0,0909	1,2499	1,4164
3/4	19,0500	0,7500	10	2,5400	0,1000	1,3749	1,5580
7/8	22,2250	0,8750	9	2,8222	0,1111	1,5277	1,7311
1	25,4000	1,0000	8	3,1750	0,1250	1,7186	1,9475
1 1/8	28,5750	1,1250	7	3,6286	0,1429	1,9642	2,2258
1 1/4	31,7500	1,2500	7	3,6286	0,1429	1,9642	2,2258
1 3/8	34,9250	1,3750	6	4,2333	0,1667	2,2915	2,5967
1 1/2	38,1000	1,5000	6	4,2333	0,1667	2,2915	2,5967
1 5/8	41,2750	1,6250	5	5,0800	0,2000	2,7498	3,1161
1 3/4	44,4500	1,7500	5	5,0800	0,2000	2,7498	3,1161
2	50,8000	2,0000	5	5,6444	0,2222	3,0553	3,4623
2 1/4	57,1500	2,2500	5	5,6444	0,2222	3,0553	3,4623
2 1/2	63,5000	2,5000	4	6,3500	0,2500	3,4373	3,8951
2 3/4	69,8500	2,7500	4	6,3500	0,2500	3,4373	3,8951
3	76,2000	3,0000	4	6,3500	0,2500	3,4373	3,8951



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 11.6 Fine pitch Unified American thread — UNF (NF, SAE)

The threads must be defined in millimeters or inches. For example, to define an American thread with a 1/4 pitch, enter 6.3500 mm or 0.2500 inches.

The CNC calculates the pitch and the depth with the formulae:

- Pitch in millimeters = 25.4 / number of edges
- Pitch in inches = 1 / number of edges
- Depth in internal threads = 0.5413 x Pitch
- Depth in external threads = 0.6134 x Pitch

Thread	Pitch		Edges	Depth (mm)	
	(mm)	(inches)		Internal	External
0,0600	1,5240	0,0600	80	0,1719	0,1948
0,0730	1,8542	0,0730	72	0,1910	0,2164
0,0860	2,1844	0,0860	64	0,2148	0,2435
0,0990	2,5146	0,0990	56	0,2455	0,2782
0,1120	2,8448	0,1120	48	0,2865	0,3246
0,1250	3,1750	0,1250	44	0,3125	0,3541
0,1380	3,5052	0,1380	40	0,3437	0,3895
0,1640	4,1656	0,1640	36	0,3819	0,4328
0,1900	4,8260	0,1900	32	0,4296	0,4869
19/88	5,4864	0,2160	28	0,4910	0,5564
1/4	6,3500	0,2500	28	0,4910	0,5564
5/16	7,9375	0,3125	24	0,5729	0,6492
3/8	9,5250	0,3750	24	0,5729	0,6492
7/16	11,1125	0,4375	20	0,6875	0,7790
1/2	12,7000	0,5000	20	0,6875	0,7790
9/16	14,2875	0,5625	18	0,7638	0,8656
5/8	15,8750	0,6250	18	0,7638	0,8656
3/4	19,0500	0,7500	16	0,8593	0,9738
7/8	22,2250	0,8750	14	0,9821	1,1129
1	25,4000	1,0000	12	1,1458	1,2984
1 1/8	28,5750	1,1250	12	1,1458	1,2984
1 1/4	31,7500	1,2500	12	1,1458	1,2984
1 1/2	38,1000	1,5000	12	1,1458	1,2984

# 11.

## STANDARD THREADS

Fine pitch Unified American thread — UNF (NF, SAE)



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 11.7 Whitworth gas thread — BSP

The threads must be defined in millimeters or inches. For example, to define a Whitworth thread with a 1/8 pitch, enter 3,175 mm or 0.125 inches.

