

CNC

8060

8065

Canned cycles (-T- model)

(Ref: 1502)



FAGOR AUTOMATION

MACHINE SAFETY

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

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

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

HARDWARE EXPANSIONS

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

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

COMPUTER VIRUSES

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

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

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

DUAL-USE PRODUCTS

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



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

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

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

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

The content of this manual and its validity for the product described here has been verified. Even so, involuntary errors are possible, thus no absolute match is guaranteed. Anyway, the contents of the manual is periodically checked making and including the necessary corrections in a future edition. We appreciate your suggestions for improvement.

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

INDEX

About the product	7
Version history - CNC 8060	11
Version history - CNC 8065	13
Safety conditions	15
Warranty terms	19
Material returning terms	21
CNC maintenance	23

CHAPTER 1 MACHINING CANNED CYCLES (ISO)

1.1	General concepts	25
1.2	G81. Turning canned cycle with straight sections	27
1.3	G82. Facing canned cycle with straight sections	30
1.4	G83. Drilling / tapping canned cycle	33
1.5	G84. Turning canned cycle with arcs	36
1.6	G85. Facing canned cycle with arcs	39
1.7	G86. Longitudinal threading canned cycle	42
1.8	G87. Face threading canned cycle	48
1.9	G88. Grooving canned cycle along the X axis	54
1.10	G89. Grooving canned cycle along the Z axis	55
1.11	G66. Pattern repeat canned cycle	56
1.12	G68. Stock removal cycle along X axis	62
1.13	G69. Stock removal canned cycle along Z axis	69
1.14	G160. Drilling / tapping on the face of the part	76
1.15	G161. Drilling / tapping on the side of the part	80
1.16	G162. Slot milling canned cycle along the side of the part	84
1.17	G163. Radial slot milling cycle (on the face of the part)	86

CHAPTER 2 CYCLE EDITOR

2.1	Configuring the cycle editor.	91
2.2	Teach-in mode.	92
2.3	Selecting data, profiles and icons	93
2.4	Simulate a canned cycle.	94

CHAPTER 3 CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

3.1	Simple turning cycle	98
3.1.1	Basic operation.	101
3.2	Turning cycle with vertex rounding.	103
3.2.1	Basic operation.	107
3.3	Simple facing cycle	109
3.3.1	Basic operation.	112
3.4	Facing cycle with vertex rounding	114
3.4.1	Basic operation.	118
3.5	Vertex chamfering cycle	120
3.5.1	Basic operation.	125
3.6	"Chamfering between points" cycle.	127
3.6.1	Basic operation.	132
3.7	Vertex chamfering cycle 2	134
3.7.1	Basic operation.	139
3.8	Vertex rounding cycle.	141
3.8.1	Basic operation.	146
3.9	"Rounding between points" cycle	148
3.9.1	Basic operation.	153

CHAPTER 4 CANNED CYCLES OF THE EDITOR. TAPPING.

4.1	Longitudinal threading cycle.	156
4.1.1	Basic operation.	160
4.2	Taper threading cycle.	161
4.2.1	Basic operation.	166
4.3	Face threading cycle	167
4.3.1	Basic operation.	171



CNC 8060
CNC 8065

(REF: 1502)

4.4	Thread repair cycle.....	172
4.4.1	Basic operation.....	178
4.5	Multi-entry threading cycle.....	179
4.5.1	Basic operation.....	184

CHAPTER 5 CANNED CYCLES OF THE EDITOR. GROOVING.

5.1	Calibrate the grooving tool (location code, shape).....	186
5.1.1	Outside longitudinal grooving.....	186
5.1.2	Inside longitudinal grooving.....	187
5.1.3	Face grooving.....	188
5.2	Simple longitudinal grooving cycle.....	189
5.2.1	Basic operation.....	193
5.3	Simple face grooving cycle.....	195
5.3.1	Basic operation.....	199
5.4	Inclined longitudinal grooving cycle.....	201
5.4.1	Basic operation.....	206
5.5	Inclined face grooving cycle.....	208
5.5.1	Basic operation.....	213
5.6	Cut-off cycle.....	215
5.6.1	Basic operation.....	218

CHAPTER 6 CANNED CYCLES OF THE EDITOR. PROFILES.

6.1	Point-to-point turning cycle.....	220
6.1.1	Basic operation.....	226
6.1.2	Programming example.....	227
6.2	Profile turning cycle.....	228
6.2.1	Basic operation.....	234
6.2.2	Programming example.....	235
6.3	ZC/YZ plane profiling cycle.....	241
6.3.1	Basic operation.....	245
6.4	Rectangular pocket cycle in the ZC/YZ plane.....	246
6.4.1	Basic operation.....	251
6.5	Circular pocket cycle in the ZC/YZ plane.....	252
6.5.1	Basic operation.....	256
6.6	Circular pre-empted pocket cycle in the ZC/YZ plane.....	257
6.6.1	Basic operation.....	261
6.7	2D profile pocket cycle in the ZC/YZ plane.....	262
6.7.1	Basic operation.....	266
6.8	XC/XY plane profiling cycle.....	267
6.8.1	Basic operation.....	271
6.9	Rectangular pocket cycle in the XC/XY plane.....	272
6.9.1	Basic operation.....	277
6.10	Circular pocket cycle in the XC/XY plane.....	278
6.10.1	Basic operation.....	282
6.11	Circular pre-empted pocket in the XC/XY plane.....	283
6.11.1	Basic operation.....	287
6.12	2D profile pocket cycle in the XC/XY plane.....	288
6.12.1	Basic operation.....	292

CHAPTER 7 CANNED CYCLES OF THE EDITOR. LONGITUDINAL MACHINING.

7.1	Center punching.....	294
7.1.1	Basic operation.....	297
7.2	Drilling.....	298
7.2.1	Basic operation.....	301
7.3	Tapping.....	302
7.3.1	Basic operation.....	304
7.4	Multiple center-punching.....	305
7.4.1	Basic operation.....	308
7.5	Multiple drilling.....	309
7.5.1	Basic operation.....	313
7.6	Multiple tapping.....	314
7.6.1	Basic operation.....	317
7.7	Multiple reaming.....	318
7.7.1	Basic operation.....	321
7.8	Multiple boring.....	322
7.8.1	Basic operation.....	326
7.9	Multiple bore milling.....	327
7.9.1	Basic operation.....	330
7.10	Multiple thread milling.....	331
7.10.1	Basic operation.....	335



CNC 8060
CNC 8065

(REF: 1502)

Canned cycles (.T. model)

7.11	Multiple slot milling.....	336
7.11.1	Basic operation.....	340

CHAPTER 8 CANNED CYCLES OF THE EDITOR. POSITIONING.

8.1	Positioning cycle.....	342
8.2	Positioning cycle with M functions.....	344

CHAPTER 9 CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.

9.1	Multiple machining in straight line.....	348
9.1.1	Programming example.....	349
9.2	Multiple machining in arc pattern.....	350
9.2.1	Programming example.....	351
9.3	Multiple machining in rectangular pattern.....	352
9.3.1	Programming example.....	353
9.4	Multiple machining in grid pattern.....	354
9.4.1	Programming example.....	355
9.5	Random multiple machining.....	356
9.5.1	Programming example.....	356

CHAPTER 10 DYNAMIC DISTRIBUTION OF THE MACHINING OPERATIONS BETWEEN CHANNELS.

10.1	Activating and canceling dynamic machining distribution.....	359
10.1.1	Distribution of passes between channels.....	361
10.1.2	Synchronized identical passes.....	362

CHAPTER 11 STANDARD THREADS

11.1	Regular pitch metric thread — M (S.I.).....	365
11.2	Fine pitch metric thread — M (S.I.F.).....	366
11.3	Regular pitch Whitworth thread — BSW (W.).....	367
11.4	Fin pitch Whitworth thread — BSF.....	368
11.5	Regular pitch Unified American thread — UNC (NC, USS).....	369
11.6	Fine pitch Unified American thread — UNF (NF, SAE).....	370
11.7	Whitworth gas thread — BSP.....	371
11.8	Threads for pipes of the oil sector — A.P.I.	372

CHAPTER 12 USER CYCLES.



CNC 8060
CNC 8065

(REF: 1502)

ABOUT THE PRODUCT

BASIC CHARACTERISTICS.

Basic characteristics.	8060 M	8060 T	8065 M	8065 T
PC-based system.	Closed system		Open system	
Number of axes.	3 to 6		3 to 28	
Number of spindles.	1 to 2	1 to 3	1 to 4	
Number of tool magazines.	1	1 to 2	1 to 4	
Number of execution channels.	1	1 to 2	1 to 4	
Number of handwheels.	1 to 3		1 to 12	
Type of servo system.	Analog / Sercos Digital		Analog / Digital Sercos / Digital Mechatrolink	
Communications.	Ethernet		RS485 / RS422 / RS232 Ethernet	
Integrated PLC. PLC execution time. Digital inputs / Digital outputs. Marks / Registers. Timers / Counters. Symbols.			< 1ms/K 1024 / 1024 8192 / 1024 512 / 256 Unlimited	
Block processing time.	< 1.5 ms		< 1 ms	

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

Customizing (on an open system only).

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



CNC 8060
CNC 8065

(REF: 1502)

SOFTWARE OPTIONS.

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

Software options (.M· model).

	8060 M	8065 M		8065 M Power	
		Basic	Pack 1	Basic	Pack 1
Open system. Access to the administrator mode.	---	---	---	Option	Option
Number of execution channels	1	1	1	1	1 to 4
Number of axes	3 to 6	3 to 6	5 to 8	5 to 12	8 to 28
Number of spindles	1 to 2	1	1 to 2	1 to 4	1 to 4
Maximum number of axes and spindles	7	7	10	16	32
Number of tool magazines	1	1	1	1 to 2	1 to 4
Limited to 4 interpolated axes	Standard	Option	Option	Option	Option
IEC 61131 language	---	---	Option	Option	Option
HD graphics	---	Option	Option	Standard	Standard
Conversational IIP	Option	Option	Standard	Standard	Standard
Dual-purpose machines (M-T)	---	---	---	Option	Standard
"C" axis	Standard	Standard	Standard	Standard	Standard
Dynamic RTCP	Option	---	Option	Option	Standard
HSSA machining system.	Option	Standard	Standard	Standard	Standard
Probing canned cycles	Option	Option	Standard	Standard	Standard
Tandem axes	Option	---	Option	Standard	Standard
Synchronism and cams	Option	---	---	Option	Standard
Tangential control	Option	---	Standard	Standard	Standard
Volumetric compensation (up to 10 m ³).	---	---	---	Option	Option
Volumetric compensation (more than 10 m ³).	---	---	---	Option	Option
ProGTL3 language.	---	Option	Option	Option	Option
Part-program translator.	---	Option	Option	Option	Option



CNC 8060
CNC 8065

(REF: 1502)

Software options (-T- model).

	8060 T	8065 T		8065 T Power	
		Basic	Pack 1	Basic	Pack 1
Open system. Access to the administrator mode.	---	---	---	Option	Option
Number of execution channels	1 to 2	1	1 to 2	1 to 2	1 to 4
Number of axes	3 to 6	3 to 5	5 to 7	5 to 12	8 to 28
Number of spindles	1 to 3	2	2	3 to 4	3 to 4
Maximun number of axes and spindles	7	7	9	16	32
Number of tool magazines	1 to 2	1	1 to 2	1 to 2	1 to 4
Limited to 4 interpolated axes	Standard	Option	Option	Option	Option
IEC 61131 language	---	---	Option	Option	Option
HD graphics	---	Option	Option	Standard	Standard
Conversational IIP	Option	Option	Standard	Standard	Standard
Dual-purpose machines (T-M)	---	---	---	Option	Standard
"C" axis	Standard	Option	Standard	Standard	Standard
Dynamic RTCP	Option	---	---	Option	Standard
HSSA machining system.	Option	Option	Standard	Standard	Standard
Probing canned cycles	Option	Option	Standard	Standard	Standard
Tandem axes	Option	---	Option	Standard	Standard
Synchronism and cams	Option	---	Option	Option	Standard
Tangential control	Option	---	---	Option	Standard
Volumetric compensation (up to 10 m³).	---	---	---	Option	Option
Volumetric compensation (more than 10 m³).	---	---	---	Option	Option
ProGTL3 language.	---	---	---	---	---
Part-program translator.	---	---	---	---	---

VERSION HISTORY - CNC 8060

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

Ref. 1402

Software V01.00
8060. First version.

Ref. 1412

Software V01.10	
User cycles.	
Software V01.10.03	
ISO coded cycles. <ul style="list-style-type: none"> G86. Longitudinal threading canned cycle. G87. Face threading canned cycle. 	The cycle offers the following novelties. <ul style="list-style-type: none"> Parameter M. Variable-pitch threads. Parameter H. New 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). Parameter U. Partial thread repair or machining a thread by entering directly into it without doing it from the outside (parameter "K"). Parameter N. Blind threads.
ISO coded cycles. <ul style="list-style-type: none"> G66. Pattern repeat canned cycle. G68. Stock removal cycle along X axis. 	The cycle offers the following novelties. <ul style="list-style-type: none"> Parameter J. Roughing by segments.
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"> Center punching. Multiple center-punching. Multiple reaming. Multiple boring. Multiple bore milling. Multiple thread milling. Circular pre-empted pocket cycle in the ZC/YZ plane. Circular pre-empted pocket in the XC/XY plane. 	New cycles in the editor.
Cycles of the editor. <ul style="list-style-type: none"> Longitudinal threading. Taper threading. Thread repair (except face threading). Multi-entry (start) threading 	Standard threads A.P.I.
Cycles of the editor. <ul style="list-style-type: none"> Taper threading. Face threading. Thread repair. Multi-entry (start) threading 	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"> Longitudinal threading. Taper threading. Multi-entry (start) threading 	The cycles offers the possibility of programming variable-pitch threads.
Cycles of the editor. <ul style="list-style-type: none"> Taper threading. Thread repair. 	The cycle offers the following novelties. <ul style="list-style-type: none"> The cycle offers the possibility of executing a 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".
Cycles of the editor. <ul style="list-style-type: none"> Thread repair. 	The cycle offers the following novelties. <ul style="list-style-type: none"> The cycle offers the possibility the number of entries (starts) in thread repair. The cycle offers the possibility of programming a face thread repair. Step of the programmed depth (h) of the thread maintaining the geometry.



CNC 8060
CNC 8065

(REF: 1502)

Software V01.10.03	
Cycles of the editor. <ul style="list-style-type: none"> • Longitudinal threading. • Taper threading. • Face threading. • Thread repair. • Multi-entry (start) threading 	The cycle offers the following novelties. <ul style="list-style-type: none"> • The cycle offers the possibility of making blind threads. • The cycle offers the possibility of defining the thread pitch in number of threads per inch.
Cycles of the editor. <ul style="list-style-type: none"> • ZC/YZ plane profiling cycle. 	The cycles offer the possibility of defining a profile in the YZ plane.
Cycles of the editor. <ul style="list-style-type: none"> • XC/XY plane profiling cycle. 	The cycles offer the possibility of defining a profile in the XY plane.
Cycles of the editor. <ul style="list-style-type: none"> • 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. • 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. 	The cycles offer the possibility of adding a multiple angular positioning.
Cycles of the editor. <ul style="list-style-type: none"> • Point-to-point turning. • Profile turning. 	The cycles offer the possibility of machining a profile by segments.
Cycles of the editor. <ul style="list-style-type: none"> • Multiple drilling. • Multiple tapping. • Multiple slot milling. 	The 3 parameters that define the angular positioning of the spindle now have new names α is now called ω , β is now called $\Delta\omega$ and N is now called $N\omega$.
Cycles of the editor. <ul style="list-style-type: none"> • Points in line. • Points in arc. • Points in rectangle. • Points in grid. • Random points. 	The editor lets add multiple machining (linear, arc, rectangular, grid and random) to pockets and to longitudinal machining.

Ref. 1502**Software V01.11**

Corrected errors.



CNC 8060
CNC 8065

(REF: 1502)

VERSION HISTORY - CNC 8065

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

Ref. 1103

First version.

Ref. 1201

Software V04.22	
Cycles of the editor.	The [DEL] key deletes a profile from the list.
Cycles of the editor.	When selecting it in a constant surface speed cycle, it allows selecting the gear (range) even if the gear change is automatic.
Cycles of the editor.	The cycles make the approach to the starting point on both axes of the plane at the same time.
Cycles of the editor. Point-to-point turning cycle.	The table to define the points of the profile admits 25 points.
Cycles of the editor. Point-to-point turning cycle.	New icon to delete all the points of the table.
Cycles of the editor. Multiple tapping cycle.	The cycle allows programming the dwell at the bottom.
Cycles of the editor. Multiple slot milling cycle.	The cycle allows defining several penetration passes.
Cycles of the editor.	By default, the cycles assume for Xf the value set for Xi.
<ul style="list-style-type: none"> • 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. 	

Ref. 1209

Software V04.24	
Dynamic distribution of machining operations. Program the delay between two consecutive passes using a distance (parameter D).	<ul style="list-style-type: none"> • Instruction #DINDIST. • Variables: (V.)A.ACCUDIST.xn

Ref. 1305

Software V04.26	
Cycles of the editor. Taper threading cycle.	New icon to select how to define the end point; coordinates (Xf, Zf), angle and length (α , ΔZ) or angle and end coordinate (α , Zf).

Ref. 1408

Software V05.10	
User cycles.	



CNC 8060
CNC 8065

(REF: 1502)

SAFETY CONDITIONS

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



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

PRECAUTIONS BEFORE CLEANING THE UNIT

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

PRECAUTIONS DURING REPAIR

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

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

PRECAUTIONS AGAINST PERSONAL DAMAGE

- Interconnection of modules.** Use the connection cables provided with the unit.
- Use proper cables.** To prevent risks, only use cables and Sercos fiber recommended for this unit.
To prevent a risk of electrical shock at the central unit, use the proper connector (supplied by Fagor); use a three-prong power cable (one of them being ground).
- Avoid electrical overloads.** To prevent electrical shock and fire risk, do not apply electrical voltage out of the indicated range.
- Ground connection.** In order to avoid electrical discharges, connect the ground terminals of all the modules to the main ground terminal. Before connecting the inputs and outputs of this unit, make sure that all the grounding connections are properly made.
In order to avoid electrical shock, before turning the unit on verify that the ground connection is properly made.
- Do not work in humid environments.** In order to avoid electrical discharges, always work with a relative humidity (non-condensing).
- Do not work in explosive environments.** In order to avoid risks or damages, do no work in explosive environments.



CNC 8060
CNC 8065

(REF: 1502)

PRECAUTIONS AGAINST PRODUCT DAMAGE

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

SAFETY SYMBOLS

Symbols that may appear on the manual.



Danger or prohibition symbol.

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



Warning symbol.

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



Obligation symbol.

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



Information symbol.

This symbol indicates notes, warnings and advises.



Symbol for additional documentation.

This symbol indicates that there is another document with more detailed and specific information.

FAGOR

CNC 8060
CNC 8065

(REF: 1502)

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).



CNC 8060
CNC 8065

(REF: 1502)

WARRANTY TERMS

INITIAL WARRANTY

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

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

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

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

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

EXCLUDING CLAUSES

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

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

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



CNC 8060
CNC 8065

(REF: 1502)

WARRANTY ON REPAIRS

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

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

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

For sold parts the warranty is 12 months length.

SERVICE CONTRACTS

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



CNC 8060
CNC 8065

(REF: 1502)

MATERIAL RETURNING TERMS

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

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



CNC 8060
CNC 8065

(REF: 1502)

CNC MAINTENANCE

CLEANING

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

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

The plastics used on the front panel are resistant to grease and mineral oils, bases and bleach, dissolved detergents and alcohol. Avoid the action of solvents such as chlorine hydrocarbons, 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 manipulate the inside of this unit.

MACHINING CANNED CYCLES (ISO)

1

1.1 General concepts

Some canned cycles are edited in ISO code (described in this chapter) and others are generated at the editor. The canned cycles edited in ISO code are defined using a "G" function and its relevant parameters.

G81	Turning cycle with straight sections.
G82	Facing cycle with straight sections.
G83	Drilling / tapping canned cycle.
G84	Turning cycle with circular sections.
G85	Facing cycle with circular sections.
G86	Longitudinal threading.
G87	Face threading.
G88	Grooving cycle along X axis.
G89	Grooving cycle along Z axis.
G66	Pattern repeat cycle.
G68	Stock removal along the X axis.
G69	Stock removal along the Z axis.

Machining canned cycles with a live tool:

G160	Drilling / tapping canned cycle on the face of the part.
G161	Drilling / tapping canned cycle on the side of the part.
G162	Slot milling canned cycle along the side of the part.
G163	Slot milling canned cycle along the face of the part.

A canned cycle may be defined anywhere in the program, that is, in the main program as well as in a subroutine.

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

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

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

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



CNC 8060
CNC 8065

(REF: 1502)

On a mill model CNC (milling software installed).

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

- #LATHECY ON - To activate the turning canned cycles.
- #LATHECY OFF - To deactivate the turning canned cycles.

G81 ...	Drilling canned cycle.
#LATHECY ON	Activates the turning canned cycles.
G81 ...	
G87 ...	
#LATHECY OFF	Deactivates the turning canned cycles.

On a lathe model CNC (lathe software installed).

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

- #MILLCY ON - To activate the milling canned cycles.
- #MILLCY OFF - To deactivate the milling canned cycles.

G81 ...	Turning canned cycle with straight sections.
#MILLCY ON	Activates the milling canned cycles.
G81 ...	
G86 ...	
#MILLCY OFF	Deactivates the milling canned cycles.

1.

MACHINING CANNED CYCLES (ISO)
General concepts



CNC 8060
CNC 8065

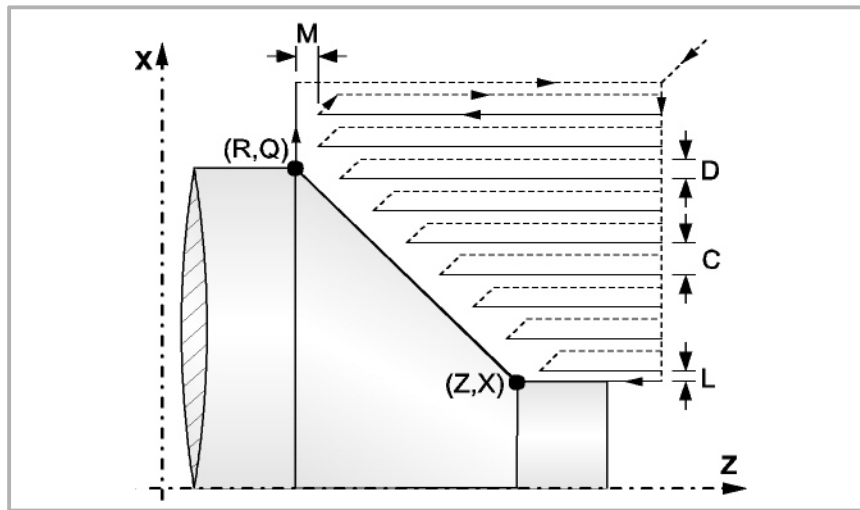
(REF: 1502)

1.2 G81. Turning canned cycle with straight sections

This cycle turns the programmed section maintaining the indicated pass between the successive turning passes. This cycle allows to select whether or not it will run a finishing pass after completing the programmed turning operation.