The CNC calculates the pitch and the depth with the formulae:

- Pitch in millimeters = 25.4 / number of edges
- Pitch in inches = 1 / number of edges
- Depth in internal threads = 0.6403 x Pitch
- Depth in external threads = 0.6403 x Pitch

# 11.

STANDARD THREADS  
Whitworth gas thread — BSP

Thread	Pitch		Edges	Pitch		Depth (mm)	
	(mm)	(inches)		(mm)	(inches)	Internal	External
1/8	3,1750	0,1250	28	0,9071	0,0357	0,58082	0,58082
1/4	6,3500	0,2500	19	1,3368	0,0526	0,85595	0,85595
3/8	9,5250	0,3750	19	1,3368	0,0526	0,85595	0,85595
1/2	12,7000	0,5000	14	1,8143	0,0714	1,16170	1,16170
5/8	15,8750	0,6250	14	1,8143	0,0714	1,16170	1,16170
3/4	19,0500	0,7500	14	1,8143	0,0714	1,16170	1,16170
7/8	22,2250	0,8750	14	1,8143	0,0714	1,16170	1,16170
1	25,4000	1,0000	11	2,3091	0,0909	1,47852	1,47852
1 1/8	28,5750	1,1250	11	2,3091	0,0909	1,47852	1,47852
1 1/4	31,7500	1,2500	11	2,3091	0,0909	1,47852	1,47852
1 3/8	34,9250	1,3750	11	2,3091	0,0909	1,47852	1,47852
1 1/2	38,1000	1,5000	11	2,3091	0,0909	1,47852	1,47852
1 3/4	44,4500	1,7500	11	2,3091	0,0909	1,47852	1,47852
2	50,8000	2,0000	11	2,3091	0,0909	1,47852	1,47852
2 1/4	57,1500	2,2500	11	2,3091	0,0909	1,47852	1,47852
2 1/2	63,5000	2,5000	11	2,3091	0,0909	1,47852	1,47852
2 3/4	69,8500	2,7500	11	2,3091	0,0909	1,47852	1,47852
3	76,2000	3,0000	11	2,3091	0,0909	1,47852	1,47852
3 1/4	82,5500	3,2500	11	2,3091	0,0909	1,47852	1,47852
3 1/2	88,9000	3,5000	11	2,3091	0,0909	1,47852	1,47852
3 3/4	95,2500	3,7500	11	2,3091	0,0909	1,47852	1,47852
4	101,6000	4,0000	11	2,3091	0,0909	1,47852	1,47852
4 1/2	114,3000	4,5000	11	2,3091	0,0909	1,47852	1,47852
5	127,0000	5,0000	11	2,3091	0,0909	1,47852	1,47852
5 1/2	139,7000	5,5000	11	2,3091	0,0909	1,47852	1,47852
6	152,4000	6,0000	11	2,3091	0,0909	1,47852	1,47852
7	177,8000	7,0000	10	2,5400	0,1000	1,62636	1,62636
8	203,2000	8,0000	10	2,5400	0,1000	1,62636	1,62636
9	228,6000	9,0000	10	2,5400	0,1000	1,62636	1,62636
10	254,0000	10,0000	10	2,5400	0,1000	1,62636	1,62636
11	279,4000	11,0000	8	3,1750	0,1250	2,03295	2,03295
12	304,8000	12,0000	8	3,1750	0,1250	2,03295	2,03295



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

## 11.8 Threads for pipes of the oil sector — A.P.I.

Thread		TPI <sup>1</sup> (Thread per inch)	TPF <sup>2</sup> (Taper per foot)	Depth (H)	Pitch
API 5	CR 08	8	3/4	1.810	-3,175
	CR 10	10	3/4	1.412	-2,54
	CR 05	5	3/4	1,575	-5,08
API 7	V38 4-2	4	2	3.095	-6,35
	V38 4-3	4	3	3.083	-6,35
	V40 5-3	5	3	2.993	-5,08
	V50 4-2	4	2	3.741	-6,35
	V50 4-3	4	3	3.754	-6,35

(1) TPI; Threads per inch. (2) TPF; Taper per foot.

# 11.

**STANDARD THREADS**

Threads for pipes of the oil sector — A.P.I.

**FAGOR** 

FAGOR AUTOMATION

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

(REF: 1709)

# 11.

## STANDARD THREADS

Threads for pipes of the oil sector — A.P.I.