Programming format in Cartesian coordinates:

G81 X Z Q R C D L M F H



X±5.5 Defines the X coordinate of the initial point of the profile. It must be programmed in absolute values and in the active units (radius or diameter).

Z±5.5 Defines the Z coordinate of the initial point of the profile. It must be programmed in absolute values.

Q±5.5 Defines the X coordinate of the final point of the profile. It must be programmed in absolute values and in the active units (radius or diameter).

R±5.5 Defines the Z coordinate of the final point of the profile.

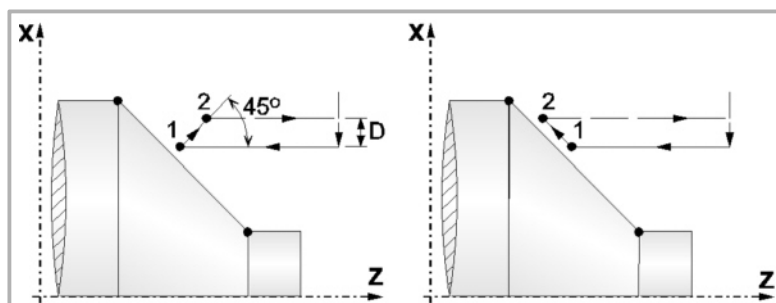
C5.5 Defines the turning pass. It is given by a positive radius value. The whole turning operation is carried out with the same pass which is the same or smaller than the programmed one "C".

If programmed with a 0 value, the CNC will display the corresponding error message.

D 5.5 Defines the safety distance to withdraw the tool after each pass.

- When programming "D" with a value other than "0", the cutter withdraws at a 45° angle up to the safety distance (left drawing).
- When programming "D" with a "0" value, the exit path is the same as the entry path.

When not programming "D", the tool withdrawal is carried out following the profile up to the last pass, "C" distance (right drawing). It must be borne in mind that when "D" is not programmed, the cycle execution time is greater; but the amount of stock to be removed on the finishing pass is less.



1.

MACHINING CANNED CYCLES (ISO)

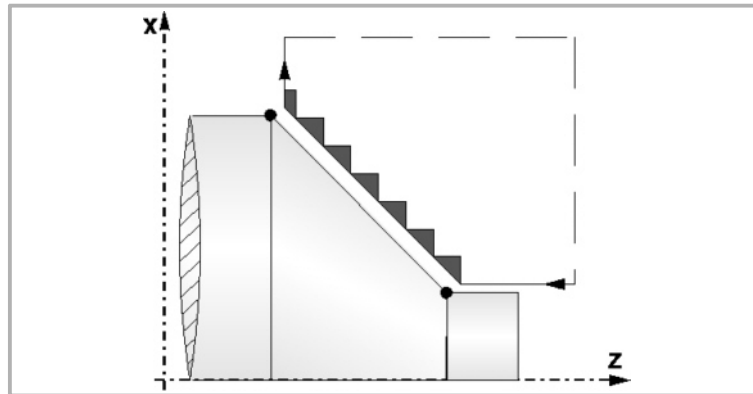
G81. Turning canned cycle with straight sections

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

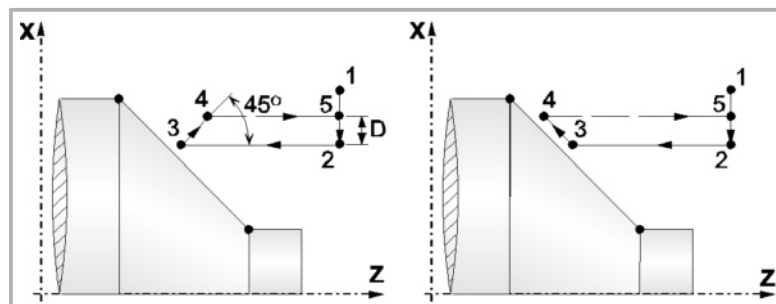
- L5.5 Defines the finishing stock along X and it is given in radius.
If not programmed, a value of 0 is assumed.
- M5.5 Defines the finishing stock along Z.
If not programmed, a value of 0 is assumed.
- F5.5 Defines the feedrate for the final roughing pass. If not programmed or programmed with a value of "0", it means that there is no final roughing pass.



- H5.5 Defines the feedrate for the finishing pass.
If not programmed or programmed with a value of 0, it is considered that no finishing pass is desired.

Machining considerations

- The machining conditions (feedrate, spindle speed, etc.) as well as the tool radius compensation (G41, G42) must be programmed before calling the cycle. Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle.
- The canned cycle will analyze the programmed profile performing, if necessary, a horizontal turning operation until the defined profile is reached.
- The whole turning operation is carried out with the same pass which is the same or smaller than the programmed one "C".
- Each turning pass is carried out as follows:



"1-2" move in rapid (G00). Move "2-3" in G01 at programmed feedrate (F).

If "D" has been programmed, the 3-4 move is done in rapid (G00); but if "D" has not been programmed, the 3-4 move is done following the profile in G01 at the programmed feedrate (F).

The withdrawal "4-5" is done in rapid (G00).

1.

MACHINING CANNED CYCLES (ISO)

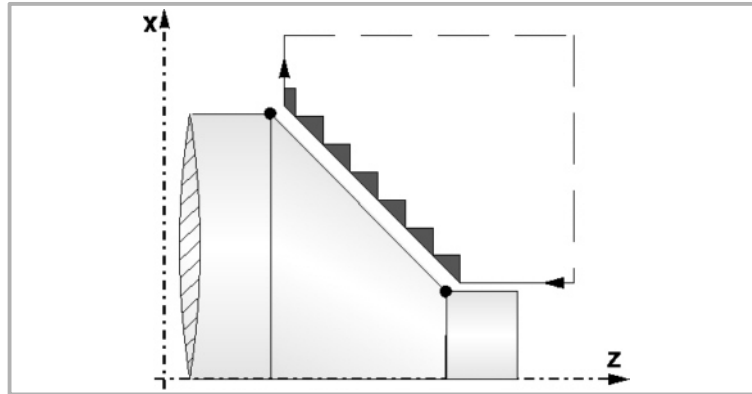
G81. Turning canned cycle with straight sections



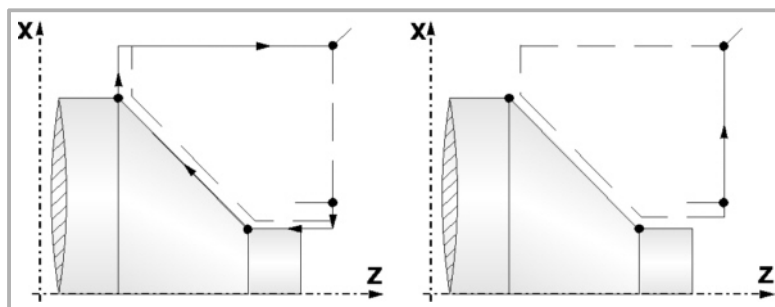
CNC 8060
CNC 8065

(REF: 1502)

- If a final roughing pass has been programmed, the CNC will run a pass parallel to the profile maintaining an "L" and "M" stock at the set feedrate "F". This final roughing pass will eliminate the ridges that were left in the roughing stage.



- After the turning operation (with or without finishing pass) the canned cycle will always end at the cycle calling point.



1.

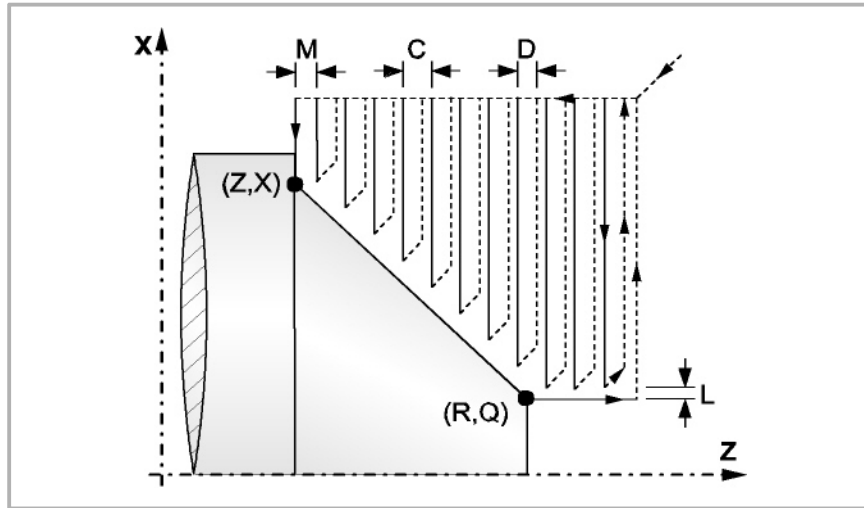
MACHINING CANNED CYCLES (ISO)
G81. Turning canned cycle with straight sections

1.3 G82. Facing canned cycle with straight sections

This cycle faces the programmed section maintaining the indicated pass between the successive facing passes. This cycle allows to select whether or not it will run a finishing pass after completing the programmed facing operation.

Programming format in Cartesian coordinates:

G82 X Z Q R C D L M F H

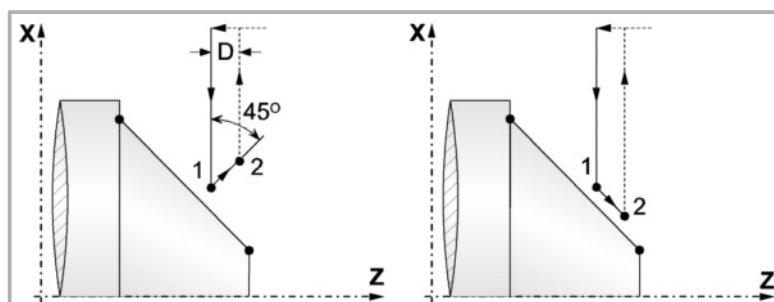


- $X_{\pm 5.5}$ Defines the X coordinate of the initial point of the profile. It must be programmed in absolute values and in the active units (radius or diameter).
- $Z_{\pm 5.5}$ Defines the Z coordinate of the initial point of the profile. It must be programmed in absolute values.
- $Q_{\pm 5.5}$ Defines the X coordinate of the final point of the profile. It must be programmed in absolute values and in the active units (radius or diameter).
- $R_{\pm 5.5}$ Defines the Z coordinate of the final point of the profile.
- $C_{5.5}$ Defines the turning pass. It is given by a positive radius value. The whole turning operation is carried out with the same pass which is the same or smaller than the programmed one "C".

If programmed with a 0 value, the CNC will display the corresponding error message.

- $D_{5.5}$ Defines the safety distance to withdraw the tool after each pass.
- When programming "D" with a value other than "0", the cutter withdraws at a 45° angle up to the safety distance (left drawing).
 - When programming "D" with a "0" value, the exit path is the same as the entry path.

When not programming "D", the tool withdrawal is carried out following the profile up to the last pass, "C" distance (right drawing). It must be borne in mind that when "D" is not programmed, the cycle execution time is greater; but the amount of stock to be removed on the finishing pass is less.



1.

MACHINING CANNED CYCLES (ISO)

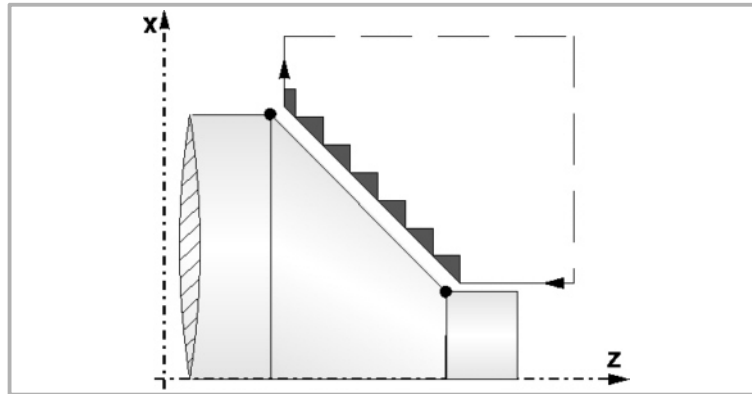
G82. Facing canned cycle with straight sections

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

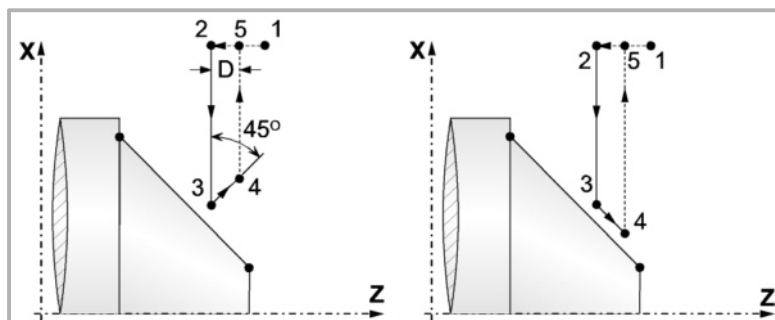
- L5.5 Defines the finishing stock along X and it is given in radius.
If not programmed, a value of 0 is assumed.
- M5.5 Defines the finishing stock along Z.
If not programmed, a value of 0 is assumed.
- F5.5 Defines the feedrate for the final roughing pass. If not programmed or programmed with a value of "0", it means that there is no final roughing pass.



- H5.5 Defines the feedrate for the finishing pass.
If not programmed or programmed with a value of 0, it is considered that no finishing pass is desired.

Machining considerations

- The machining conditions (feedrate, spindle speed, etc.) as well as the tool radius compensation (G41, G42) must be programmed before calling the cycle. Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle.
- The canned cycle will analyze the programmed profile performing, if necessary, a facing operation until the defined profile is reached.
- The whole facing operation is carried out with the same pass, this being equal to or smaller than the programmed one (C).
- Each facing pass is carried out in the following manner:

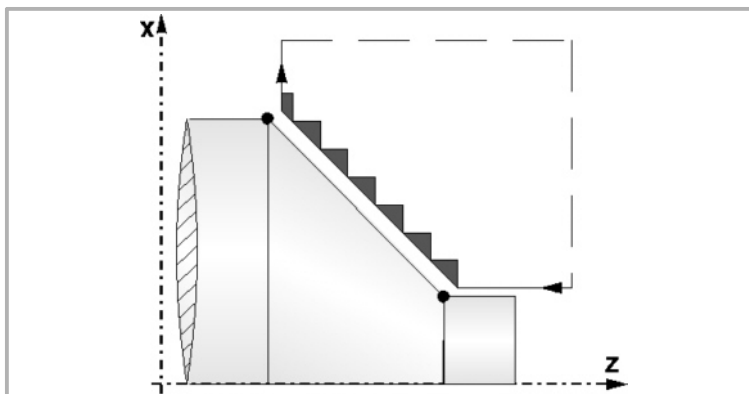


"1-2" move in rapid (G00). Move "2-3" in G01 at programmed feedrate (F).

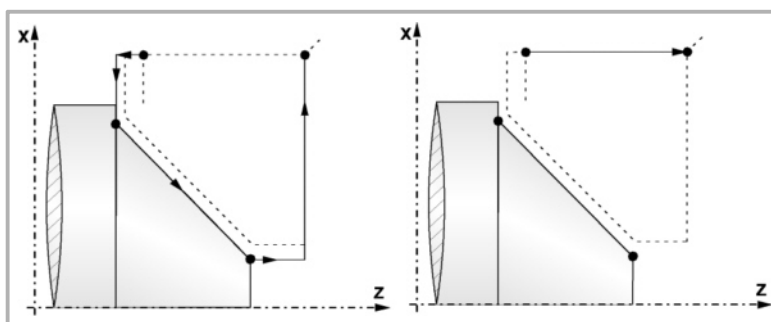
If "D" has been programmed, the 3-4 move is done in rapid (G00); but if "D" has not been programmed, the 3-4 move is done following the profile in G01 at the programmed feedrate (F).

The withdrawal "4-5" is done in rapid (G00).

- If a final roughing pass has been programmed, the CNC will run a pass parallel to the profile maintaining an "L" and "M" stock at the set feedrate "F". This final roughing pass will eliminate the ridges that were left in the roughing stage.



- After the facing operation (with or without finishing pass) the canned cycle will always end at the cycle calling point.



1.

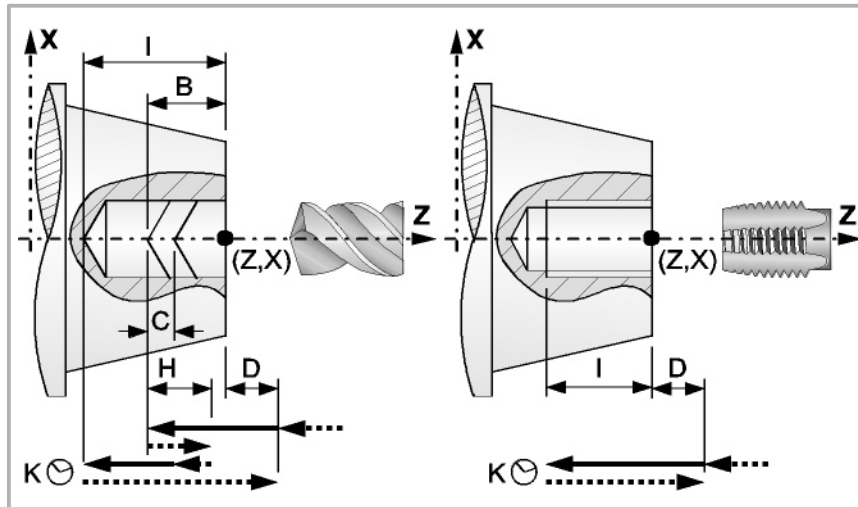
MACHINING CANNED CYCLES (ISO)

G82. Facing canned cycle with straight sections

1.4 G83. Drilling / tapping canned cycle

This cycle may be used to carry out axial drilling or axial tapping. Whether one operation or the other will be performed depends on the format being used.

For tapping, set parameter "B=0", and for axial drilling, set "B>0".



Programming format in Cartesian coordinates:

Axial drilling	G83 X Z I B D K H C R
Axial tapping	G83 X Z I B0 D K R

X±5.5 Defines the X coordinate of the initial point of the profile. It must be programmed in absolute values and in the active units (radius or diameter).

Z±5.5 Defines the Z coordinate of the initial point of the profile. It must be programmed in absolute values.

I±5.5 Defines the total depth. It is referred to the drilling point (X, Z); thus, it will have a positive value when drilling in the negative Z direction and vice versa.

If programmed with a 0 value, the CNC will display the corresponding error message.

B5.5 Defines the type of operation to be run.

- If B=0, the CNC will carry out an axial tapping.
- If B>0 it will carry out an axial drilling where B indicates the drilling step.

D 5.5 Defines the safety distance and it indicates at what distance the tool is positioned in the approach move. If not programmed, a value of 0 is assumed.

K5 Defines the dwell, in hundredths of a second, at the bottom of the hole before starting the withdrawal. If not programmed, a value of 0 is assumed.

H5.5 Defines the withdrawal distance after each drilling step (in G00). If not programmed or programmed a value of 0, it will withdraw to the approach point.

C5.5 Defines the approach distance along Z from the previous drilling step to start the next drilling step (in G00). If not programmed, a value of 1 millimeter.

1.

MACHINING CANNED CYCLES (ISO)
G83. Drilling / tapping canned cycle

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

1.

MACHINING CANNED CYCLES (ISO)
 G83. Drilling / tapping canned cycle

R5.5 In the drilling cycle, it indicates the step that reduces the drilling peck "B". If it is not programmed or is programmed with a value of "0", it assumes a value of "1".

With $R = 1$, all the drilling pecks will be the same (the programmed "B" value). If R is other than 1, the first drilling peck will be "B", the second one "R B", The third one "R (RB)" and so on. In other words, from the second peck on, the new peck will be the product of the R factor times the previous peck.

In the tapping cycle, it defines the type of tapping. "R0" means tapping with a clutch and "R1" means rigid tapping. If not programmed, it takes the value of "0", tapping with a clutch.

In order to execute a rigid tapping, the spindle must be ready to operate in closed loop; in other words, that it must have a servo drive-motor system with rotary encoder.

Drilling. Basic operation

- 1 Rapid move to the approach point located at a "D" distance from the first hole position.
- 2 First drilling operation. The drilling axis moves in G01 to the programmed incremental depth "D+B".
- 3 Drilling loop. The following steps will be repeated until the programmed depth "I" is reached.

Withdrawal in rapid (G00) a distance "H" or to the approach point. Rapid approach (G00) to a "C" distance from the next peck. New drilling peck at work feedrate (G01) to the next incremental according to B and R.

- 4 Dwell "K" in hundredths of a second at the bottom of the hole, if programmed.
- 5 Withdrawal in rapid (G00) to the approach point.

Tapping with a clutch. Basic operation

- 1 Rapid movement to the approach point located at a safety distance "D" from the tap point.
- 2 Tapping. The drilling axis moves in G01 to the programmed incremental depth "D+B".
- 3 Reverse the spindle turning direction.

If "K" has been programmed, the spindle stops and after the programmed dwell has elapsed, the spindle starts turning in the other way.

- 4 Withdrawal in work feedrate to the approach point.

Rigid tapping. Basic operation

- 1 Tapping takes place at the center of the part (X0). Rapid movement to the approach point located at a safety distance "D" from the tap point.
- 2 Tapping. Tapping to the incremental depth programmed in "D+B".

It is carried out by interpolating the main spindle (that is turning) with the Z axis.

Rigid tapping cannot be interrupted, nor the machining conditions be changed. It is done at 100% of the programmed "S" and "F".

- 3 Reverse the spindle turning direction.

If "K" has been programmed, the spindle stops and after the programmed dwell has elapsed, the spindle starts turning in the other way.

- 4 Withdrawal in work feedrate to the approach point.

When the cycle is completed, the spindle stops (M5).



CNC 8060
CNC 8065

(REF: 1502)

Machining considerations

The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the cycle. Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle. Only the tool radius compensation will be canceled (G40) if it was active.

When tapping (regular or rigid), the general logic output "TAPPING" (M5517) stays active during the execution of the cycle.

1.