FAGOR AUTOMATION

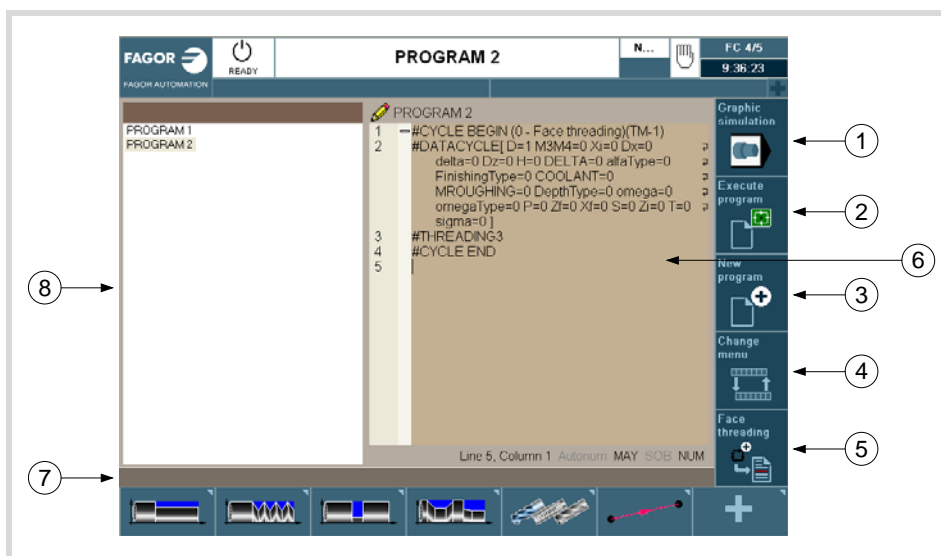
CNC 8060  
CNC 8065

(REF: 1709)

Part-programs may be edited, saved, simulated and executed. Each one of these programs is made up by concatenating simple operations or single cycles.

## 12.1 List of saved programs

Press [EDIT] to access the list of saved part-programs. The CNC will display the following screen:



- 1 Softkey to start simulating the selected program.
- 2 Softkey to start executing the selected program.
- 3 Softkey to create a new program.
- 4 Softkey that changes the horizontal softkey menu. The softkeys appearing when pressing this softkey are:
  - Open program.
  - Operations with blocks.
  - Undo/Redo.
  - Customizing.
- 5 Replace or insert the edited cycle.
- 6 Window that shows the cycles and ISO-coded blocks that make up the selected part. To edit a cycle, put the cursor on the cycle press [RECALL].
- 7 Message bar.
- 8 This softkey shows the list of saved programs. In this window, it is possible to move through the list of programs. When selecting a program, the right window will show its contents. To change the working folder, press [RECALL] while the focus is on the list and a folder explorer will open to choose the new folder.

To toggle the focus between the list of programs and the part editor, press [CTRL][F2].

## 12.2 Edit a new part-program

To edit a new program, proceed as follows:

- 1 Press [EDIT] to access the list of saved part-programs.
- 2 Press the vertical softkey "New program". The following options will appear:
  - New: Entering a name in the dialog box and pressing the New button will create a new empty file with that name.
  - Copy: Entering a name in the dialog box and pressing the Copy button will create a file with that name and all the cycles and ISO-coded blocks of the program selected at the time on the list of programs will be copied into it.
  - Cancel: To cancel the creation of a new program.

Keyboard shortcuts [CTRL][C] and [CTRL][V] may be used on the programs list to copy and paste a program.

## 12.3 Delete a new part program

To delete a part-program, proceed as follows:

- 1 Press [EDIT] to access the list of saved part-programs.
- 2 Select, with the pointer, the part-program to be deleted from the left column. A program cannot be deleted if it is being executed or simulated.
- 3 Press the [DEL] key.

The bottom of the screen will show a message requesting confirmation of the deleting operation.

- If [ENTER] is pressed, the CNC will delete the selected program and will update the list of saved programs.
- If [ESC] is pressed, the program will not be deleted and it will quit deleting operation.