MACHINING CANNED CYCLES (ISO)

G83. Drilling / tapping canned cycle

FAGOR 

**CNC 8060
CNC 8065**

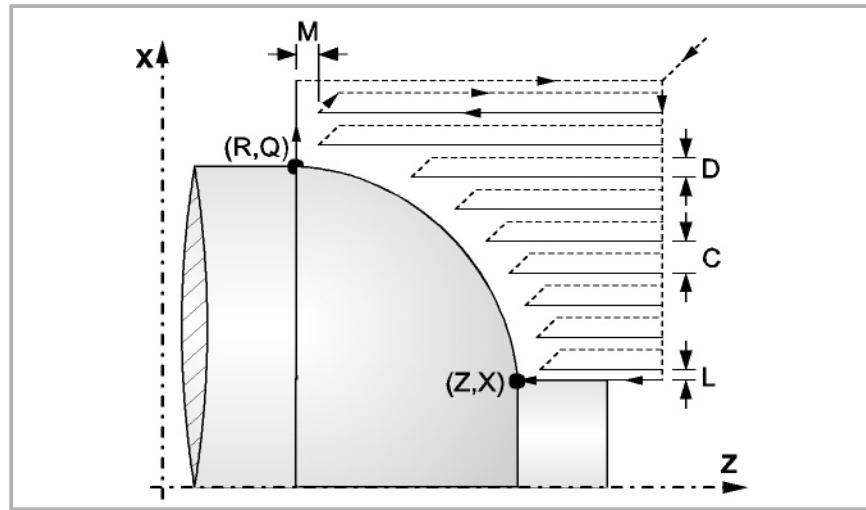
(REF: 1502)

1.5 G84. Turning canned cycle with arcs

This cycle turns the programmed section maintaining the indicated pass between the successive turning passes. It allows to select whether or not the canned cycle will perform a finishing pass after completing the programmed turning operation.

Programming format in Cartesian coordinates:

G84 X Z Q R C D L M F H I K

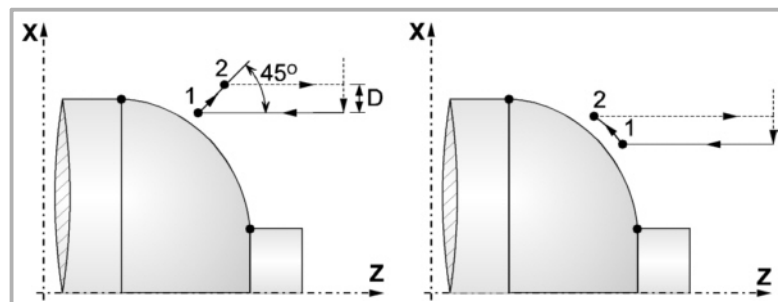


- $X_{\pm 5.5}$ Defines the X coordinate of the initial point of the profile. It must be programmed in absolute values and in the active units (radius or diameter).
- $Z_{\pm 5.5}$ Defines the Z coordinate of the initial point of the profile. It must be programmed in absolute values.
- $Q_{\pm 5.5}$ Defines the X coordinate of the final point of the profile. It must be programmed in absolute values and in the active units (radius or diameter).
- $R_{\pm 5.5}$ Defines the Z coordinate of the final point of the profile.
- $C_{5.5}$ Defines the turning pass. It is given by a positive radius value. The whole turning operation is carried out with the same pass which is the same or smaller than the programmed one "C".

If programmed with a 0 value, the CNC will display the corresponding error message.

- $D_{5.5}$ Defines the safety distance to withdraw the tool after each pass.
- When programming "D" with a value other than "0", the cutter withdraws at a 45° angle up to the safety distance (left drawing).
 - When programming "D" with a "0" value, the exit path is the same as the entry path.

When not programming "D", the tool withdrawal is carried out following the profile up to the last pass, "C" distance (right drawing). It must be borne in mind that when "D" is not programmed, the cycle execution time is greater; but the amount of stock to be removed on the finishing pass is less.



1.

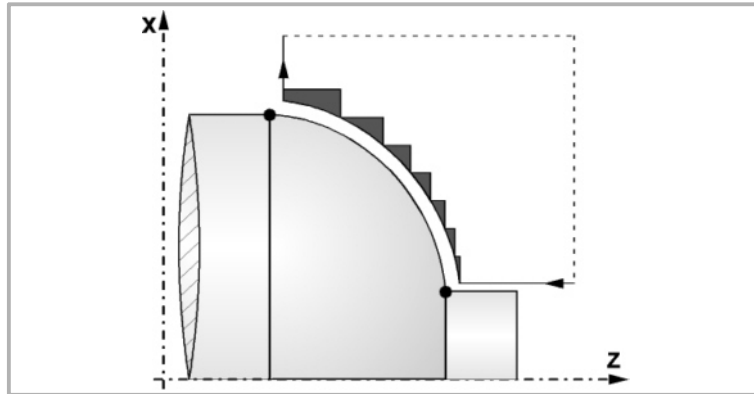
MACHINING CANNED CYCLES (ISO)
G84. Turning canned cycle with arcs

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

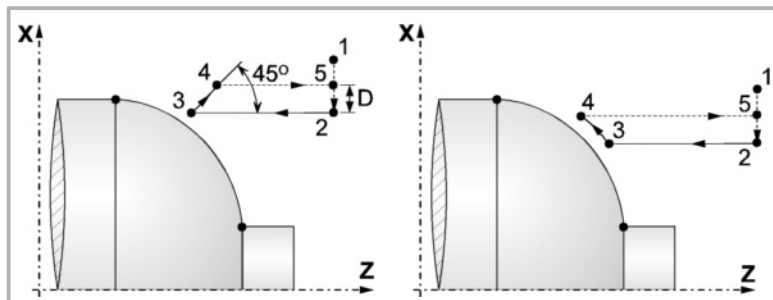
- L5.5 Defines the finishing stock along X and it is given in radius.
If not programmed, a value of 0 is assumed.
- M5.5 Defines the finishing stock along Z.
If not programmed, a value of 0 is assumed.
- F5.5 Defines the feedrate for the final roughing pass. If not programmed or programmed with a value of "0", it means that there is no final roughing pass.



- H5.5 Defines the feedrate for the finishing pass.
If not programmed or programmed with a value of 0, it is considered that no finishing pass is desired.
- I±5.5 Defines the distance, in radius, from the initial point (X, Z) to the arc's center along the X axis. It is programmed in incremental values with respect to the initial point like the "I" for circular interpolations (G02, G03).
- K±5.5 Defines the distance from the initial point (X, Z) to the arc's center along the Z axis. It is programmed in incremental values with respect to the initial point like the "K" for circular interpolations (G02, G03).

Machining considerations

- The machining conditions (feedrate, spindle speed, etc.) as well as the tool radius compensation (G41, G42) must be programmed before calling the cycle. Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle.
- The canned cycle will analyze the programmed profile performing, if necessary, a horizontal turning operation until the defined profile is reached.
- The whole turning operation is carried out with the same pass which is the same or smaller than the programmed one "C".
- Each turning pass is carried out as follows:



- "1-2" move in rapid (G00). Move "2-3" in G01 at programmed feedrate (F).
- If "D" has been programmed, the 3-4 move is done in rapid (G00); but if "D" has not been programmed, the 3-4 move is done following the profile in G01 at the programmed feedrate (F).
- The withdrawal "4-5" is done in rapid (G00).

1.

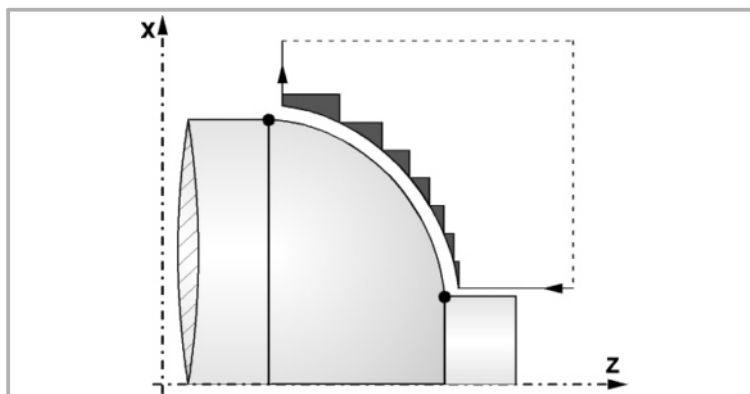
MACHINING CANNED CYCLES (ISO)
 G84. Turning canned cycle with arcs



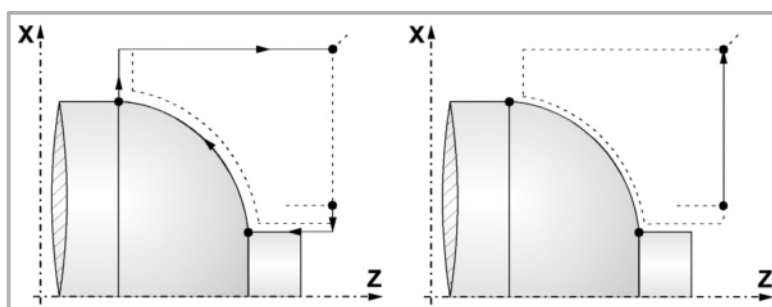
CNC 8060
CNC 8065

(REF: 1502)

- If a final roughing pass has been programmed, the CNC will run a pass parallel to the profile maintaining an "L" and "M" stock at the set feedrate "F". This final roughing pass will eliminate the ridges that were left in the roughing stage.



- After the turning operation (with or without finishing pass) the canned cycle will always end at the cycle calling point.



1.

MACHINING CANNED CYCLES (ISO)

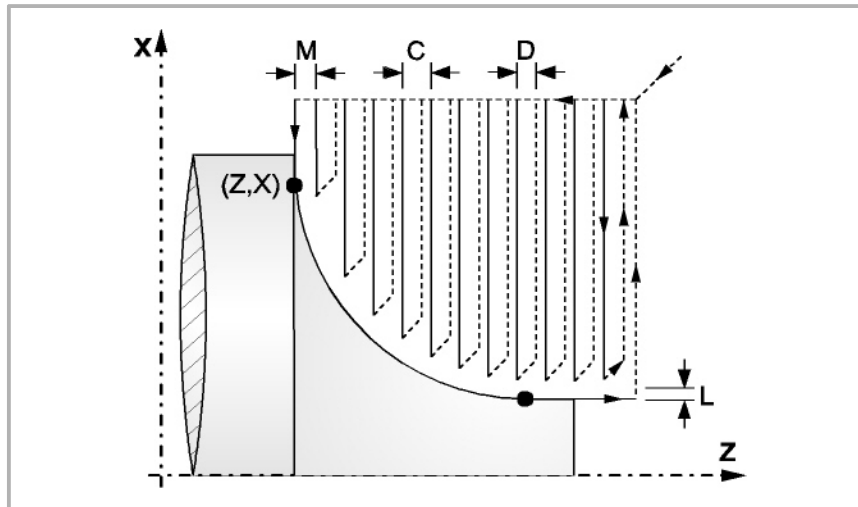
G84. Turning canned cycle with arcs

1.6 G85. Facing canned cycle with arcs

This cycle faces the programmed section maintaining the indicated pass between the successive facing passes. It allows to select whether or not the canned cycle will perform a finishing pass after completing the programmed turning operation.

Programming format in Cartesian coordinates:

G85 X Z Q R C D L M F H I K



X±5.5 Defines the X coordinate of the initial point of the profile. It must be programmed in absolute values and in the active units (radius or diameter).

Z±5.5 Defines the Z coordinate of the initial point of the profile. It must be programmed in absolute values.

Q±5.5 Defines the X coordinate of the final point of the profile. It must be programmed in absolute values and in the active units (radius or diameter).

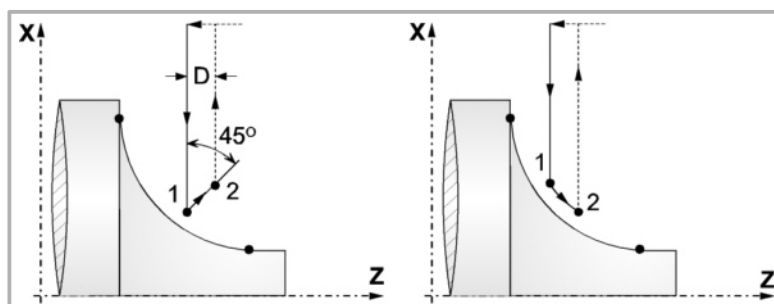
R±5.5 Defines the Z coordinate of the final point of the profile.

C5.5 Defines the facing pass. The whole facing operation is carried out with the same pass, this being equal to or smaller than the programmed one (C).

If programmed with a 0 value, the CNC will display the corresponding error message.

- D 5.5 Defines the safety distance to withdraw the tool after each pass.
- When programming "D" with a value other than "0", the cutter withdraws at a 45° angle up to the safety distance (left drawing).
 - When programming "D" with a "0" value, the exit path is the same as the entry path.

When not programming "D", the tool withdrawal is carried out following the profile up to the last pass, "C" distance (right drawing). It must be borne in mind that when "D" is not programmed, the cycle execution time is greater; but the amount of stock to be removed on the finishing pass is less.



L5.5 Defines the finishing stock along X and it is given in radius.

If not programmed, a value of 0 is assumed.

1.

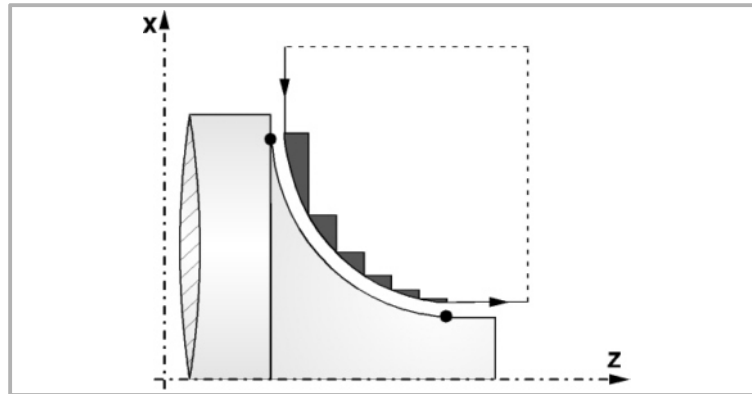
MACHINING CANNED CYCLES (ISO)
G85. Facing canned cycle with arcs



CNC 8060
CNC 8065

(REF: 1502)

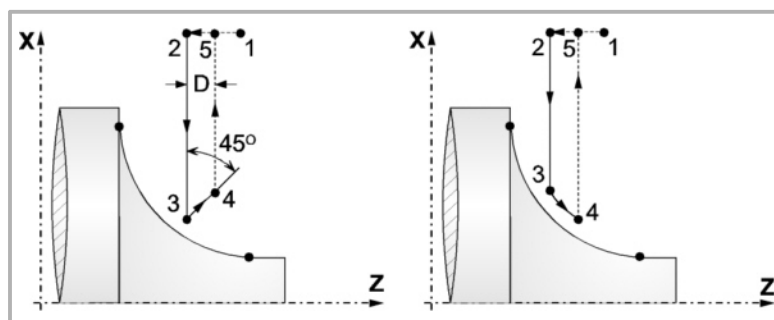
- M5.5 Defines the finishing stock along Z.
If not programmed, a value of 0 is assumed.
- F5.5 Defines the feedrate for the final roughing pass. If not programmed or programmed with a value of "0", it means that there is no final roughing pass.



- H5.5 Defines the feedrate for the finishing pass.
If not programmed or programmed with a value of 0, it is considered that no finishing pass is desired.
- I±5.5 Defines the distance, in radius, from the initial point (X, Z) to the arc's center along the X axis. It is programmed in incremental values with respect to the initial point like the "I" for circular interpolations (G02, G03).
- K±5.5 Defines the distance from the initial point (X, Z) to the arc's center along the Z axis. It is programmed in incremental values with respect to the initial point like the "K" for circular interpolations (G02, G03).

Basic operation

- The machining conditions (feedrate, spindle speed, etc.) as well as the tool radius compensation (G41, G42) must be programmed before calling the cycle. Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle.
- The canned cycle will analyze the programmed profile performing, if necessary, a facing operation until the defined profile is reached.
- The whole facing operation is carried out with the same pass, this being equal to or smaller than the programmed one (C).
- Each facing pass is carried out in the following manner:



"1-2" move in rapid (G00). Move "2-3" in G01 at programmed feedrate (F).

If "D" has been programmed, the 3-4 move is done in rapid (G00); but if "D" has not been programmed, the 3-4 move is done following the profile in G01 at the programmed feedrate (F).

The withdrawal "4-5" is done in rapid (G00).

1.

MACHINING CANNED CYCLES (ISO)

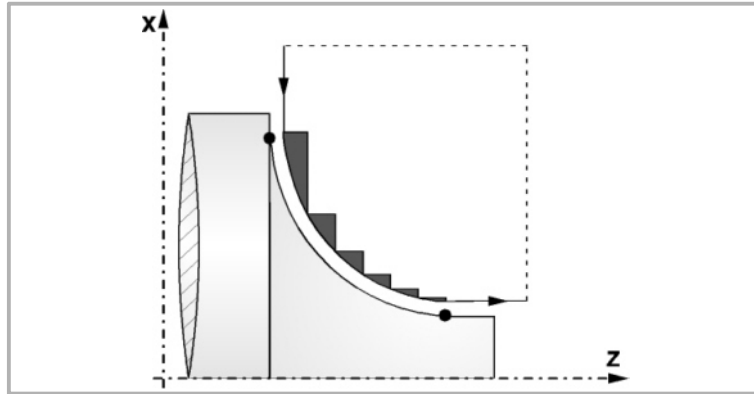
G85. Facing canned cycle with arcs

FAGOR 

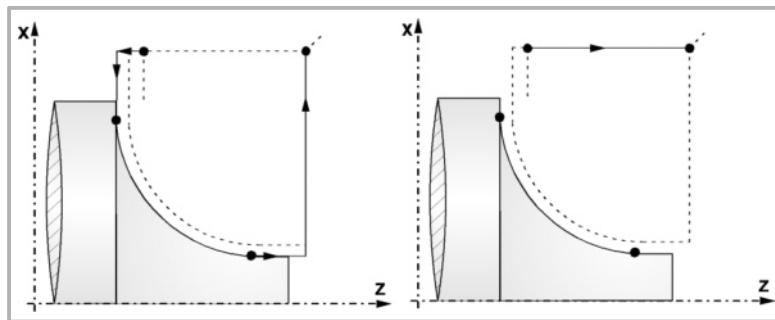
CNC 8060
CNC 8065

(REF: 1502)

- If a final roughing pass has been programmed, the CNC will run a pass parallel to the profile maintaining an "L" and "M" stock at the set feedrate "F". This final roughing pass will eliminate the ridges that were left in the roughing stage.



After the facing operation (with or without finishing pass) the canned cycle will always end at the cycle calling point.



1.

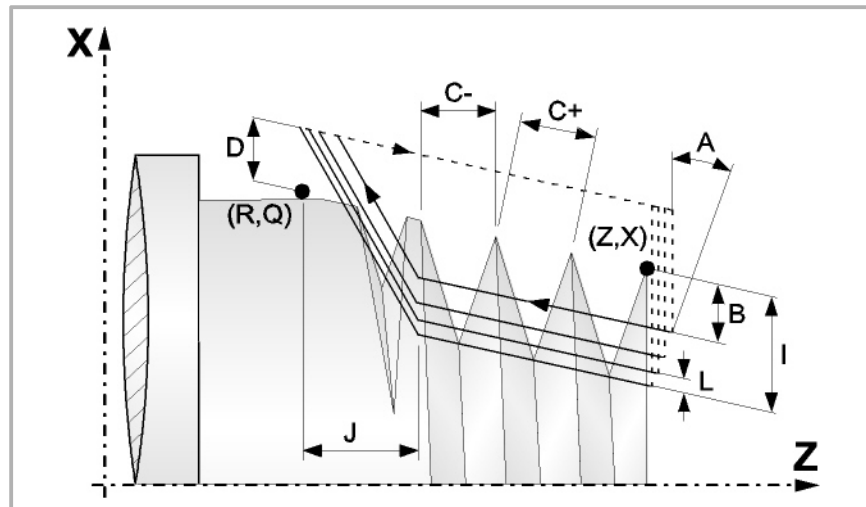
MACHINING CANNED CYCLES (ISO)
G85. Facing canned cycle with arcs

1.7 G86. Longitudinal threading canned cycle

With this cycle it is possible to cut outside, inside and tapered threads of constant pitch.

Programming format in Cartesian coordinates:

G86 X Z Q R K I B E D L C J A W V M H U N



- X±5.5** Defines the X coordinate of the initial point of the thread. It must be programmed in absolute values and in the active units (radius or diameter).
- Z±5.5** Defines the Z coordinate of the initial point of the thread. It must be programmed in absolute values.
- Q±5.5** Defines the X coordinate of the final point of the thread. It must be programmed in absolute values and in the active units (radius or diameter).
- R±5.5** Defines the Z coordinate of the final point of the thread.
- K±5.5** Optional. It is used with parameter "W" for thread repair.
Defines the Z coordinate of the point where the thread is to be measured. Usually it is an intermediate point of the thread.
- I±5.5** Defines the depth of the thread in radius. Its value must be positive for outside threads and negative for inside ones.
If programmed with a 0 value, the CNC will display the corresponding error message.
- B±5.5** Defines the depth of the threading passes and it is given in radius.
If a positive value is programmed, the depth of each pass will depend on the number of the corresponding pass. This way, the penetrations along the X axis are:

$$B, B\sqrt{2}, B\sqrt{3}, B\sqrt{4}, \dots, B\sqrt{n}$$

If a negative value is programmed, the passes will have the same depth with a value equal to the one programmed (B). This way, the penetrations along the X axis are:

$$B, 2B, 3B, 4B, \dots, nB$$

If programmed with a 0 value, the CNC will display the corresponding error message.

Regardless of the sign assigned to "B", when the last roughing pass (before finishing) is smaller than the programmed amount, the canned cycle will run a pass equal to the residual stock.

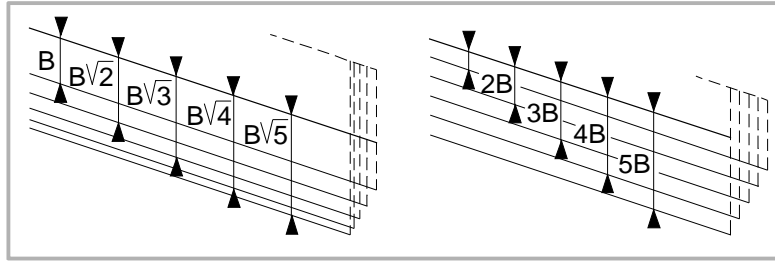
1.

MACHINING CANNED CYCLES (ISO)
G86. Longitudinal threading canned cycle

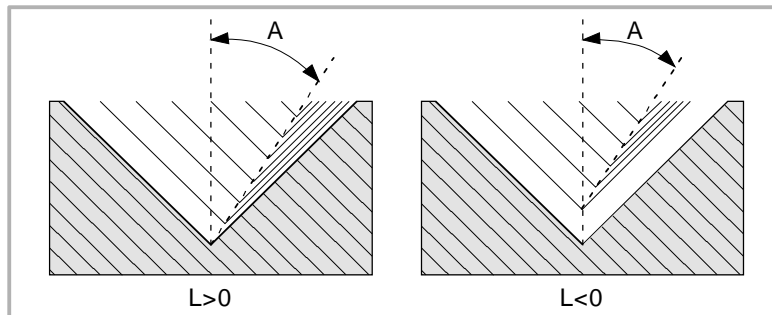


CNC 8060
CNC 8065

(REF: 1502)



- E±5.5** It is related to parameter "B".
- It indicates the minimum value to be reached the penetration pitch when parameter "B" has been programmed with a positive value.
- If not programmed, a value of 0 is assumed.
- D±5.5** Defines the safety distance, along X, and it indicates at what distance from the initial point must the tool be positioned in the approach move. It is programmed in radius.
- The return to the initial point after each pass will be carried out maintaining this distance (D) from the programmed section.
- If the programmed value is positive, this withdrawal will be performed in round corner (G05) and if negative, in square corner (G07).
- If not programmed, a value of 0 is assumed.
- L±5.5** Defines the finishing pass and it is programmed in radius.
- If a positive value is programmed, the finishing pass will be performed maintaining the same penetration angle "A" as for the other passes.
- If a negative value is programmed, the finishing pass will be carried out with radial penetration.
- If a value of 0 is programmed, the previous pass is repeated.



- C5.5** Defines the thread's pitch.
- With a positive sign if the pitch is programmed along the taper
- With a negative sign if the pitch is programmed along the associated axis.
- If programmed with a 0 value, the CNC will display the corresponding error message.
- The right-hand and left-hand threads will be programmed by indicating the direction of the spindle rotation M03 or M04.

1.

MACHINING CANNED CYCLES (ISO)
G86. Longitudinal threading canned cycle



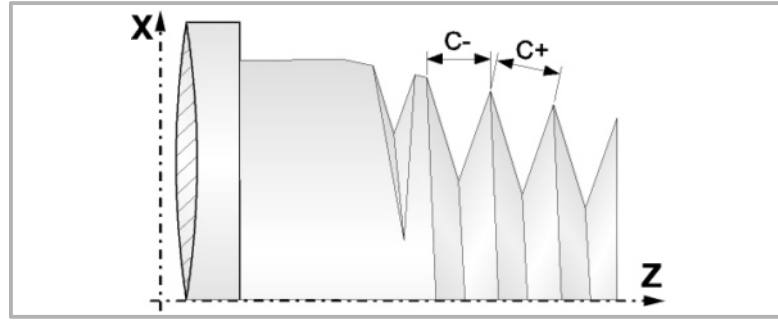
CNC 8060
CNC 8065

(REF: 1502)

1.

MACHINING CANNED CYCLES (ISO)

G86. Longitudinal threading canned cycle

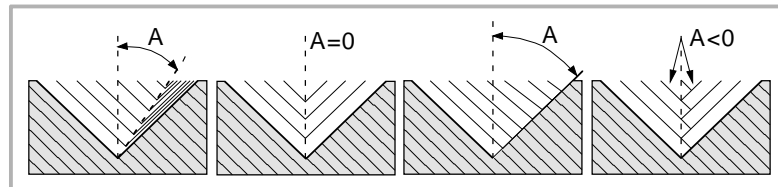


J5.5 Exit from the thread. It defines at what distance from the final point (R, Q), along the Z axis, starts the withdrawal from the thread.

If not programmed, a value of 0 is assumed.

A±5.5 Tool penetration angle, referred to the X axis and if not programmed, a value of 30° will be taken. If A=0 is programmed, the thread will be made with radial penetration. If A is programmed with a negative value, the penetration will be done alternating the flanks of the thread.

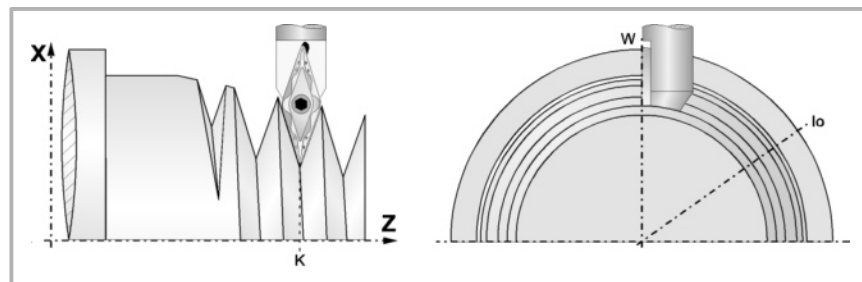
We recommend to use a penetration angle smaller than half the tool angle, never larger. The thread cannot be machined if the penetration angle is larger than half the tool angle. If the penetration angle is equal to half the tool angle, it will rub the flank of the thread at each cutting pass.



W±5.5 Optional. Its meaning depends on parameter "K".

If parameter "K" has not been defined, it indicates the angular position of the spindle corresponding to the thread's starting point. With this, it is possible to make multiple entry threads.

If "K" has been defined, it is a thread repair cycle. It indicates the angular spindle position corresponding to the thread measuring point.



V±5.5 Optional. Number of thread entries (starts).

If not programmed or defined with a 0 value, the thread will only have one entry.

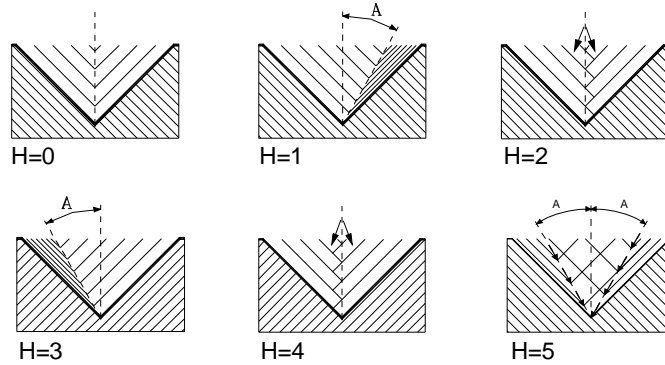
When parameter "W" has been programmed, it means the number of entries (starts) to perform starting at the angular position of the spindle set by that parameter.

The following example shows how to make a 3-entry-point thread.

G86 X Z Q R K I B E D L C J A W0 V3

(REF: 1502)

- M±5.5 Optional. Increment (positive M) or decrement (negative M) of the thread pitch per spindle turn.
 If not programmed or defined with a 0 value, the thread will have a constant pitch.
 This parameter is incompatible with parameter K (thread repair); therefore, if both are programmed together, the CNC will issue the corresponding error message.
- H1 Optional. Thread entry (start) type. If not programmed, the entry to the thread will be defined by parameter A.



- H=0 Radial entry.
 The entry to the thread will be radial, even if the value of parameter A is not 0.
- H=1 Entry by starting flank.
 The entry to the thread will be by starting flank and with the absolute value of the angle given in parameter A.
- H=2 Entry in radial zig-zag radial from the starting center.
 The entry to the thread will be in radial zig-zag, depending on the absolute value of the angle given in parameter A, alternating between a radial entry and an entry slightly towards the starting flank.
- H=3 Entry by final flank.
 The entry to the thread will be by final flank and with the absolute value of the angle given in parameter A.
- H=4 Entry in radial zig-zag radial from the final center.
 The entry to the thread will be in radial zig-zag, depending on the absolute value of the angle given in parameter A, alternating between a radial entry and an entry slightly towards the final flank.
- H=5 Entry in zig-zag by the flanks.
 The entry to the thread will be radial zig-zag by the flanks, depending on the absolute value of the angle given in parameter A, alternating between a starting flank and the final flank.
 Alternating flank-flank, instead of center-flank, extends tool life vs using radial zig-zag entries (H=2 and H=4).
 To do this type of entry, the tool being used must have the shape of the thread to be obtained so it fits perfectly at the bottom of the thread. Before executing this cycle, it may be necessary to pre-empty the thread with a regular trapezoidal tool. This way, this cycle will eliminate the material remaining on the flanks of the thread.

1.

MACHINING CANNED CYCLES (ISO)
 G86. Longitudinal threading canned cycle



CNC 8060
 CNC 8065

(REF: 1502)

- U1 Optional. Partial thread repair (parameters K V) or machining a thread by entering directly into it without doing it from the outside (parameter "K"). If not programmed, a value of 0 is assumed.
- When repairing part of a leadscrew by welding, the leadscrew must be repaired using the thread repair cycle. Instead of repairing the whole leadscrew, this parameter may be used to only repair the portion of the leadscrew that needs repair, thus saving time.
 - To machine a thread by entering directly into it, an entry point and an exit point must be defined in intermediate points between the beginning and the exit of the leadscrew, where the the tool goes in and comes out following an oblique path with respect to the part so the thread is not damaged during the operation.
- U=0 Full threading or 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.
- U=1 Partial threading or 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.
- If the thread exit is programmed, it will be respected; otherwise, the cycle will calculate the exit path in a similar way as that of the entry.
- N1 Blind thread.
- N=0 Normal thread. Parameter J indicates at what distance from the end of the thread the tool begins exiting it.
- N=1 Blind thread. The cycle finishes the thread trying to maintain the pitch until the final point is reached and then it exits the thread. This option ignores parameter J.

Basic operation

- 1 Rapid move (G00) to the approach point located at a safety distance "D" from the initial point (X, Z).
- 2 Threading loop. The following steps will be repeated until the finishing coordinate is reached; programmed depth "I" minus the finishing stock "L".
Rapid move (G00) to the programmed depth indicated by "B". This move will be made according to the penetration angle of the selected tool (A).
It threads the programmed section with the selected exit (J). 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%.
Withdrawal in rapid (G00) to the approach point.
- 3 Thread finishing. Rapid move (G00) to the programmed depth coordinate "I".
This move will be radial or according to the tool penetration angle (A) which depends on the sign assigned to parameter "L".
- 4 It threads the programmed section with the selected exit (J).
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 Withdrawal in rapid (G00) to the approach point.

1.

MACHINING CANNED CYCLES (ISO)
G86. Longitudinal threading canned cycle



CNC 8060
CNC 8065

(REF: 1502)

Machining considerations

The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the cycle. Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle. Only the tool radius compensation will be canceled (G40) if it was active.

Considerations about thread repair.

To do a thread repair, proceed as follows:

- 1 Home (reference) the spindle.
- 2 Measure the angle of the root of the thread, K W parameters.
- 3 Define the G87 cycle for thread repair.
- 4 Execute the canned cycle.

1.

MACHINING CANNED CYCLES (ISO)

G86. Longitudinal threading canned cycle

FAGOR 

**CNC 8060
CNC 8065**

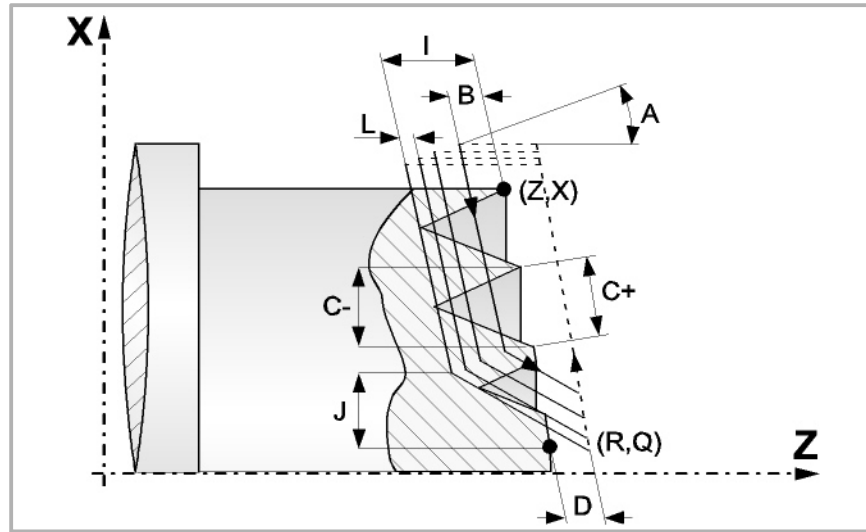
(REF: 1502)

1.8 G87. Face threading canned cycle

With this cycle it is possible to cut outside and inside threads of constant face pitch.

Programming format in Cartesian coordinates:

G87 XZQRKIBEDLCJAWVMHUN



X±5.5 Defines the X coordinate of the initial point of the thread. It must be programmed in absolute values and in the active units (radius or diameter).

Z±5.5 Defines the Z coordinate of the initial point of the thread. It must be programmed in absolute values.

Q±5.5 Defines the X coordinate of the final point of the thread. It must be programmed in absolute values and in the active units (radius or diameter).

R±5.5 Defines the Z coordinate of the final point of the thread.

K±5.5 Optional. It is used with parameter "W" for thread repair.

Defines the Z coordinate of the point where the thread is to be measured. Usually it is an intermediate point of the thread.

I±5.5 Defines the depth of the thread. Its value will be positive if threading in the negative Z direction and vice versa.

If programmed with a 0 value, the CNC will display the corresponding error message.

B±5.5 Defines the depth of the threading passes and it is given in radius.

If a positive value is programmed, the depth of each pass will depend on the number of the corresponding pass. This way, the penetrations along the Z axis are:

$$B, B\sqrt{2}, B\sqrt{3}, B\sqrt{4}, \dots, B\sqrt{n}$$

If a negative value is programmed, the passes will have the same depth with a value equal to the one programmed (B). This way, the penetrations along the Z axis are:

$$B, 2B, 3B, 4B, \dots, nB$$

If programmed with a 0 value, the CNC will display the corresponding error message.

Regardless of the sign assigned to "B", when the last roughing pass (before finishing) is smaller than the programmed amount, the canned cycle will run a pass equal to the residual stock.

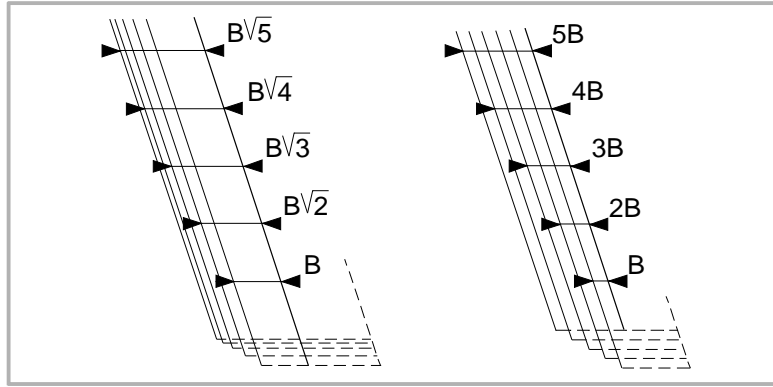
1.

MACHINING CANNED CYCLES (ISO)
G87. Face threading canned cycle

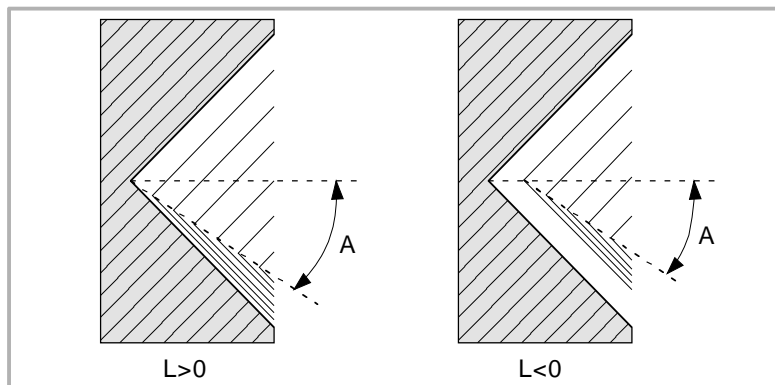
FAGOR 

CNC 8060
CNC 8065

(REF: 1502)



- E±5.5** It is related to parameter "B".
 It indicates the minimum value to be reached the penetration pitch when parameter "B" has been programmed with a positive value.
 If not programmed, a value of 0 is assumed.
- D±5.5** Defines the safety distance and it indicates at what distance, along Z, from the initial point must the tool be positioned in the approach move.
 The return to the initial point after each pass will be carried out maintaining this distance (D) from the programmed section.
 If the programmed value is positive, this withdrawal will be performed in round corner (G05) and if negative, in square corner (G07).
 If not programmed, a value of 0 is assumed.
- L±5.5** Defines the finishing pass
 If a positive value is programmed, the finishing pass will be performed maintaining the same penetration angle "A" as for the other passes.
 If a negative value is programmed, the finishing pass will be carried out with radial penetration.
 If a value of 0 is programmed, the previous pass is repeated.



- C5.5** Defines the thread's pitch.
 With a positive sign if the pitch is programmed along the taper
 With a negative sign if the pitch is programmed along the associated axis.
 If programmed with a 0 value, the CNC will display the corresponding error message.
 The right-hand and left-hand threads will be programmed by indicating the direction of the spindle rotation M03 or M04.

1.

MACHINING CANNED CYCLES (ISO)
 G87. Face threading canned cycle



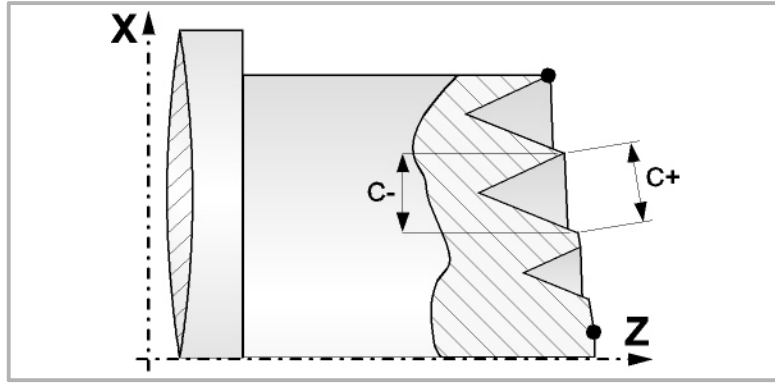
CNC 8060
 CNC 8065

(REF: 1502)

1.

MACHINING CANNED CYCLES (ISO)

G87. Face threading canned cycle

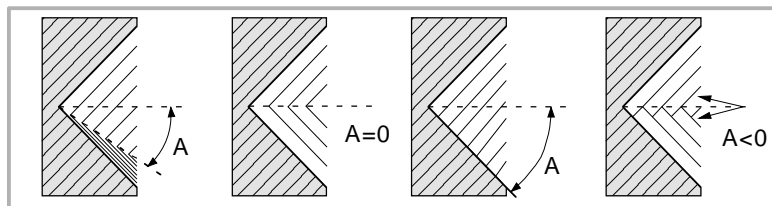


J5.5 Exit from the thread. It defines at what distance from the final point (R, Q), along the X axis, starts the withdrawal from the thread.

It is defined in radius and if not programmed, a value of 0 is assumed.

A±5.5 Tool penetration angle, referred to the Z axis and if not programmed, a value of 30° will be taken. If A=0 is programmed, the thread will be made with axial penetration. If A is programmed with a negative value, the penetration will be done alternating the flanks of the thread.

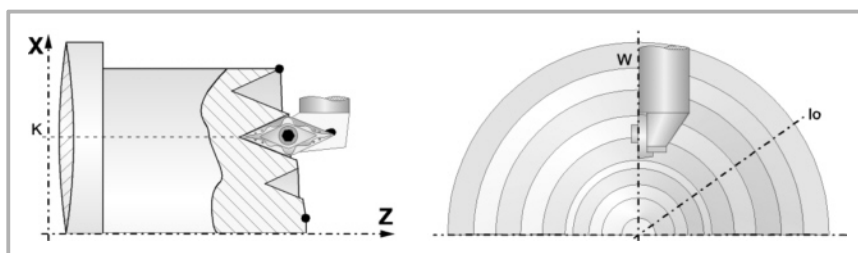
We recommend to use a penetration angle smaller than half the tool angle, never larger. The thread cannot be machined if the penetration angle is larger than half the tool angle. If the penetration angle is equal to half the tool angle, it will rub the flank of the thread at each cutting pass.



W±5.5 Optional. Its meaning depends on parameter "K".

If parameter "K" has not been defined, it indicates the angular position of the spindle corresponding to the thread's starting point. With this, it is possible to make multiple entry threads.

If "K" has been defined, it is a thread repair cycle. It indicates the angular spindle position corresponding to the thread measuring point.



V±5.5 Optional. Number of thread entries (starts).

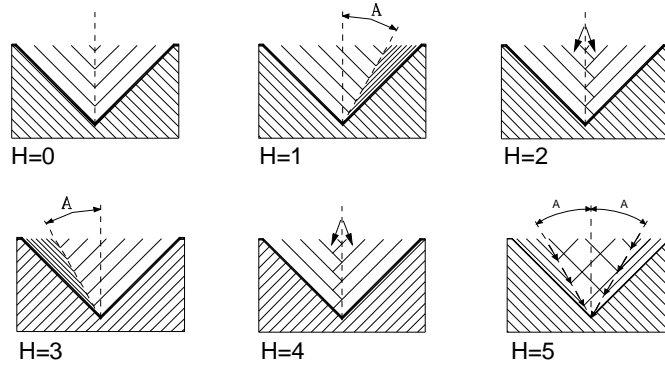
If not programmed or defined with a 0 value, the thread will only have one entry.

When parameter "W" has been programmed, it means the number of entries (starts) to perform starting at the angular position of the spindle set by that parameter.

The following example shows how to make a 3-entry-point thread.

G87 X Z Q R K I B E D L C J A W0 V3

- M±5.5 Optional. Increment (positive M) or decrement (negative M) of the thread pitch per spindle turn.
 If not programmed or defined with a 0 value, the thread will have a constant pitch.
 This parameter is incompatible with parameter K (thread repair); therefore, if both are programmed together, the CNC will issue the corresponding error message.
- H1 Optional. Thread entry (start) type. If not programmed, the entry to the thread will be defined by parameter A.



- H=0 Radial entry.
 The entry to the thread will be radial, even if the value of parameter A is not 0.
- H=1 Entry by starting flank.
 The entry to the thread will be by starting flank and with the absolute value of the angle given in parameter A.
- H=2 Entry in radial zig-zag radial from the starting center.
 The entry to the thread will be in radial zig-zag, depending on the absolute value of the angle given in parameter A, alternating between a radial entry and an entry slightly towards the starting flank.
- H=3 Entry by final flank.
 The entry to the thread will be by final flank and with the absolute value of the angle given in parameter A.
- H=4 Entry in radial zig-zag radial from the final center.
 The entry to the thread will be in radial zig-zag, depending on the absolute value of the angle given in parameter A, alternating between a radial entry and an entry slightly towards the final flank.
- H=5 Entry in zig-zag by the flanks.
 The entry to the thread will be radial zig-zag by the flanks, depending on the absolute value of the angle given in parameter A, alternating between a starting flank and the final flank.
 Alternating flank-flank, instead of center-flank, extends tool life vs using radial zig-zag entries (H=2 and H=4).
 To do this type of entry, the tool being used must have the shape of the thread to be obtained so it fits perfectly at the bottom of the thread. Before executing this cycle, it may be necessary to pre-empty the thread with a regular trapezoidal tool. This way, this cycle will eliminate the material remaining on the flanks of the thread.

1.

MACHINING CANNED CYCLES (ISO)
 G87. Face threading canned cycle



CNC 8060
 CNC 8065

(REF: 1502)

1.