## 12.4 Inserting comments in any part-program.

The block or cycle may be added at the end of the program, after the last operation or insert it between 2 existing operations or replace it with one of the cycles of the program. To save the cycle, proceed as follows:

- 1 Define the desired cycle assigning the relevant data to it.
- 2 Press the softkey "Save cycle" to access the list of part-programs.
- 3 Select the desired program in the left window. The right window shows the contents of the part-program.
- 4 In the right window, place the cursor at the position where the edited cycle is to be inserted or replaced.
- 5 Press the vertical softkey to replace or insert the edited cycle and select the desired option.

## 12.5 Inserting a cycle in the current part-program.

The editor lets insert the cycle being edited directly into the current program in the cursor position.

- 1 Define the desired cycle assigning the relevant data to it.
- 2 Press the [INS] key to insert the cycle into the current program in the cursor position.

# 12.

**SAVING PROGRAMS**  
Edit a new part-program



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)



The simulation may be used to graphically show a part-program or an operation with the data used to define it. This way, the simulation may be used to check the part-program or the operation before executing it or saving it and, therefore, correct or modify its data.

It is possible to execute or simulate a part-program or any operation. The simulation or execution may be done from beginning to end or step by step by pressing the [SINGLE] key.

## 13.1 Execute a part-program

Proceed as follows to execute a part-program:

- 1 Press [EDIT] to access the list of part-programs stored.
- 2 Select on the left column the program to be executed.
- 3 Press the vertical softkey "Execute program".

After pressing this softkey, the CNC shows the standard screen for the TC work mode. When pressing the two-color key, the CNC shows the auxiliary screen for the TC mode. When pressing the vertical softkey "Graphics", the CNC shows the graphics screen in execution.

- 4 Once the desired screen has been selected, press [START].

### 13.1.1 Execute a portion of a part-program

Proceed as follows to execute a portion of a part-program:

- 1 Press [EDIT] to access the list of part-programs stored.
- 2 Select the program from the left column and the operation (on the right column) from which to begin executing the part program.
- 3 Press the vertical softkey "Execute program".

After pressing this softkey, the CNC shows the standard screen for the TC work mode. When pressing the two-color key, the CNC shows the auxiliary screen for the TC mode. When pressing the vertical softkey "Graphics", the CNC shows the graphics screen in execution.

- 4 Once the desired screen has been selected, press [START].