MACHINING CANNED CYCLES (ISO)
 G87. Face threading canned cycle

- U1 Optional. Partial thread repair (parameters K V) or machining a thread by entering directly into it without doing it from the outside (parameter "K"). If not programmed, a value of 0 is assumed.
- When repairing part of a leadscrew by welding, the leadscrew must be repaired using the thread repair cycle. Instead of repairing the whole leadscrew, this parameter may be used to only repair the portion of the leadscrew that needs repair, thus saving time.
 - To machine a thread by entering directly into it, an entry point and an exit point must be defined in intermediate points between the beginning and the exit of the leadscrew, where the the tool goes in and comes out following an oblique path with respect to the part so the thread is not damaged during the operation.
- U=0 Full threading or 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.
- U=1 Partial threading or 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.
- If the thread exit is programmed, it will be respected; otherwise, the cycle will calculate the exit path in a similar way as that of the entry.
- N1 Blind thread.
- N=0 Normal thread. Parameter J indicates at what distance from the end of the thread the tool begins exiting it.
- N=1 Blind thread. The cycle finishes the thread trying to maintain the pitch until the final point is reached and then it exits the thread. This option ignores parameter J.

Basic operation

- 1 Rapid move (G00) to the approach point located at a safety distance "D" from the initial point (X, Z).
- 2 Threading loop. The following steps will be repeated until the finishing coordinate is reached; programmed depth "I" minus the finishing stock "L".
Rapid move (G00) to the programmed depth indicated by "B". This move will be made according to the penetration angle of the selected tool (A).
It threads the programmed section with the selected exit (J). 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%.
Withdrawal in rapid (G00) to the approach point.
- 3 Thread finishing. Rapid move (G00) to the programmed depth coordinate "I".
This move will be radial or according to the tool penetration angle (A) which depends on the sign assigned to parameter "L".
- 4 It threads the programmed section with the selected exit (J).
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 Withdrawal in rapid (G00) to the approach point.



CNC 8060
CNC 8065

(REF: 1502)

Machining considerations

The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the cycle. Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle. Only the tool radius compensation will be canceled (G40) if it was active.

Considerations about thread repair.

To do a thread repair, proceed as follows:

- 1 Home (reference) the spindle.
- 2 Measure the angle of the root of the thread, K W parameters.
- 3 Define the G87 cycle for thread repair.
- 4 Execute the canned cycle.

1.

MACHINING CANNED CYCLES (ISO)

G87. Face threading canned cycle

FAGOR 

**CNC 8060
CNC 8065**

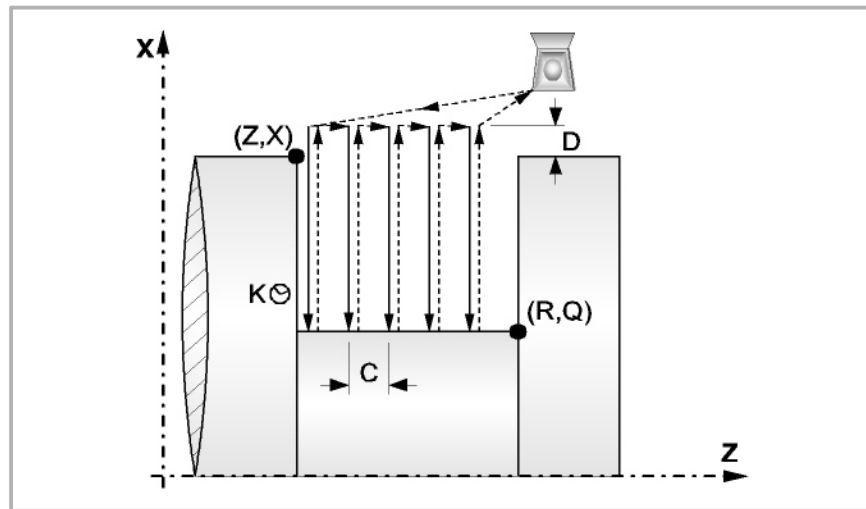
(REF: 1502)

1.9 G88. Grooving canned cycle along the X axis

This cycle performs a grooving operation along the X axis maintaining the same pass between the successive passes being this equal to or smaller than the one programmed.

Programming format in Cartesian coordinates:

G88 X Z Q R C D K



- $X_{\pm 5.5}$ Defines the X coordinate of the initial point of the groove. It must be programmed in absolute values and in the active units (radius or diameter).
- $Z_{\pm 5.5}$ Defines the Z coordinate of the initial point of the groove. It must be programmed in absolute values.
- $Q_{\pm 5.5}$ Defines the X coordinate of the final point of the groove. It must be programmed in absolute values and in the active units (radius or diameter).
If the depth of the groove is 0, the CNC will display the corresponding error message.
- $R_{\pm 5.5}$ Defines the Z coordinate of the final point of the groove.
If the width of the groove is smaller than the width of the cutter (NOSEW), the CNC will display the corresponding error message.
- $C_{5.5}$ Defines the grooving pass.
If not programmed, the cycle will assume the width of the cutter (NOSEW) of the active tool. If programmed with a 0 value, the CNC will display the corresponding error message.
- $D_{5.5}$ Defines the safety distance and it must have a positive value in radius.
- K_5 Defines the dwell, in hundredths of a second, after each penetration until the withdrawal begins.
If not programmed, a value of 0 is assumed.

Machining considerations

- The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the cycle. Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle. Only the tool radius compensation will be canceled (G40) if it was active.
- The whole grooving operation is done with the same pass being this equal to or smaller than "C".
- Each grooving pass is carried out in the following manner:
The penetration is done at programmed feedrate (F). The withdrawal and the move to the next penetration point are done in rapid (G00).
- After completing the grooving operation, the canned cycle will always end at the cycle calling point.

1.

MACHINING CANNED CYCLES (ISO)

G88. Grooving canned cycle along the X axis

FAGOR 

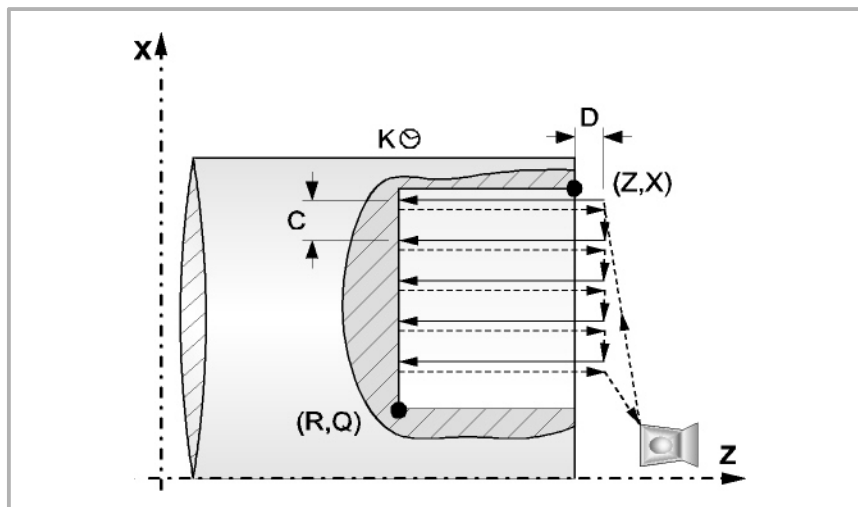
CNC 8060
CNC 8065

(REF: 1502)

1.10 G89. Grooving canned cycle along the Z axis

This cycle performs a grooving operation along the Z axis maintaining the same pass between the successive passes being this equal to or smaller than the one programmed. Programming format in Cartesian coordinates:

G89 X Z Q R C D K



X±5.5 Defines the X coordinate of the initial point of the groove. It must be programmed in absolute values and in the active units (radius or diameter).

Z±5.5 Defines the Z coordinate of the initial point of the groove. It must be programmed in absolute values.

Q±5.5 Defines the X coordinate of the final point of the groove. It must be programmed in absolute values and in the active units (radius or diameter).

If the width of the groove is smaller than the width of the cutter (NOSEW), the CNC will display the corresponding error message.

R±5.5 Defines the Z coordinate of the final point of the groove.

If the depth of the groove is 0, the CNC will display the corresponding error message.

C5.5 Defines the grooving pass. It is programmed in radius.

If not programmed, the cycle will assume the width of the cutter (NOSEW) of the active tool. If programmed with a 0 value, the CNC will display the corresponding error message.

D5.5 Defines the safety distance.

If not programmed, a value of 0 is assumed.

K5 Defines the dwell, in hundredths of a second, after each penetration until the withdrawal begins. If not programmed, a value of 0 is assumed.

Machining considerations

- The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the cycle. Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle. Only the tool radius compensation will be canceled (G40) if it was active.
- The whole grooving operation is done with the same pass being this equal to or smaller than "C".
- Each grooving pass is carried out in the following manner:
The penetration is done at programmed feedrate (F). The withdrawal and the move to the next penetration point are done in rapid (G00).
- After completing the grooving operation, the canned cycle will always end at the cycle calling point.

1.

MACHINING CANNED CYCLES (ISO)

G89. Grooving canned cycle along the Z axis

FAGOR 

CNC 8060
CNC 8065

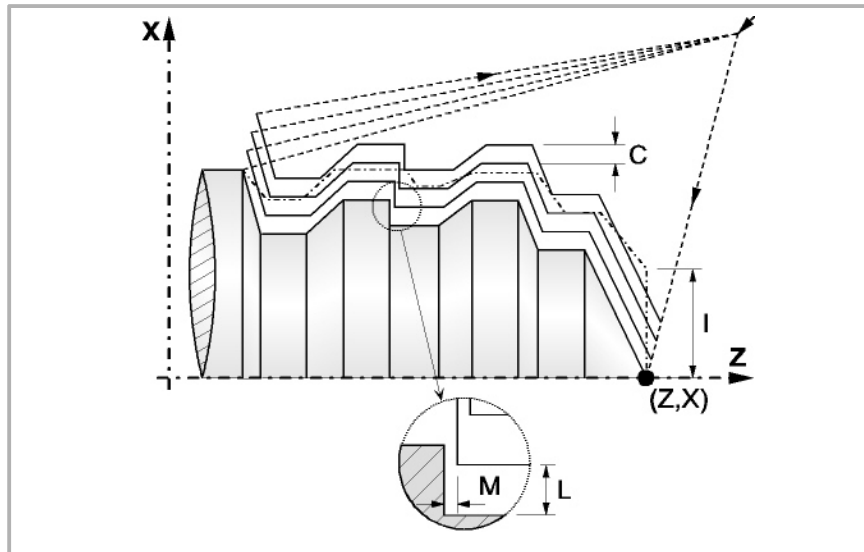
(REF: 1502)

1.11 G66. Pattern repeat canned cycle

Pattern repeat canned cycle, this cycle machines the programmed profile maintaining the specified pass between the successive machining passes. It admits triangular, round and square tools.

Programming format in Cartesian coordinates:

G66 X Z I C A L M J H S E P



X±5.5 Defines the X coordinate of the initial point of the profile. It must be programmed in absolute values and in the active units (radius or diameter).

Z±5.5 Defines the Z coordinate of the initial point of the profile. It must be programmed in absolute values.

I5.5 Defines the residual stock, i.e. the amount to be removed from the original part. It is given in radius and it will be taken as X residual or Z residual depending on the value given to parameter "A".

If its value is not greater than the finishing stock (L or M) and "H" is other than zero, the CNC will only run the finishing pass.

C5.5 Defines the machining pass. It is given in radius and it will be taken as pass along X or along Z, like "I", depending on the value given to parameter "A".

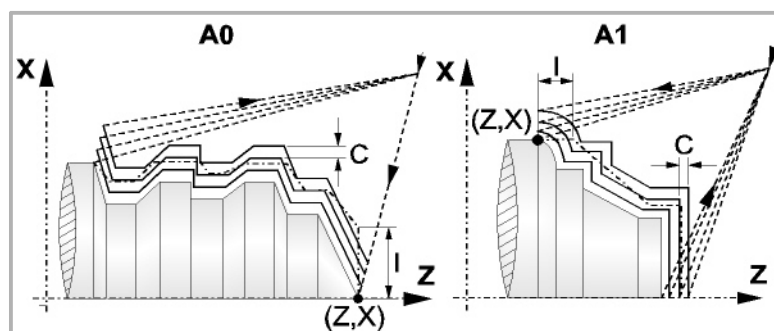
All the machining passes are performed with this value, except the last one which will eliminate the remaining stock.

If programmed with a 0 value, the CNC will display the corresponding error message.

A1 Defines the main machining axis.

When programming "A0", the Z axis will be the main axis, "I" will be taken as X residual value and "C" as the pass along X.

When programming "A1", the X axis will be the main axis, "I" will be taken as Z residual value and "C" as the pass along Z.



1.

MACHINING CANNED CYCLES (ISO)

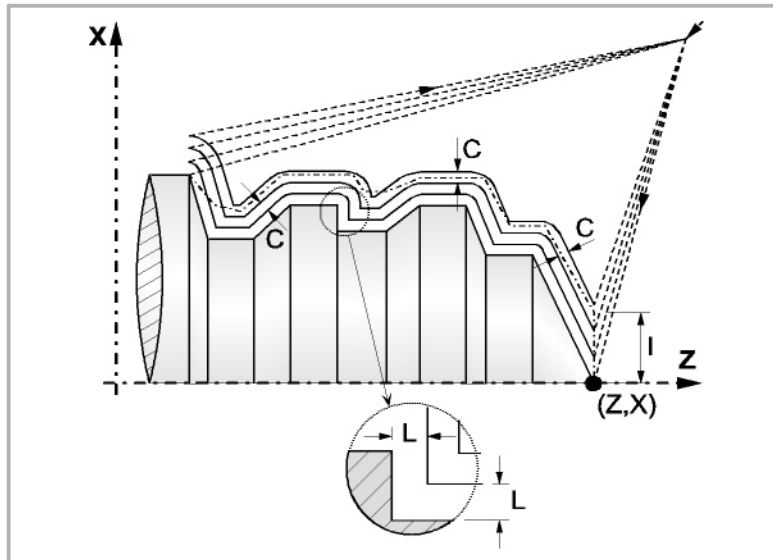
G66. Pattern repeat canned cycle



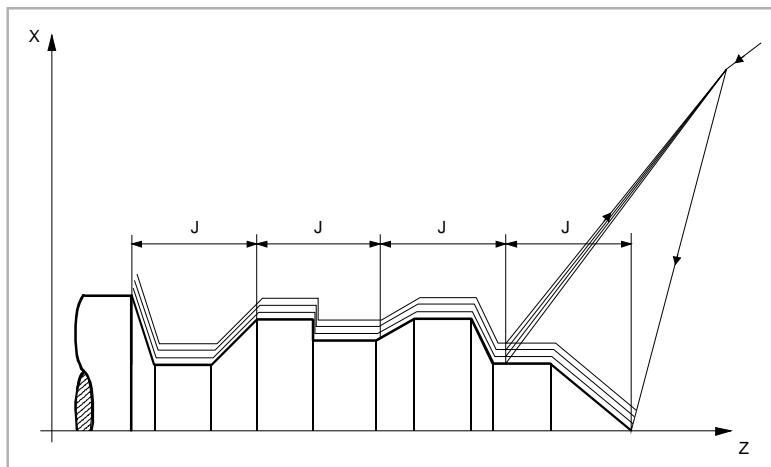
CNC 8060
CNC 8065

(REF: 1502)

- L±5.5 Defines the finishing stock along X. It is defined in radius and if not programmed, a value of 0 is assumed.
 - M±5.5 Defines the finishing stock along Z.
- If "M" is not programmed, the X and Z residual value will be the one indicated by parameter "L" and the roughing passes will be the same maintaining the "C" distance between two consecutive passes.



- J5.5 Optional. Maximum travel in each section on the machining axis. If not programmed, a value of 0 (machining in a single segment) will be assumed. Parameter valid when A0 has been defined and for inside and outside profiles.
- 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.



- H5.5 Defines the feedrate for the finishing pass.
- If not programmed or programmed with a value of 0, it is considered that no finishing pass is desired.
- S4 Label number of the first block describing the geometry of the profile.
 - E4 Label number of the last block describing the geometry of the profile.
- The profile may be defined either in the current program or in any other program (parameter "Q").

1.

MACHINING CANNED CYCLES (ISO)
G66. Pattern repeat canned cycle



CNC 8060
CNC 8065

(REF: 1502)

- P Name of the local subroutine that contains the profile. The local subroutine may be located either in the current program or in any other program (parameter "Q").
- The cycle considers that the whole subroutine makes up the profile; if parameter "P" is programmed, the cycle ignores parameters "E" and "S".
- Q Name of the global subroutine, of the program that contains the definition of the profile (parameters "E" and "S") or of the program that contains the local subroutine that contains the profile (parameter "P").

In short, the profile may be defined in one of the following ways.

Parameters.	Profile location.
E + S	The profile is defined between the "E" and "S" blocks of the current program.
P	The profile is defined in the local subroutine "P" of the current program.
Q	The profile is defined in global subroutine "Q".
Q + E + S	The profile is defined between the "E" and "S" blocks of the program "Q".
Q + P	The profile is defined in the local subroutine "P" of the program "Q".

1.

MACHINING CANNED CYCLES (ISO)

G66. Pattern repeat canned cycle

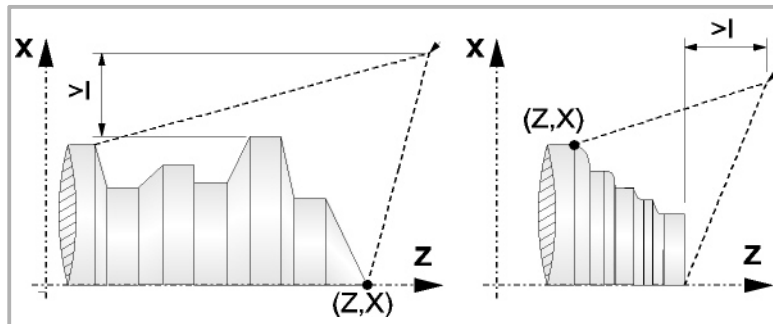


CNC 8060
CNC 8065

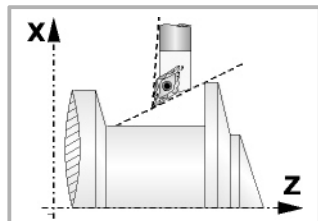
(REF: 1502)

Machining considerations

- The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the canned cycle. Once the canned cycle has ended, the active feedrate will be the last one programmed, i.e. the one corresponding to the roughing operation (F) or the finishing operation (H). The CNC will assume functions G00, G40 and G90.
- The canned cycle calling position must be off the part to be machined and at a distance greater than that defined as residual stock (I) of the outmost profile of the part. If the tool position is not correct to execute the cycle, the CNC will display the corresponding error message.



- In those cases when the programmed profile cannot be machined (valleys) with the selected tool, a warning message will be displayed at the beginning of the cycle execution.



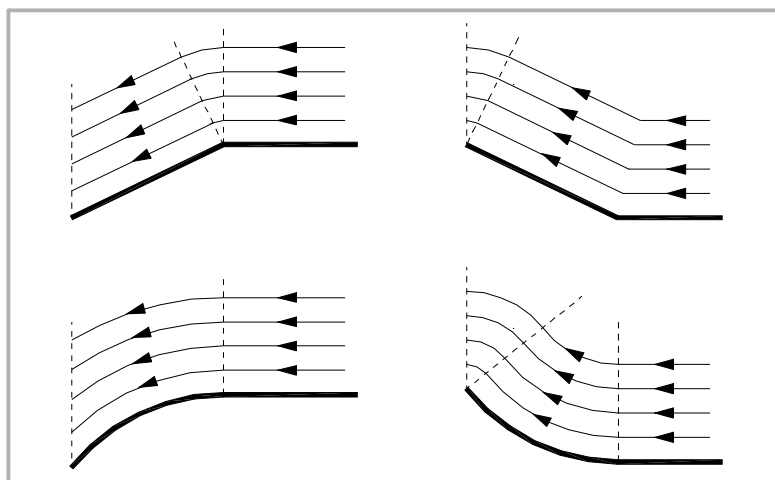
The operator will be able to stop the execution and select the proper tool.

If he/she doesn't do that, a new profile is calculated in the areas that are not accessible for the selected tool and the CNC machines as far as it is possible.

- Once the profile to be executed has been calculated, the CNC will calculate all the passes required to eliminate the programmed residual stock (I).

The machining operation will be performed in whatever G05 or G07 that was active when calling the cycle.

When not programming parameter "M", the passes will be the same maintaining the "C" distance between 2 consecutive passes. Also, if the last section of the profile is an arc or a cone, the CNC will calculate the different passes without exceeding the maximum programmed position.



1.

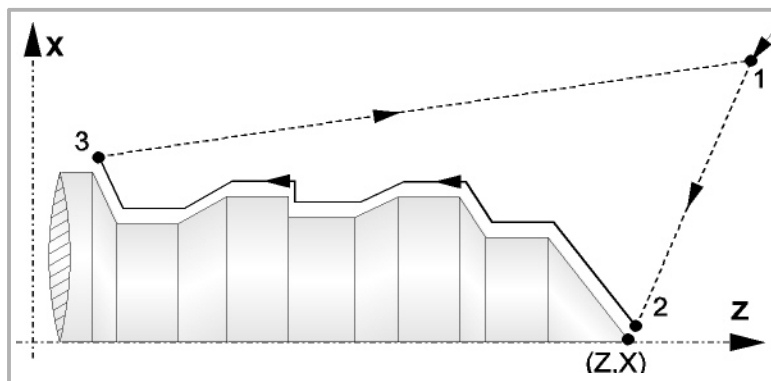
MACHINING CANNED CYCLES (ISO)
G66. Pattern repeat canned cycle

FAGOR 

CNC 8060
CNC 8065

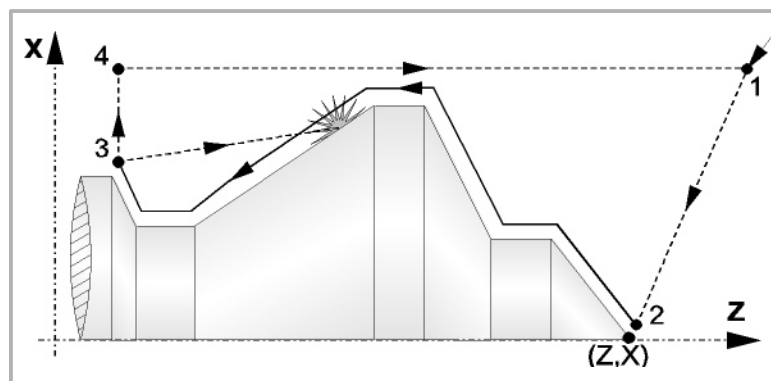
(REF: 1502)

- Each pass is performed in the following way:



The approach move "1-2" is done in rapid (G00). The "2-3" move is done at programmed feedrate (F). The withdrawal "3-1" is done in rapid (G00).

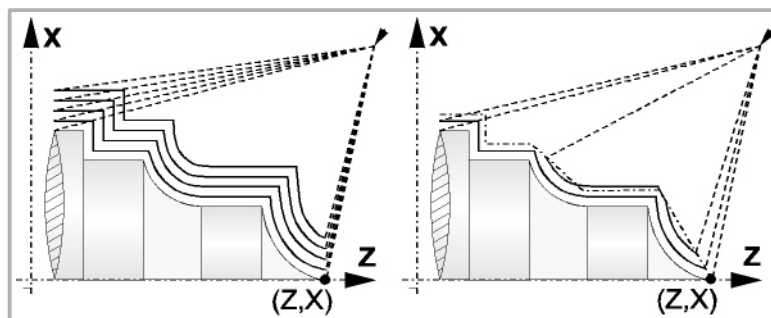
If the possibility of collision exists, the move between points "3-1", it will be carried out in two stages and in G00 ("3-4" and "4-1") as shown in the next figure.



- The canned cycle will always end at the point from where it was called.

Optimizing the machining process

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 part profile is known, it is recommended to define both profiles, that of the rough part and the desired final profile. The machining is faster since only the stock between both profiles is removed.

Profile programming syntax

When defining the profile, there is no need to program the initial point since it is already indicated by cycle defining parameters X and Z.

When defining 2 profiles, the final profile must be defined first and then that of the rough part.

The first and last profile defining blocks must have a label number. These labels will indicate the beginning and end of the profile geometry definition for this canned cycle.

The profile programming syntax must follow these rules:

1.

MACHINING CANNED CYCLES (ISO)

G66. Pattern repeat canned cycle

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

- It may be programmed in absolute or incremental coordinates and it may consist of simple geometrical elements such as straight and curved lines, roundings and chamfers which will be programmed following their own syntax rules.
- Function G00 indicates that the profile definition is done and this block is the beginning of the definition of the rough part profile.
- Program G01, G02 or G03 in the next block, because G00 is modal, thus preventing the CNC from displaying the corresponding error message.
- The profile description must not contain: mirror images, scaling factor changes, pattern rotation or zero offsets.
- It must not contain high level blocks such as jumps, calls to subroutines or parametric programming.
- It must not contain other canned cycles.

Only the following G functions may be programmed in the profile definition:

G01	Linear interpolation
G02	Clockwise circular interpolation
G03	Counter-clockwise circular interpolation
G06	Circle center in absolute coordinates
G08	Arc tangent to previous path
G09	Arc defined by three points
G36	Automatic radius blend
G39	Chamfer
G53	Programming with respect to machine zero
G70	Programming in inches
G71	Programming in millimeters
G90	Absolute programming:
G91	Incremental programming
G30	Polar origin preset

It is also possible to program the following functions although they will be ignored by the canned cycle:

G05	Round corner
G07	Square corner
G50	Controlled corner rounding

The F, S, T, D and M functions



MACHINING CANNED CYCLES (ISO)
G66. Pattern repeat canned cycle



CNC 8060
CNC 8065

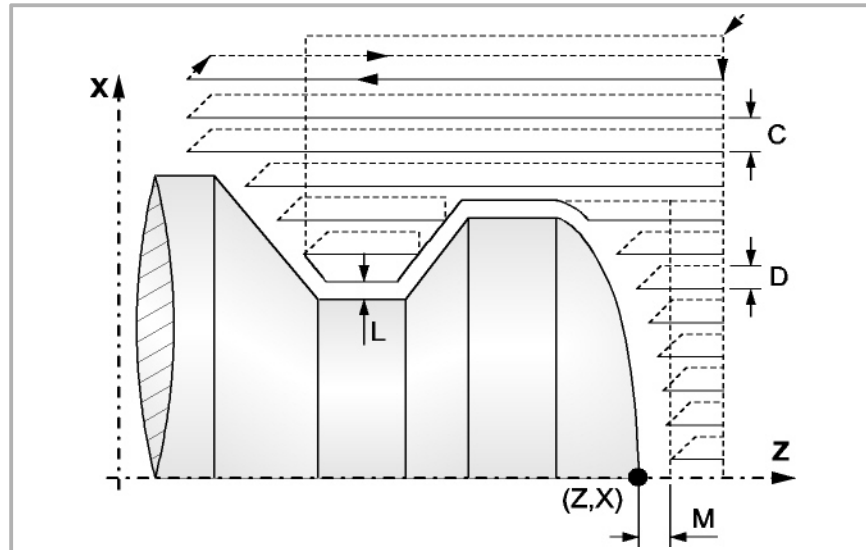
(REF: 1502)

1.12 G68. Stock removal cycle along X axis

Pattern repeat canned cycle, this cycle machines the programmed profile maintaining the specified pass between the successive machining passes. It admits triangular, round and square tools.

Programming format in Cartesian coordinates:

G68 X Z C D L M J K F H S E P



X±5.5 Defines the X coordinate of the initial point of the profile. It must be programmed in absolute values and in the active units (radius or diameter).

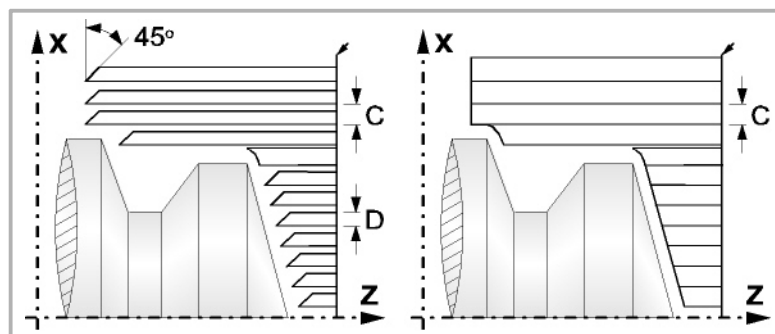
Z±5.5 Defines the Z coordinate of the initial point of the profile. It must be programmed in absolute values.

C5.5 Defines the machining pass. It is given by a positive radius value. If programmed with a 0 value, the CNC will display the corresponding error message.

All the machining passes are performed with this value, except the last one which will eliminate the remaining stock.

- D 5.5 Defines the safety distance to withdraw the tool after each pass.
- When programming "D" with a value other than "0", the cutter withdraws at a 45° angle up to the safety distance (left drawing).
 - When programming "D" with a "0" value, the exit path is the same as the entry path. This may be interesting for grooving complex profiles, to use these cycles on cylindrical grinders, etc.

When not programming "D", the tool withdrawal is carried out following the profile up to the last pass, "C" distance (right drawing). It must be borne in mind that when "D" is not programmed, the cycle execution time is greater; but the amount of stock to be removed on the finishing pass is less.



L±5.5 Defines the finishing stock along X. It is defined in radius and if not programmed, a value of 0 is assumed.

1.

MACHINING CANNED CYCLES (ISO)
G68. Stock removal cycle along X axis

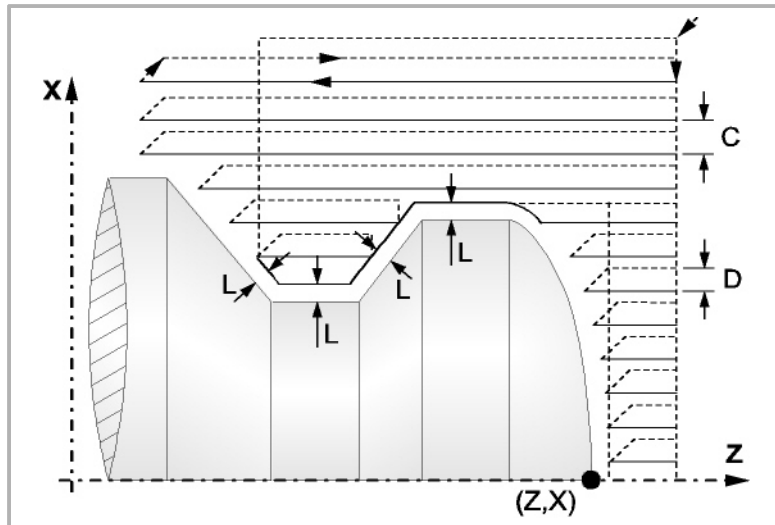


CNC 8060
CNC 8065

(REF: 1502)

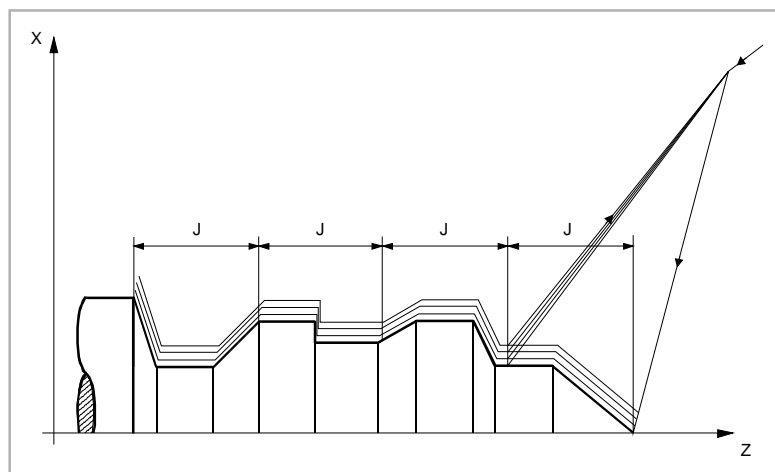
M±5.5 Defines the finishing stock along Z.

If "M" is not programmed, the X and Z residual value will be the one indicated by parameter "L" and the roughing passes will be the same maintaining the "C" distance between two consecutive passes.



J5.5 Optional. Maximum travel in each section on the machining axis. If not programmed, a value of 0 (machining in a single segment) will be assumed. Parameter valid when A0 has been defined and for inside and outside profiles.

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.



K5.5 Defines the penetration feedrate for the roots (valleys). If not programmed or programmed with a value of "0", it assumes the machining feedrate that was programmed before calling the cycle.

F5.5 Defines the feedrate for the final roughing pass. If not programmed or programmed with a value of "0", it means that there is no final roughing pass.

H5.5 Defines the feedrate for the finishing pass. If not programmed or programmed with a value of 0, it is considered that no finishing pass is desired.

1.

MACHINING CANNED CYCLES (ISO)
G68. Stock removal cycle along X axis



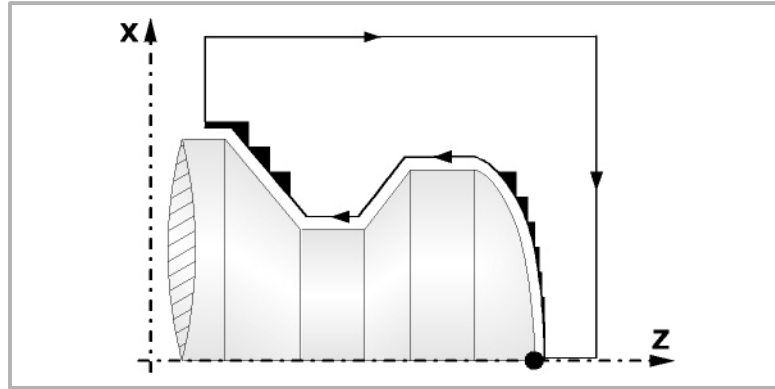
CNC 8060
CNC 8065

(REF: 1502)

1.

MACHINING CANNED CYCLES (ISO)

G68. Stock removal cycle along X axis



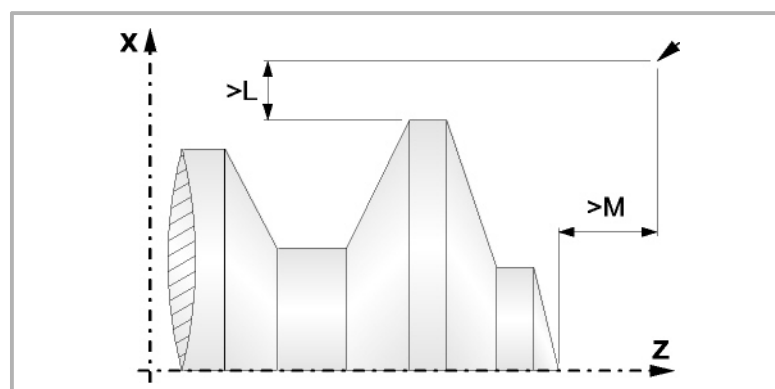
- S4 Label number of the first block describing the geometry of the profile.
- E4 Label number of the last block describing the geometry of the profile.
- The profile may be defined either in the current program or in any other program (parameter "Q").
- P Name of the local subroutine that contains the profile. The local subroutine may be located either in the current program or in any other program (parameter "Q").
- The cycle considers that the whole subroutine makes up the profile; if parameter "P" is programmed, the cycle ignores parameters "E" and "S".
- Q Name of the global subroutine, of the program that contains the definition of the profile (parameters "E" and "S") or of the program that contains the local subroutine that contains the profile (parameter "P").

In short, the profile may be defined in one of the following ways.

Parameters.	Profile location.
E + S	The profile is defined between the "E" and "S" blocks of the current program.
P	The profile is defined in the local subroutine "P" of the current program.
Q	The profile is defined in global subroutine "Q".
Q + E + S	The profile is defined between the "E" and "S" blocks of the program "Q".
Q + P	The profile is defined in the local subroutine "P" of the program "Q".

Machining considerations

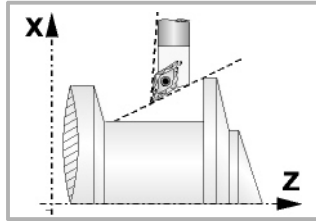
- The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the canned cycle. Once the canned cycle has ended, the active feedrate will be the last one programmed, i.e. the one corresponding to the roughing operation (F) or the finishing operation (H). The CNC will assume functions G00, G40 and G90.
- The canned cycle calling position must be off the part to be machined and at a distance greater than that defined as finishing stock (L, M) along the X and Z axes. If the tool position is not correct to execute the cycle, the CNC will display the corresponding error message.



CNC 8060
CNC 8065

(REF: 1502)

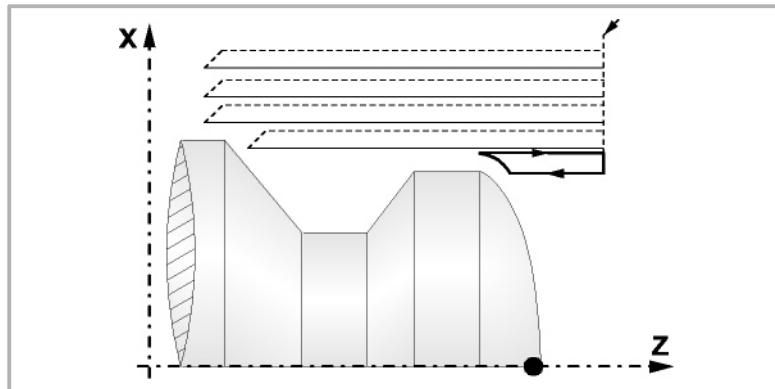
- In those cases when the programmed profile cannot be machined (valleys) with the selected tool, a warning message will be displayed at the beginning of the cycle execution.



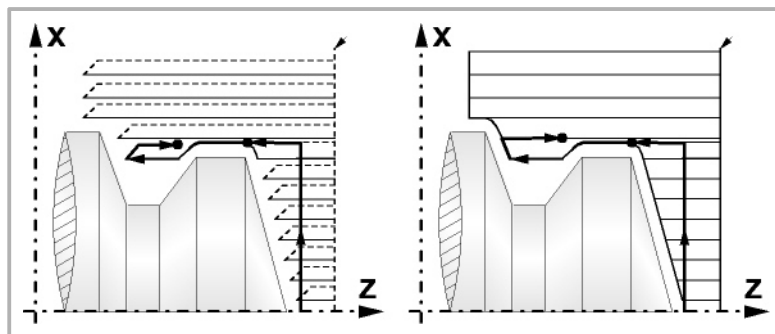
The operator will be able to stop the execution and select the proper tool.

If he/she doesn't do that, a new profile is calculated in the areas that are not accessible for the selected tool and the CNC machines as far as it is possible.

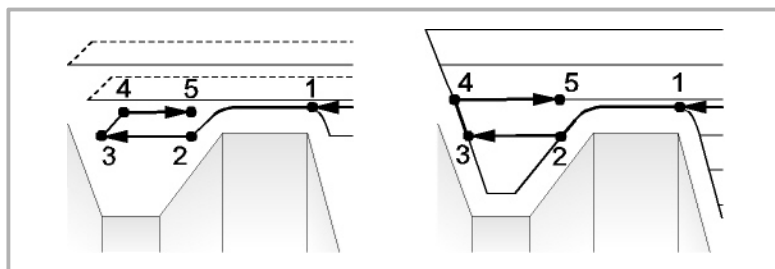
- If when making a roughing pass a channel is detected, the CNC will continue the operation ignoring the channel. The profile may have an unlimited number of channels.



- Once the residual profile is machined, the detected channels will be machined.



To do so, it will return in G00 to the point where the profile machining was interrupted (1). From this point on G01, it will follow the programmed contour maintaining the finishing stock until the selected pass "C" is reached. Section 1-2.



Move "2-3" in G01 at programmed feedrate (F).

If "D" has been programmed, the 3-4 move is done in rapid (G00); but if "D" has not been programmed, the 3-4 move is done following the profile in G01 at the programmed feedrate (F).

The withdrawal "4-5" is done in rapid (G00).

- If when machining a channel, other channels inside it are detected, the same procedure explained above will be followed.

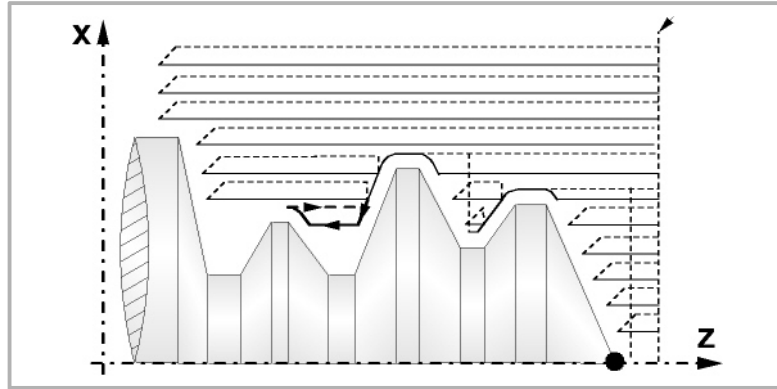
1.

MACHINING CANNED CYCLES (ISO)
G68. Stock removal cycle along X axis

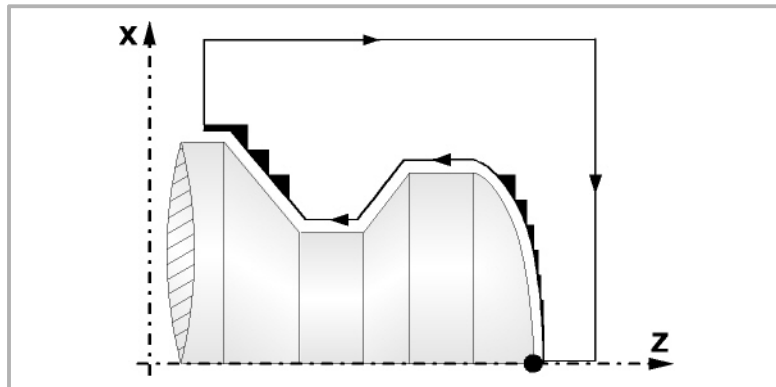
1.

MACHINING CANNED CYCLES (ISO)

G68. Stock removal cycle along X axis

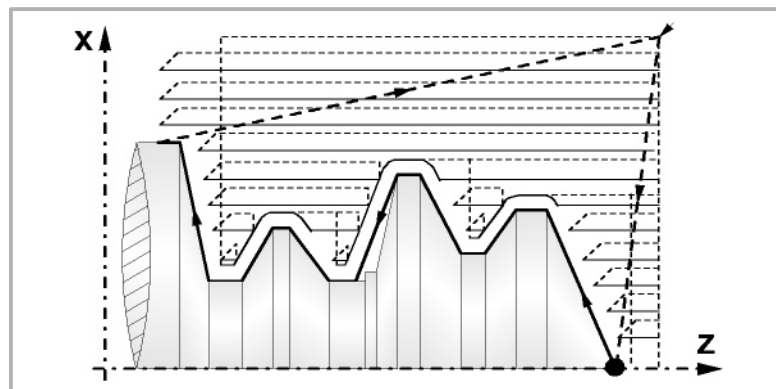


- If a final roughing pass has been programmed, the CNC will run a pass parallel to the profile maintaining an "L" stock at the set feedrate "F". This final roughing pass will eliminate the ridges that were left in the roughing stage.



- If a finishing pass has been defined, it will be carried out with tool radius compensation and at the indicated feedrate "H".

This profile will coincide or not with the programmed profile depending on whether there are areas not accessible to the selected tool.



- Once the finishing pass has ended, the tool will return to the point from where the cycle was called.

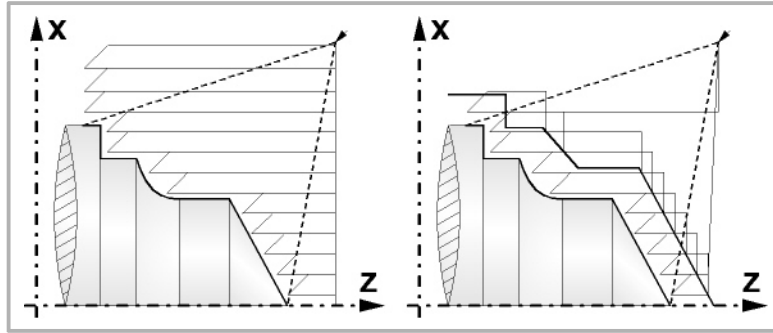


CNC 8060
CNC 8065

(REF: 1502)

Optimizing the machining process

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.

Profile programming syntax

When defining the profile, there is no need to program the initial point since it is already indicated by cycle defining parameters X and Z.

When defining 2 profiles, the final profile must be defined first and then that of the rough part.

The first and last profile defining blocks must have a label number. These labels will indicate the beginning and end of the profile geometry definition for this canned cycle.

The profile programming syntax must follow these rules:

- It may be programmed in absolute or incremental coordinates and it may consist of simple geometrical elements such as straight and curved lines, roundings and chamfers which will be programmed following their own syntax rules.
- Function G00 indicates that the profile definition is done and this block is the beginning of the definition of the rough part profile.
- Program G01, G02 or G03 in the next block, because G00 is modal, thus preventing the CNC from displaying the corresponding error message.
- The profile description must not contain: mirror images, scaling factor changes, pattern rotation or zero offsets.
- It must not contain high level blocks such as jumps, calls to subroutines or parametric programming.
- It must not contain other canned cycles.