# 13.

EXECUTION AND SIMULATION  
Execute a part-program

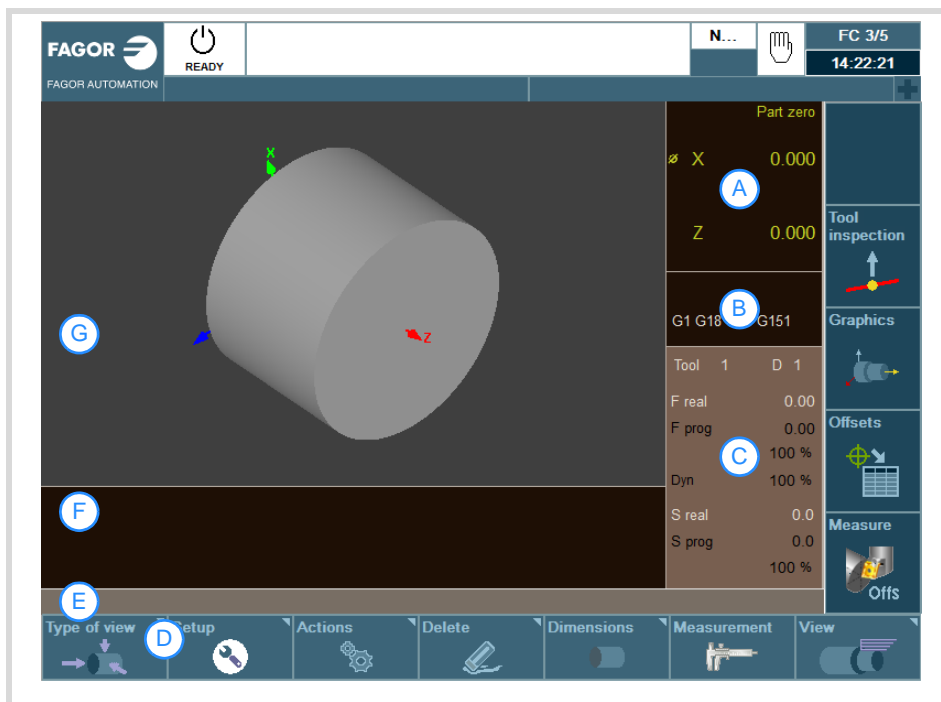


FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

### 13.1.2 Graphics screen in execution.



- A Window that informs about the position of the axes during the execution of the program.
- B Window that shows the status of the G functions.
- C Window that shows the selected tool T, the tool offset D, the real F and the programmed F as well as the real S and the programmed S.
- D Softkeys with graphic display options.
  - Type of view.
  - Configuration.
  - Actions.
  - Delete.
  - Dimensions.
  - Measurement.
- E Message bar.
- F Window that shows the lines of the program being executed.
- G Window that shows the lines of the program being executed through graphics.

# 13.

## EXECUTION AND SIMULATION Execute a part-program

## 13.2 Simulating a part-program

Proceed as follows to simulate a part-program:

- 1 Press [EDIT] to access the list of part-programs stored.
- 2 Select on the left column the program to be simulated.
- 3 Press the vertical softkey "Graphic simulation". After pressing this softkey, the CNC shows the graphics screen in simulation.
- 4 Press the vertical softkey "Start simulation".

# 13.

**EXECUTION AND SIMULATION**  
Simulating a part-program

### 13.2.1 Simulate a portion of a part-program

Proceed as follows to simulate a portion of a part-program:

- 1 Press [EDIT] to access the list of part-programs stored.
- 2 Select the program from the left column and the operation (on the right column) from which to begin executing the part program.
- 3 Press the vertical softkey "Graphic simulation". After pressing this softkey, the CNC shows the graphics screen in simulation.
- 4 Press the vertical softkey "Start simulation".

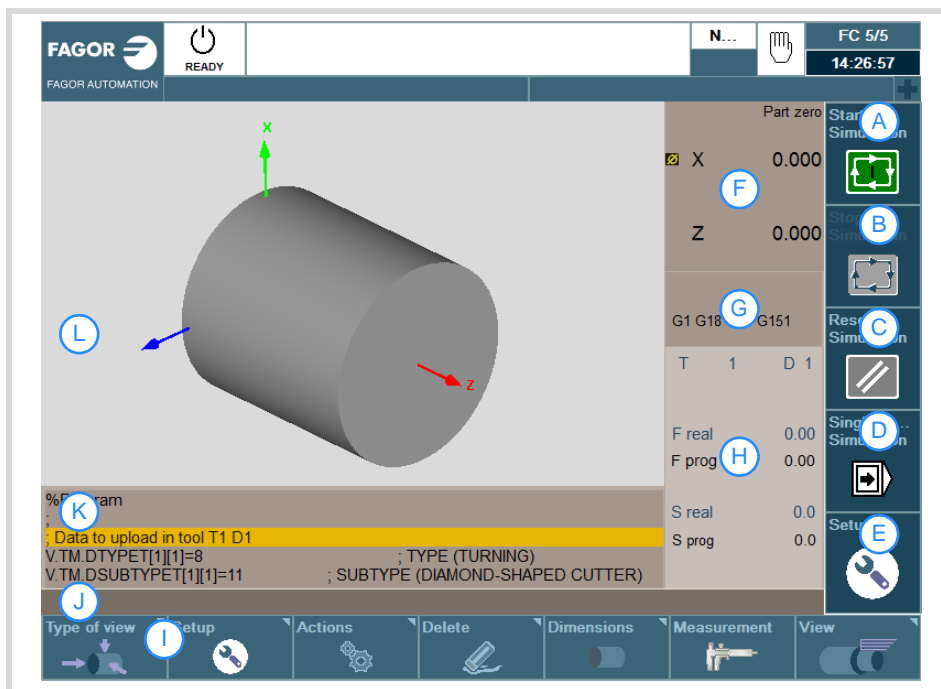


FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

### 13.2.2 Graphics screen in simulation.



Description of the page items:

- 1 Softkey to start simulating the selected program.
- 2 Softkey to stop simulating the selected program.
- 3 Softkey to reset the execution of the selected program.
- 4 Softkey to simulate the program block by block.
- 5 Simulation options.
- 6 Window that informs about the position of the axes during the execution of the program.
- 7 Window that shows the status of the G functions.
- 8 Window that shows the selected tool T, the tool offset D, the real F and the programmed F as well as the real S and the programmed S.
- 9 Softkeys with graphic display options.
- 10 Message bar.
- 11 Window that shows the lines of the program being executed.
- 12 Window that shows the lines of the program being executed through graphics.

# 13.

**EXECUTION AND SIMULATION**  
Simulating a part-program

### 13.2.3 Simulation options.

The available simulation options are accessed from the icon menu. Pressing the icon displays a window that shows the following options.



#### Tool radius compensation

It activates or cancels tool radius compensation to simulate the program.

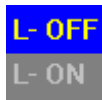
Tool radius compensation being off, the simulation will ignore the programmed tool radius compensation.



#### Conditional stop during simulation

It simulates the external "conditional stop" switch.

While active, the program simulation will be interrupted at the blocks having a "conditional stop" function "M01". The simulation will resume when pressing the START icon.



#### Software and work zone limits.

Activate or deactivate the software limits and the work zones for program simulation.

- Having this option active, if during simulation the axes reach the software limits or the work zones, the CNC interrupts simulation and issues the corresponding error message.
- Having this option deactivated, the CNC will ignore the software limits and the work zones during simulation. The CNC ignores the option to assume the active zones in execution.



#### Block jump

It simulates the external "block skip" switch.

While active, the blocks having the block-skip character "/" will be ignored (not simulated).



#### Cancel channel synchronization

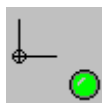
There is one icon for each channel. It cancels the channel synchronization wait periods during simulation.

When active, the wait period will end immediately and it will resume the execution of the program.



#### Synchronizing spindles.

There is an icon for each spindle where to indicate the spindle to synchronize with. The -0- value cancels the synchronization.



#### Assume the active origins for execution.

With this option, when starting the simulation or pressing the simulation reset, the CNC applies to the simulation the origins set in the execution environment (for example, the part zero set in jog mode).



#### Assume the active zones in execution.

Assume or not, for simulation, the active zones in execution.

# 13.

EXECUTION AND SIMULATION  
Simulating a part-program



FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

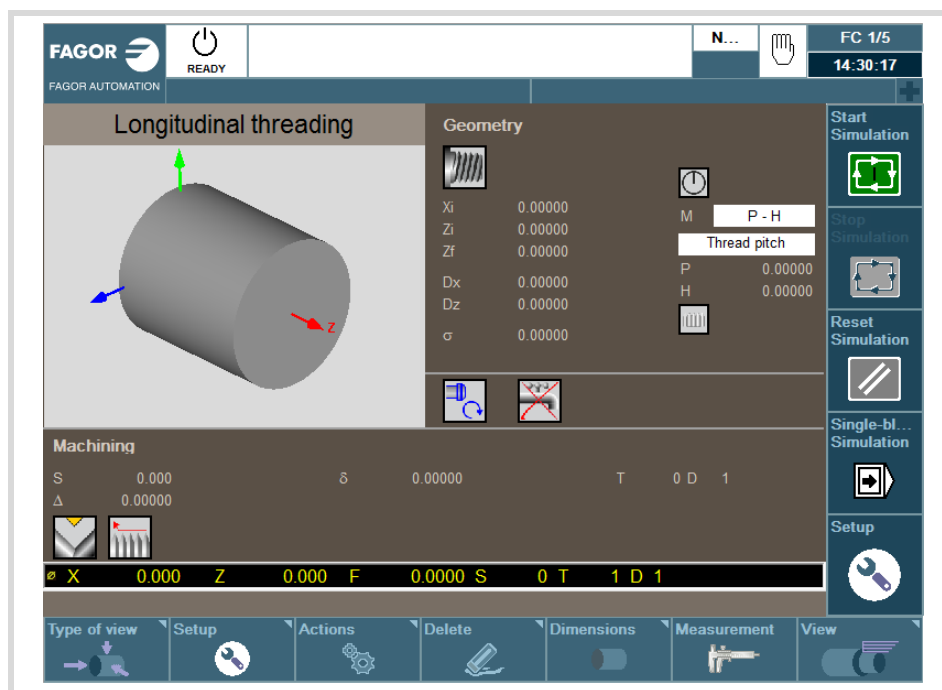
## 13.3 Simulating or executing an operation that has been saved