Only the following G functions may be programmed in the profile definition:

G01	Linear interpolation
G02	Clockwise circular interpolation
G03	Counter-clockwise circular interpolation
G06	Circle center in absolute coordinates
G08	Arc tangent to previous path
G09	Arc defined by three points
G36	Automatic radius blend
G39	Chamfer
G53	Programming with respect to machine zero
G70	Programming in inches
G71	Programming in millimeters
G90	Absolute programming:
G91	Incremental programming
G30	Polar origin preset

1.

MACHINING CANNED CYCLES (ISO)
G68. Stock removal cycle along X axis

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

It is also possible to program the following functions although they will be ignored by the canned cycle:

G05 Round corner

G07 Square corner

G50 Controlled corner rounding

The F, S, T, D and M functions

1.

MACHINING CANNED CYCLES (ISO)

G68. Stock removal cycle along X axis



CNC 8060
CNC 8065

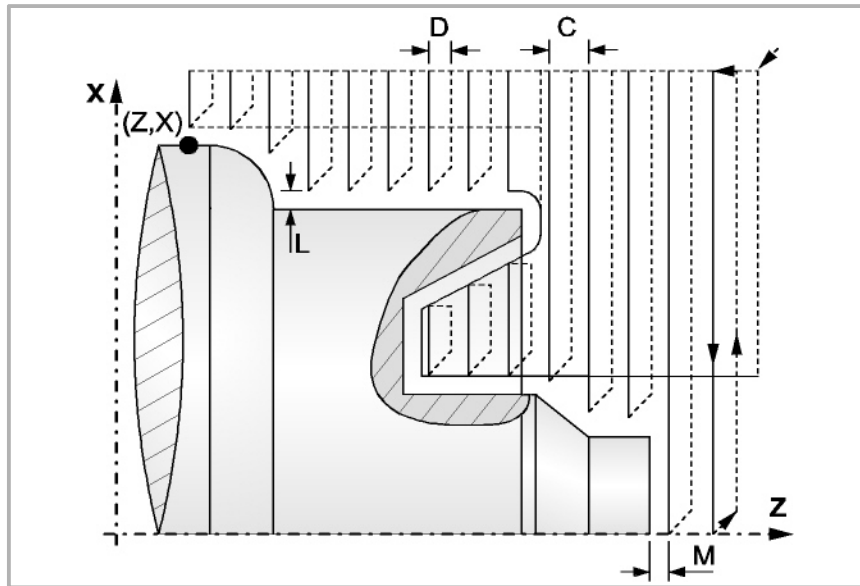
(REF: 1502)

1.13 G69. Stock removal canned cycle along Z axis

Pattern repeat canned cycle, this cycle machines the programmed profile maintaining the specified pass between the successive machining passes. It admits triangular, round and square tools.

Programming format in Cartesian coordinates:

G69 X Z C D L M K F H S E P



X±5.5 Defines the X coordinate of the initial point of the profile. It must be programmed in absolute values and in the active units (radius or diameter).

Z±5.5 Defines the Z coordinate of the initial point of the profile. It must be programmed in absolute values.

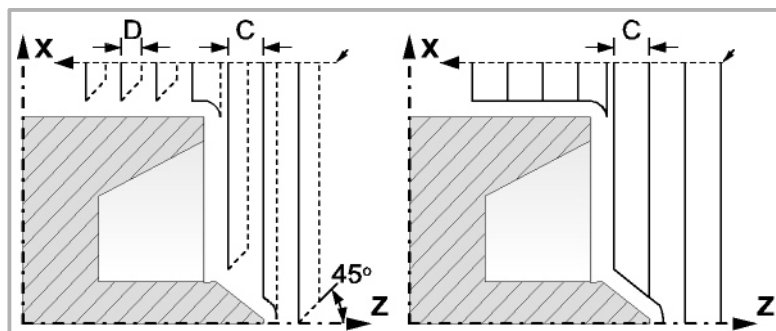
C5.5 Defines the machining pass. If programmed with a 0 value, the CNC will display the corresponding error message.

All the machining passes are performed with this value, except the last one which will eliminate the remaining stock.

- D 5.5 Defines the safety distance to withdraw the tool after each pass.
- When programming "D" with a value other than "0", the cutter withdraws at a 45° angle up to the safety distance (left drawing).
 - When programming "D" with a "0" value, the exit path is the same as the entry path. This may be interesting for grooving complex profiles, to use these cycles on cylindrical grinders, etc.

When not programming "D", the tool withdrawal is carried out following the profile up to the last pass, "C" distance (right drawing).

It must be borne in mind that when "D" is not programmed, the cycle execution time is greater; but the amount of stock to be removed on the finishing pass is less.



L±5.5 Defines the finishing stock along X. It is defined in radius and if not programmed, a value of 0 is assumed.

1.

MACHINING CANNED CYCLES (ISO)
G69. Stock removal canned cycle along Z axis

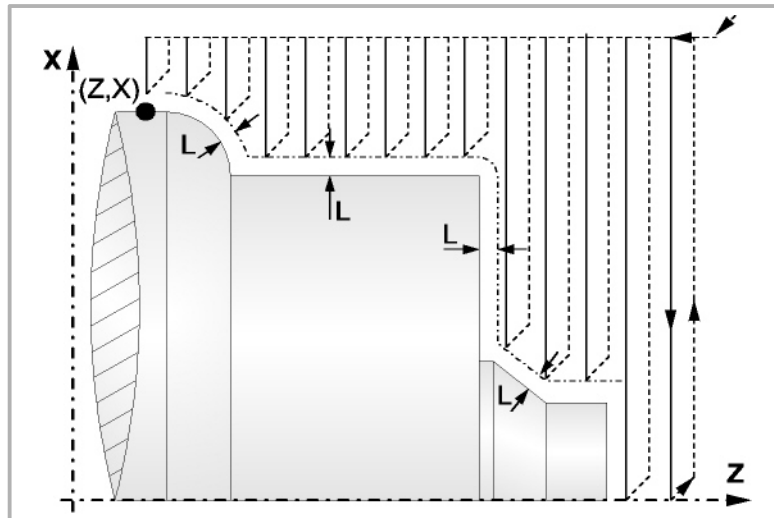


CNC 8060
CNC 8065

(REF: 1502)

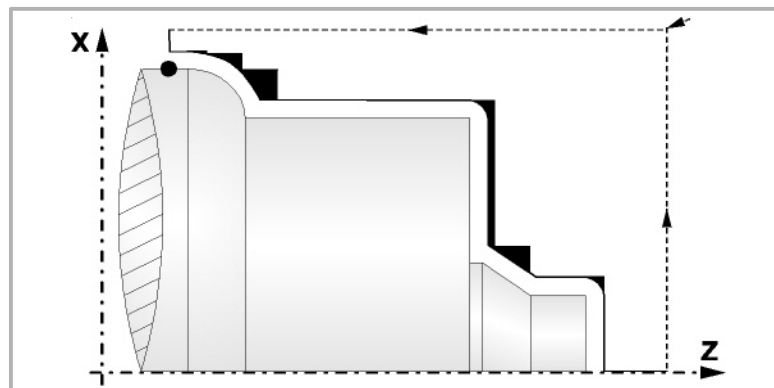
M±5.5 Defines the finishing stock along Z.

If "M" is not programmed, the X and Z residual value will be the one indicated by parameter "L" and the roughing passes will be the same maintaining the "C" distance between two consecutive passes.



K5.5 Defines the penetration feedrate for the roots (valleys). If not programmed or programmed with a value of "0", it assumes the machining feedrate that was programmed before calling the cycle.

F5.5 Defines the feedrate for the final roughing pass. If not programmed or programmed with a value of "0", it means that there is no final roughing pass.



H5.5 Defines the feedrate for the finishing pass. If not programmed or programmed with a value of 0, it is considered that no finishing pass is desired.

S4 Label number of the first block describing the geometry of the profile.

E4 Label number of the last block describing the geometry of the profile.

The profile may be defined either in the current program or in any other program (parameter "Q").

P Name of the local subroutine that contains the profile. The local subroutine may be located either in the current program or in any other program (parameter "Q").

The cycle considers that the whole subroutine makes up the profile; if parameter "P" is programmed, the cycle ignores parameters "E" and "S".

Q Name of the global subroutine, of the program that contains the definition of the profile (parameters "E" and "S") or of the program that contains the local subroutine that contains the profile (parameter "P").

1.

MACHINING CANNED CYCLES (ISO)

G69. Stock removal canned cycle along Z axis



CNC 8060
CNC 8065

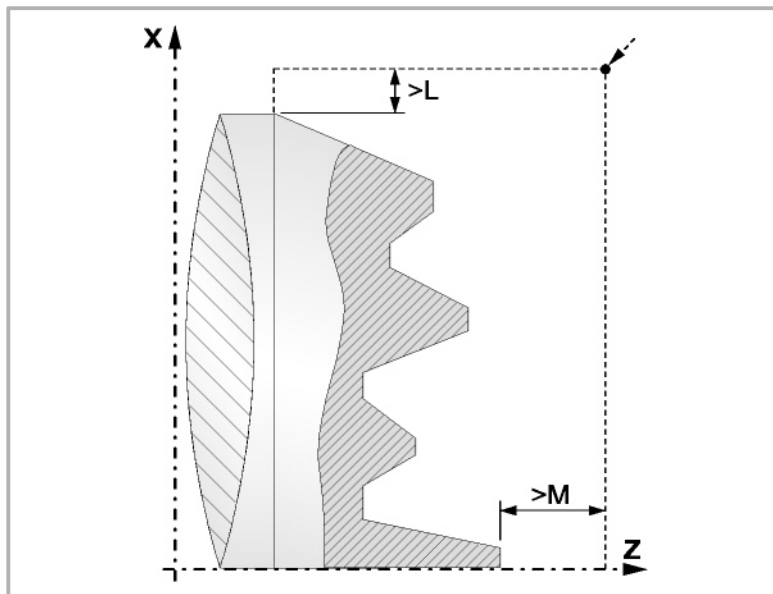
(REF: 1502)

In short, the profile may be defined in one of the following ways.

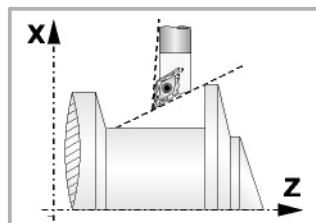
Parameters.	Profile location.
E + S	The profile is defined between the "E" and "S" blocks of the current program.
P	The profile is defined in the local subroutine "P" of the current program.
Q	The profile is defined in global subroutine "Q".
Q + E + S	The profile is defined between the "E" and "S" blocks of the program "Q".
Q + P	The profile is defined in the local subroutine "P" of the program "Q".

Machining considerations

- The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the canned cycle. Once the canned cycle has ended, the active feedrate will be the last one programmed, i.e. the one corresponding to the roughing operation (F) or the finishing operation (H). The CNC will assume functions G00, G40 and G90.
- The canned cycle calling position must be off the part to be machined and at a distance greater than that defined as finishing stock (L, M) along the X and Z axes. If the tool position is not correct to execute the cycle, the CNC will display the corresponding error message.



- In those cases when the programmed profile cannot be machined (valleys) with the selected tool, a warning message will be displayed at the beginning of the cycle execution.



The operator will be able to stop the execution and select the proper tool.

If he/she doesn't do that, a new profile is calculated in the areas that are not accessible for the selected tool and the CNC machines as far as it is possible.

- If when making a roughing pass a channel is detected, the CNC will continue the operation ignoring the channel. The profile may have an unlimited number of channels.

1.

MACHINING CANNED CYCLES (ISO)
G69. Stock removal canned cycle along Z axis



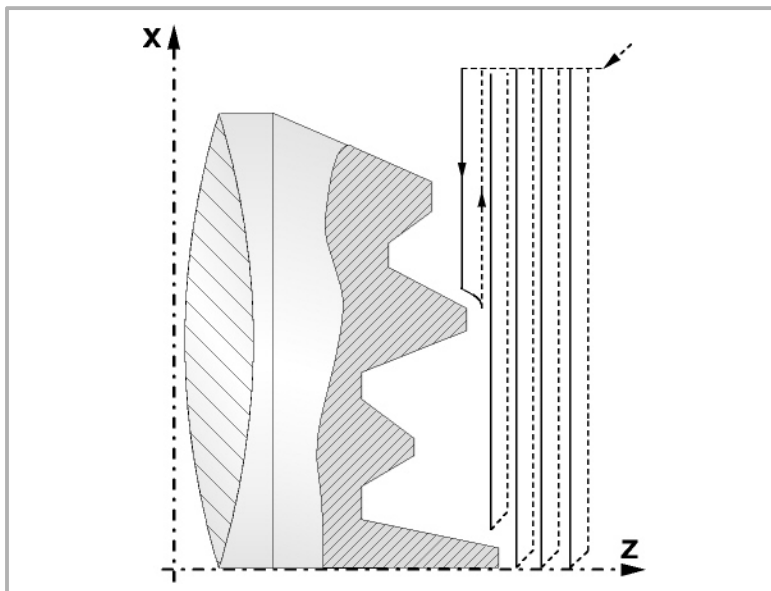
CNC 8060
CNC 8065

(REF: 1502)

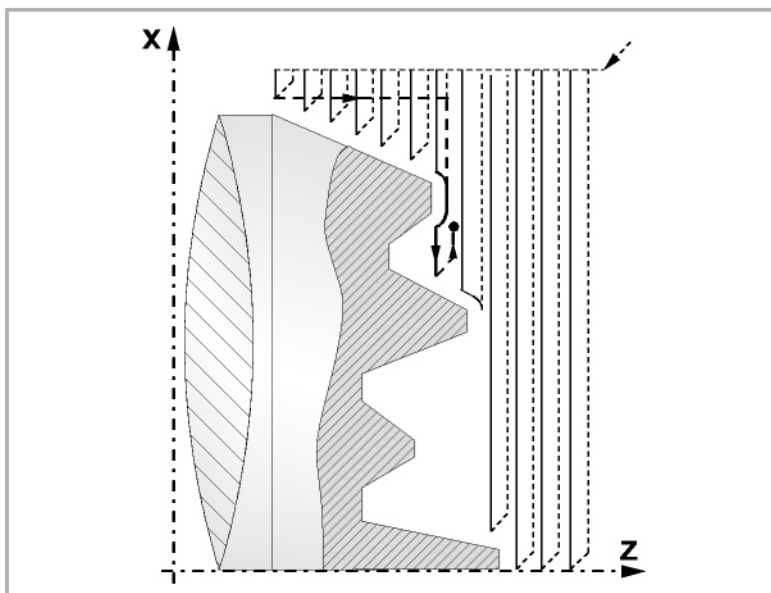
1.

MACHINING CANNED CYCLES (ISO)

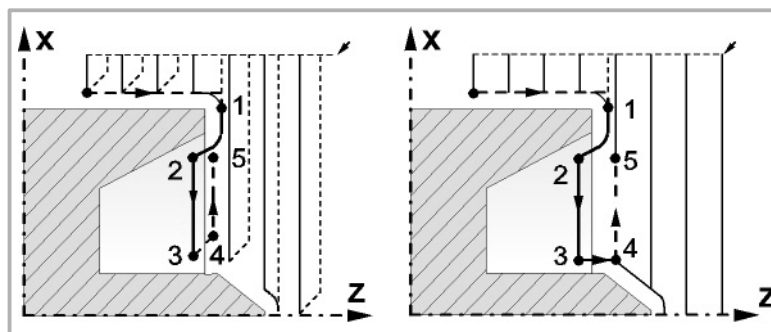
G69. Stock removal canned cycle along Z axis



- Once the residual profile is machined, the detected channels will be machined.



To do so, it will return in G00 to the point where the profile machining was interrupted (1). From this point on G01, it will follow the programmed contour maintaining the finishing stock until the selected pass "C" is reached. Section 1-2.



Move "2-3" in G01 at programmed feedrate (F).

If "D" has been programmed, the 3-4 move is done in rapid (G00); but if "D" has not been programmed, the 3-4 move is done following the profile in G01 at the programmed feedrate (F).

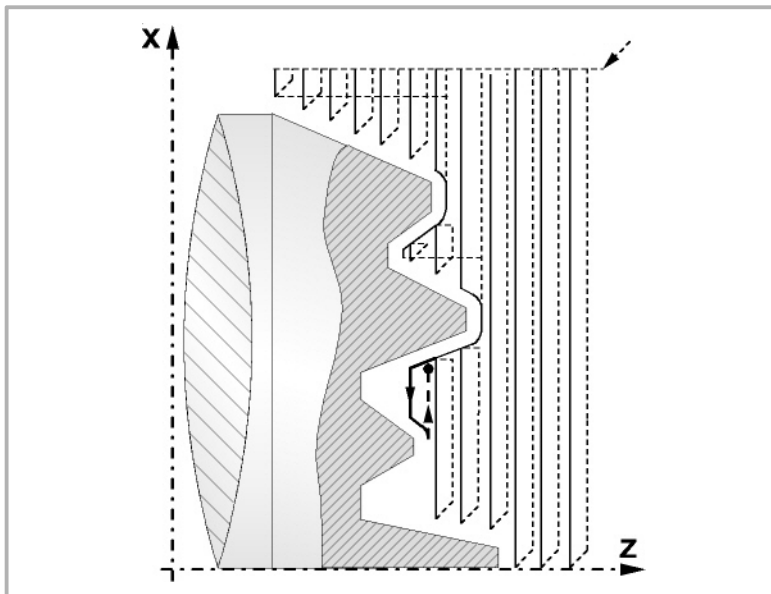
The withdrawal "4-5" is done in rapid (G00).



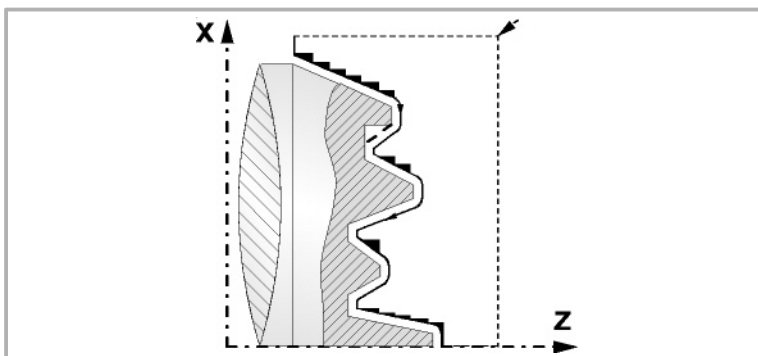
CNC 8060
CNC 8065

(REF: 1502)

- If when machining a channel, other channels inside it are detected, the same procedure explained above will be followed.

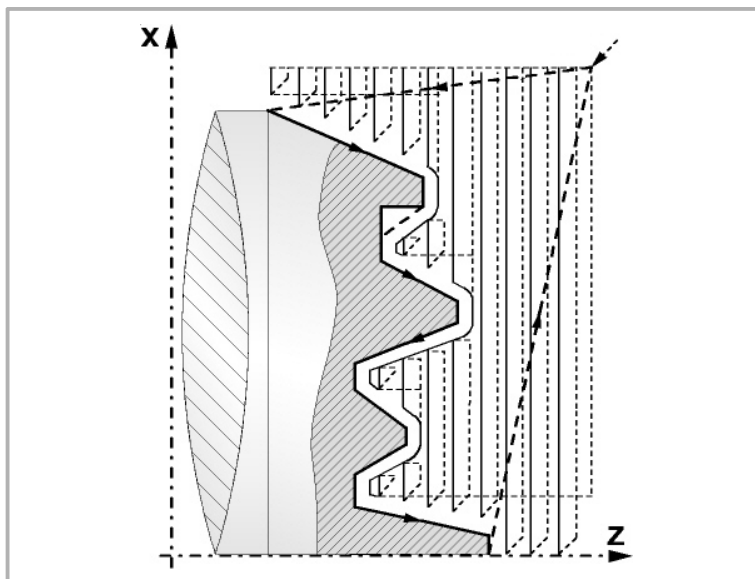


- If a final roughing pass has been programmed, the CNC will run a pass parallel to the profile maintaining an "L" stock at the set feedrate "F". This final roughing pass will eliminate the ridges that were left in the roughing stage.



- If a finishing pass has been defined, it will be carried out with tool radius compensation and at the indicated feedrate "H".

This profile will coincide or not with the programmed profile depending on whether there are areas not accessible to the selected tool.



- Once the finishing pass has ended, the tool will return to the point from where the cycle was called.

1.

MACHINING CANNED CYCLES (ISO)

G69. Stock removal canned cycle along Z axis

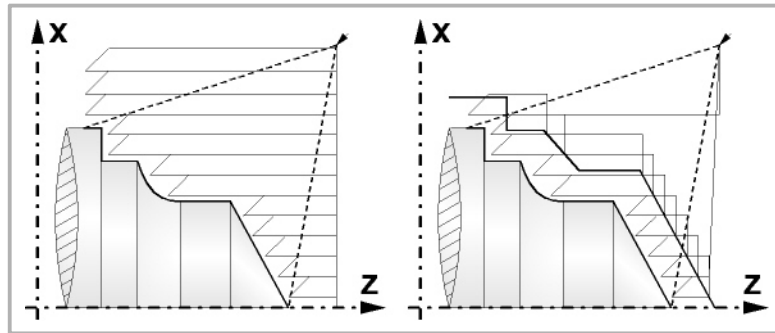


CNC 8060
CNC 8065

(REF: 1502)

Optimizing the machining process

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.

Profile programming syntax

When defining the profile, there is no need to program the initial point since it is already indicated by cycle defining parameters X and Z.

When defining 2 profiles, the final profile must be defined first and then that of the rough part.

The first and last profile defining blocks must have a label number. These labels will indicate the beginning and end of the profile geometry definition for this canned cycle.

The profile programming syntax must follow these rules:

- It may be programmed in absolute or incremental coordinates and it may consist of simple geometrical elements such as straight and curved lines, roundings and chamfers which will be programmed following their own syntax rules.
- Function G00 indicates that the profile definition is done and this block is the beginning of the definition of the rough part profile.
- Program G01, G02 or G03 in the next block, because G00 is modal, thus preventing the CNC from displaying the corresponding error message.
- The profile description must not contain: mirror images, scaling factor changes, pattern rotation or zero offsets.
- It must not contain high level blocks such as jumps, calls to subroutines or parametric programming.
- It must not contain other canned cycles.

Only the following G functions may be programmed in the profile definition:

G01	Linear interpolation
G02	Clockwise circular interpolation
G03	Counter-clockwise circular interpolation
G06	Circle center in absolute coordinates
G08	Arc tangent to previous path
G09	Arc defined by three points
G36	Automatic radius blend
G39	Chamfer
G53	Programming with respect to machine zero
G70	Programming in inches
G71	Programming in millimeters
G90	Absolute programming:
G91	Incremental programming
G30	Polar origin preset

1.

MACHINING CANNED CYCLES (ISO)

G69. Stock removal canned cycle along Z axis

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

Canned cycles (.T. model)

It is also possible to program the following functions although they will be ignored by the canned cycle:

G05 Round corner

G07 Square corner

G50 Controlled corner rounding

The F, S, T, D and M functions

1.