Proceed as follows to simulate or execute an operation that has been saved as a part of program:

- 1 Press [EDIT] to access the list of part-programs stored.
- 2 Select the program that contains it from the left column and the operation to be simulated or executed from the right column.
- 3 Press the [RECALL] key.
- 4 Press the vertical softkey "Simulate cycle" to simulate the operation and the vertical softkey "Execute cycle" to execute it.

### 13.3.1 Simulating a cycle

To simulate the edited cycle, press the vertical softkey "Simulate cycle".



For further information on simulating cycles, see the chapter ["2.1.4 Cycle simulation"](#).

# 13.

**EXECUTION AND SIMULATION**

Simulating or executing an operation that has been saved

**FAGOR** 

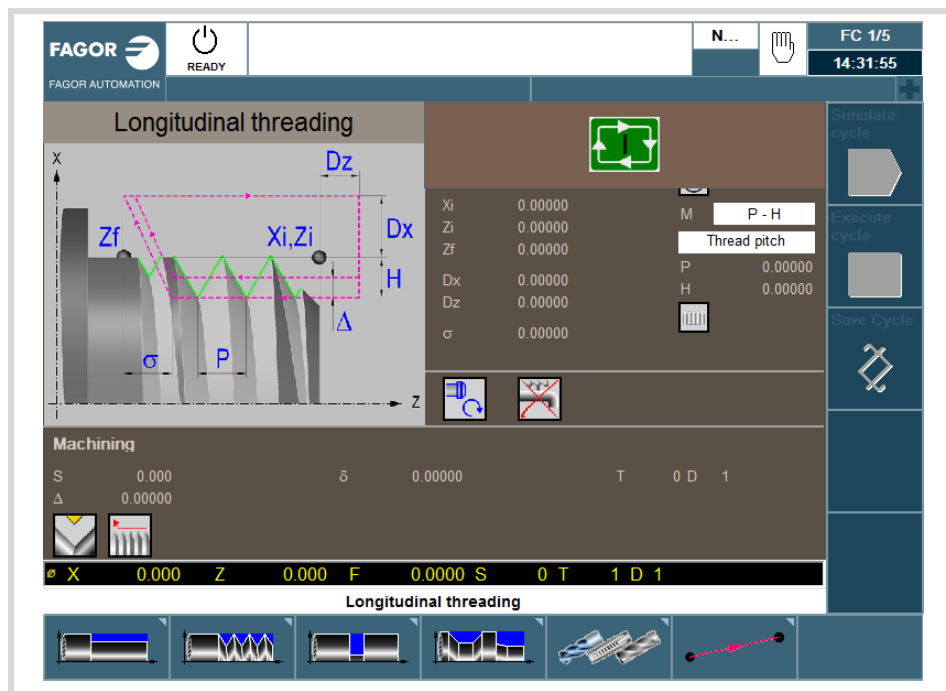
FAGOR AUTOMATION

CNC 8060  
CNC 8065

(REF: 1709)

### 13.3.2 Cycle execution

To execute the edited cycle, press the vertical softkey "Execute cycle".



For further information on executing cycles, see the chapter "2.1.5 Cycle execution".

# 13.

## EXECUTION AND SIMULATION

Simulating or executing an operation that has been saved











FAGOR AUTOMATION

**Fagor Automation S. Coop.**

Bº San Andrés, 19 - Apdo. 144  
E-20500 Arrasate-Mondragón, Spain

Tel: +34 943 719 200

+34 943 039 800

Fax: +34 943 791 712

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

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