MACHINING CANNED CYCLES (ISO)

G69. Stock removal canned cycle along Z axis

FAGOR 

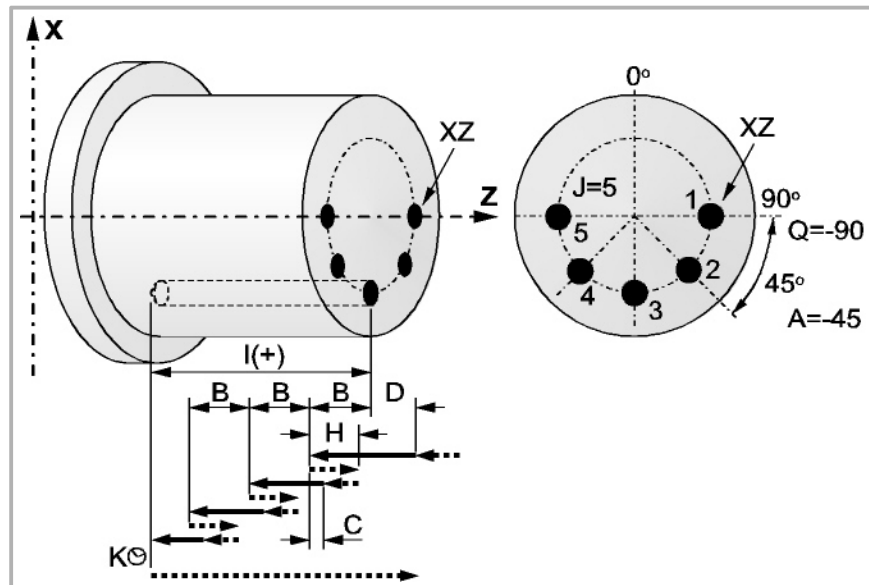
CNC 8060
CNC 8065

(REF: 1502)

1.14 G160. Drilling / tapping on the face of the part

Executing this cycle requires using a live tool. While drilling or tapping, the spindle is stopped and the tool is turning. This makes it possible to machine anywhere on the workpiece.

Whether one operation or the other will be performed depends on the format being used. Set parameter "B=0" for tapping and "B>0" for drilling.



Programming format in Cartesian coordinates:

Drilling `G160 X Z I B Q A J D K H C S R N`

Tapping `G160 X Z I B0 Q A J D S R N`

- X±5.5** Defines the X coordinate where the cycle is to be run. It must be programmed in absolute values and in the active units (radius or diameter).
- Z±5.5** Defines the Z coordinate where the cycle will be run. It must be programmed in absolute values.
- I±5.5** It defines the machining depth and it is referred to the starting point (X, Z). The cycle chooses the machining direction depending on the starting position of the tool and the starting position of the machining operation. If they are both the same, the parameter will have a positive value when drilling or threading in the negative Z direction and negative when drilling or threading in the opposite direction.
- If programmed with a 0 value, the CNC will display the corresponding error message.
- B5.5** Defines the type of operation to be run.
- Set B=0 for tapping.
- B>0 for drilling, where B indicates the drilling step.
- Q±5.5** Defines the first angular position to orient the spindle to drill or tap the first hole.
- A±5.5** Defines the angular increment between two consecutive operations. It is programmed in degrees. Positive when counter-clockwise.
- J4** Defines the number of holes to be drilled or tapped, including the first one.
- If programmed with a 0 value, the CNC will display the corresponding error message.
- D5.5** Defines the safety distance along the Z axis and indicates how far from the starting point (Z, X) the tool will approach the part. If not programmed, a value of 0 is assumed.

1.

MACHINING CANNED CYCLES (ISO)

G160. Drilling / tapping on the face of the part



CNC 8060
CNC 8065

(REF: 1502)

- K5** Defines the dwell, in hundredths of a second, at the bottom of the hole before starting the withdrawal. If not programmed, a value of 0 is assumed.
- The tapping operation ignores this parameter, thus not being necessary to program it. If programmed, it will be ignored.
- H5.5** Defines the rapid (in G00) withdrawing distance along the Z axis after each drilling peck. If not programmed or programmed a value of 0, it will withdraw to the approach point.
- The tapping operation ignores this parameter, thus not being necessary to program it. If programmed, it will be ignored.
- C5.5** Defines the rapid approaching distance for successive drilling pecks. If not programmed, a value of 1 millimeter.
- The tapping operation ignores this parameter, thus not being necessary to program it. If programmed, it will be ignored.
- S±5.5** RPM and turning direction (sign) of the live tool.
- If not programmed, the live tool does not start up
- R5.5** In the drilling cycle, it indicates the step that reduces the drilling peck "B". If it is not programmed or is programmed with a value of "0", it assumes a value of "1".
- With R =1, all the drilling pecks will be the same (the programmed "B" value). If R is other than 1, the first drilling peck will be "B", the second one "R B", The third one "R (RB)" and so on. In other words, from the second peck on, the new peck will be the product of the R factor times the previous peck.
- In the tapping cycle, it defines the type of tapping. "R0" means tapping with a clutch and "R1" means rigid tapping. If not programmed, it takes the value of "0", tapping with a clutch.
- In order to execute a rigid tapping, the spindle must be ready to operate in closed loop; in other words, that it must have a servo drive-motor system with rotary encoder.
- N** Number of the spindle for the live tool. N1 for spindle S1, N2 for spindle S2 and so on.



MACHINING CANNED CYCLES (ISO)
G160. Drilling / tapping on the face of the part



CNC 8060
CNC 8065

(REF: 1502)

1.

MACHINING CANNED CYCLES (ISO)

G160. Drilling / tapping on the face of the part

Drilling. Basic operation

- 1 Rapid move to the approach point located at a "D" distance from the first hole position.
- 2 The CNC turns on the live tool at the rpm and in the turning direction indicated by parameter "S".
- 3 Orients the spindle at the angular position set by "Q". If the spindle was turning, it will stop.
- 4 First drilling operation. The drilling axis moves in G01 to the programmed incremental depth "D+B".
- 5 Drilling loop. The following steps will be repeated until the programmed depth "I" is reached.

Withdrawal in rapid (G00) a distance "H" or to the approach point. Rapid approach (G00) to a "C" distance from the next peck. New drilling peck at work feedrate (G01) to the next incremental according to B and R.

- 6 Dwell "K" at the bottom of the hole in hundredths of a second, if programmed.
- 7 Withdrawal in rapid (G00) to the approach point.
- 8 Depending on the value assigned to parameter "J" (number of holes), the spindle moves to the new position (angular increment "A") and repeats the movements indicated in steps 4, 5, 6 and 7.
- 9 The live tool stops.

Tapping with a clutch. Basic operation

- 1 Rapid move to the approach point located at a "D" distance from the first hole position.
- 2 The CNC turns on the live tool at the rpm and in the turning direction indicated by parameter "S".
- 3 Orients the spindle at the angular position set by "Q". If the spindle was turning, it will stop.
- 4 Tapping. Movement of the tapping axis to the bottom of the hole "I". MFO, SSO, FEED-HOLD and CYCLE STOP are disabled.
- 5 Reversal of the turning direction of the live tool.
- 6 Withdrawal in work feedrate to the approach point.
- 7 Depending on the value assigned to parameter "J" (number of taps), the spindle moves to the new position (angular increment "A") and repeats the movements indicated in steps 4, 5 and 6.
- 8 The live tool stops.

Rigid tapping. Basic operation

- 1 Rapid movement to the approach point located at a safety distance "D" from the tap point.
- 2 Orients the spindle at the angular position set by "Q". If the spindle was turning, it will stop.
- 3 Tapping. It is done by interpolating the live tool with the Z axis.
The feedrate "F" must be programmed before the cycle and the spindle speed "S" is implicit in the definition of the cycle. The cycle assumes functions G94 and G97. Rigid tapping cannot be interrupted, nor the machining conditions be changed. It is done at 100% of the programmed "S" and "F".
- 4 Reversal of the turning direction of the live tool.
- 5 Withdrawal in work feedrate to the approach point.
- 6 Depending on the value assigned to parameter "J" (number of taps), the spindle moves to the new position (angular increment "A") and repeats the movements indicated in steps 4, 5 and 6.
- 7 The live tool stops.

When the cycle is completed, the second spindle stops (M5).



CNC 8060
CNC 8065

(REF: 1502)

Machining considerations

- The machining conditions (feedrate, live tool rpm, etc.) must be programmed prior to the calling the cycle. Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle. Only the tool radius compensation will be canceled (G40) if it was active.
- If when executing the cycle, it is working in G95 and it has not been working in G94 before, the CNC shows "error 1039. The F has not been programmed in G94".
- When tapping (regular or rigid), the general logic output "TAPPING" stays active during the execution of the cycle.

1.

MACHINING CANNED CYCLES (ISO)

G160. Drilling / tapping on the face of the part

FAGOR 

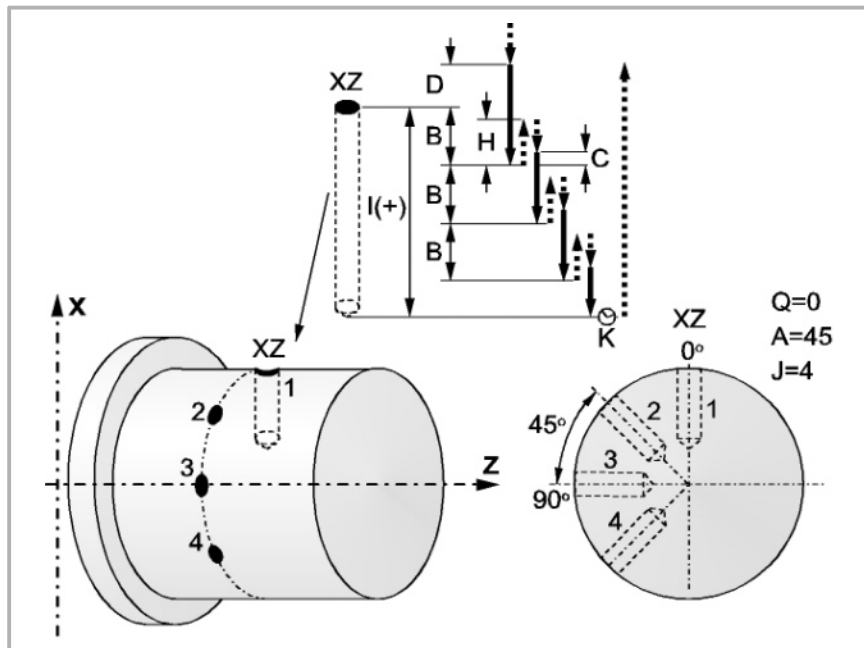
**CNC 8060
CNC 8065**

(REF: 1502)

1.15 G161. Drilling / tapping on the side of the part

Executing this cycle requires using a live tool. While drilling or tapping, the spindle is stopped and the tool is turning. This makes it possible to machine anywhere on the workpiece.

Whether one operation or the other will be performed depends on the format being used. Set parameter "B=0" for tapping and "B>0" for drilling.



Programming format in Cartesian coordinates:

Drilling `G161 X Z I B Q A J D K H C S R N`

Tapping `G161 X Z I B0 Q A J D S R N`

- X±5.5** Defines the X coordinate where the cycle is to be run. It must be programmed in absolute values and in the active units (radius or diameter).
- Z±5.5** Defines the Z coordinate where the cycle will be run. It must be programmed in absolute values.
- I±5.5** It defines the machining depth and it is referred to the starting point (X, Z). The cycle chooses the machining direction depending on the starting position of the tool and the starting position of the machining operation. If they are both the same, the parameter will have a positive value when drilling or threading in the negative X direction and negative when drilling or threading in the opposite direction.
- If programmed with a 0 value, the CNC will display the corresponding error message.
- B5.5** Defines the type of operation to be run.
- Set B=0 for tapping.
- B>0 for drilling, where B indicates the drilling step in radius.
- Q±5.5** Defines the first angular position to orient the spindle to drill or tap the first hole.
- A±5.5** Defines the angular increment between two consecutive operations. It is programmed in degrees. Positive when counter-clockwise.
- J4** Defines the number of holes to be drilled or tapped, including the first one.
- If programmed with a 0 value, the CNC will display the corresponding error message.
- D5.5** Defines (in radius) the safety distance along the X axis and indicates how far from the starting point (Z, X) the tool will approach the part. If not programmed, a value of 0 is assumed.

1.

MACHINING CANNED CYCLES (ISO)

G161. Drilling / tapping on the side of the part



CNC 8060
CNC 8065

(REF: 1502)

- K5** Defines the dwell, in hundredths of a second, at the bottom of the hole before starting the withdrawal. If not programmed, a value of 0 is assumed.
- The tapping operation ignores this parameter, thus not being necessary to program it. If programmed, it will be ignored.
- H5.5** Defines (in radius) the rapid (in G00) withdrawing distance along the X axis after each drilling peck. If not programmed or programmed a value of 0, it will withdraw to the approach point.
- The tapping operation ignores this parameter, thus not being necessary to program it. If programmed, it will be ignored.
- C5.5** Defines (in radius) the rapid approaching distance for successive drilling pecks. If not programmed, a value of 1 millimeter.
- The tapping operation ignores this parameter, thus not being necessary to program it. If programmed, it will be ignored.
- S±5.5** RPM and turning direction (sign) of the live tool.
- If not programmed, the live tool does not start up
- R5.5** In the drilling cycle, it indicates the step that reduces the drilling peck "B". If it is not programmed or is programmed with a value of "0", it assumes a value of "1".
- With R =1, all the drilling pecks will be the same (the programmed "B" value). If R is other than 1, the first drilling peck will be "B", the second one "R B", The third one "R (RB)" and so on. In other words, from the second peck on, the new peck will be the product of the R factor times the previous peck.
- In the tapping cycle, it defines the type of tapping. "R0" means tapping with a clutch and "R1" means rigid tapping. If not programmed, it takes the value of "0", tapping with a clutch.
- In order to execute a rigid tapping, the spindle must be ready to operate in closed loop; in other words, that it must have a servo drive-motor system with rotary encoder.
- N** Number of the spindle for the live tool. N1 for spindle S1, N2 for spindle S2 and so on.



MACHINING CANNED CYCLES (ISO)
G161. Drilling / tapping on the side of the part



CNC 8060
CNC 8065

(REF: 1502)

1.

MACHINING CANNED CYCLES (ISO)

G161. Drilling / tapping on the side of the part

Drilling. Basic operation

- 1 Rapid move to the approach point located at a "D" distance from the first hole position.
- 2 The CNC turns on the live tool at the rpm and in the turning direction indicated by parameter "S".
- 3 Orients the spindle at the angular position set by "Q". If the spindle was turning, it will stop.
- 4 First drilling operation. The drilling axis moves in G01 to the programmed incremental depth "D+B".
- 5 Drilling loop. The following steps will be repeated until the programmed depth "I" is reached.

Withdrawal in rapid (G00) a distance "H" or to the approach point. Rapid approach (G00) to a "C" distance from the next peck. New drilling peck at work feedrate (G01) to the next incremental according to B and R.

- 6 Dwell "K" at the bottom of the hole in hundredths of a second, if programmed.
- 7 Withdrawal in rapid (G00) to the approach point.
- 8 Depending on the value assigned to parameter "J" (number of holes), the spindle moves to the new position (angular increment "A") and repeats the movements indicated in steps 4, 5, 6 and 7.
- 9 The live tool stops.

Tapping with a clutch. Basic operation

- 1 Rapid move to the approach point located at a "D" distance from the first hole position.
- 2 The CNC turns on the live tool at the rpm and in the turning direction indicated by parameter "S".
- 3 Orients the spindle at the angular position set by "Q". If the spindle was turning, it will stop.
- 4 Tapping. Movement of the tapping axis to the bottom of the hole "I".
- 5 Reversal of the turning direction of the live tool.
- 6 Withdrawal in work feedrate to the approach point.
- 7 Depending on the value assigned to parameter "J" (number of taps), the spindle moves to the new position (angular increment "A") and repeats the movements indicated in steps 4, 5 and 6.
- 8 The live tool stops.

Rigid tapping. Basic operation

- 1 Rapid movement to the approach point located at a safety distance "D" from the tap point.
- 2 Orients the spindle at the angular position set by "Q". If the spindle was turning, it will stop.
- 3 Tapping. It is done by interpolating the live tool with the X axis.

The feedrate "F" must be programmed before the cycle and the spindle speed "S" is implicit in the definition of the cycle. The cycle assumes functions G94 and G97.

Rigid tapping cannot be interrupted, nor the machining conditions be changed. It is done at 100% of the programmed "S" and "F".

- 4 Reversal of the turning direction of the live tool.
- 5 Withdrawal in work feedrate to the approach point.
- 6 Depending on the value assigned to parameter "J" (number of taps), the spindle moves to the new position (angular increment "A") and repeats the movements indicated in steps 4, 5 and 6.
- 7 The live tool stops.

When the cycle is completed, the second spindle stops (M5).



CNC 8060
CNC 8065

(REF: 1502)

Machining considerations

- The machining conditions (feedrate, live tool rpm, etc.) must be programmed prior to the calling the cycle. Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle. Only the tool radius compensation will be canceled (G40) if it was active.
- If when executing the cycle, it is working in G95 and it has not been working in G94 before, the CNC shows "error 1039. The F has not been programmed in G94".
- When tapping (regular or rigid), the general logic output "TAPPING" (M5517) stays active during the execution of the cycle.

1.

MACHINING CANNED CYCLES (ISO)

G161. Drilling / tapping on the side of the part

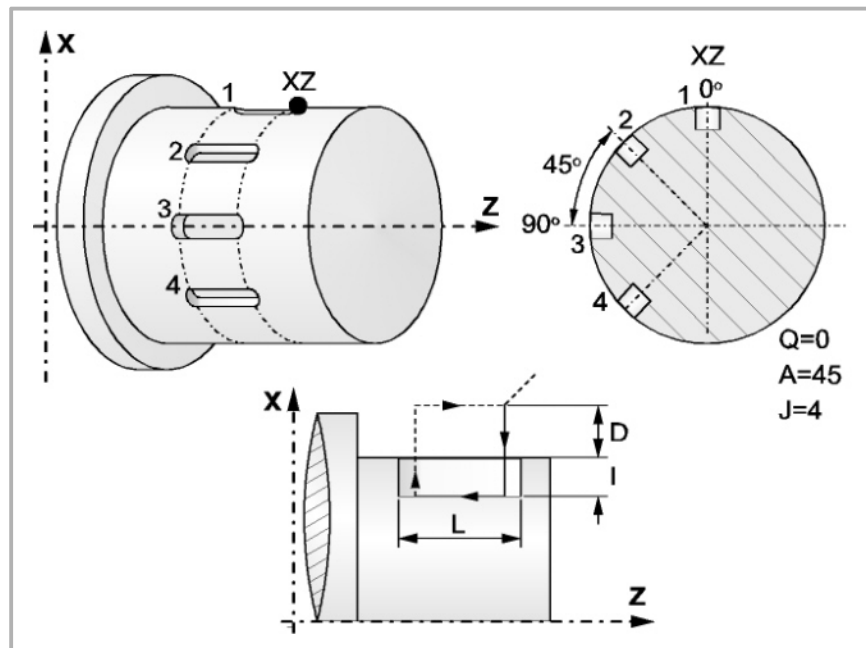
FAGOR 

**CNC 8060
CNC 8065**

(REF: 1502)

1.16 G162. Slot milling canned cycle along the side of the part

Executing this cycle requires using a live tool. While performing this operation, the spindle is stopped and the tool is turning. This makes it possible to machine anywhere on the workpiece.



Programming format in Cartesian coordinates:

G162 X Z L I Q A J D F S N

- X±5.5** Defines the X coordinate where the cycle is to be run. It must be programmed in absolute values and in the active units (radius or diameter).
- Z±5.5** Defines the Z coordinate where the cycle will be run. It must be programmed in absolute values.
- L±5.5** Defines the length of the slot. It is referred to the starting point (X, Z) and it will, therefore, have a positive value when machining in the negative Z direction and negative when doing it in the opposite direction. "L(+)" in the example above.
If programmed with a 0 value, the CNC will display the corresponding error message.
- I±5.5** Defines, in radius, the depth of the slot. It is referred to the starting point (X, Z).
If programmed with a 0 value, the CNC will display the corresponding error message.
- Q±5.5** Defines the angular position (in degrees) to orient the spindle to mill the first slot.
- A±5.5** Defines the angular increment between two consecutive operations. It is programmed in degrees. Positive when counter-clockwise.
- J 4** Defines the number of slots to be milled. If programmed with a 0 value, the CNC will display the corresponding error message.
- D5.5** Defines (in radius) the safety distance along the X axis and indicates how far from the starting point (Z, X) the tool will approach the part. If not programmed, a value of 0 is assumed.
- F5.5** Defines the slot milling feedrate.
- S±5.5** RPM and turning direction (sign) of the live tool.
If not programmed, the live tool does not start up
- N** Number of the spindle for the live tool. N1 for spindle S1, N2 for spindle S2 and so on.

1.

MACHINING CANNED CYCLES (ISO)

G162. Slot milling canned cycle along the side of the part

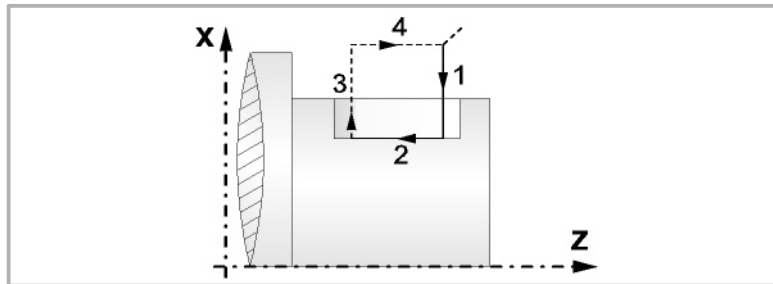


CNC 8060
CNC 8065

(REF: 1502)

Basic operation

- 1 Rapid approach up to the safety distance "D" from the slot milling point.
- 2 The CNC turns the live tool on at the rpm and turning direction set by parameter "S".
- 3 Orients the spindle at the angular position set by "Q". If the spindle was turning, it will stop.
- 4 Milling of the slot by following these steps:



In feed at the feedrate selected prior to calling the cycle.

Milling of the slot by moving the Z axis at the programmed "F" feedrate.

Rapid withdrawal to the reference point.

Rapid return to the starting point.

- 5 Depending on the value assigned to parameter "J" (number of slots), the spindle moves to the new position (angular increment "A") and repeats the movements indicated in step 4.
- 6 The live tool stops.

Machining considerations

- The machining conditions (feedrate, live tool rpm, etc.) must be programmed prior to the calling the cycle. Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle. Only the tool radius compensation will be canceled (G40) if it was active.
- If when executing the cycle, it is working in G95 and it has not been working in G94 before, the CNC shows "error 1039. The F has not been programmed in G94".

1.

MACHINING CANNED CYCLES (ISO)

G162. Slot milling canned cycle along the side of the part

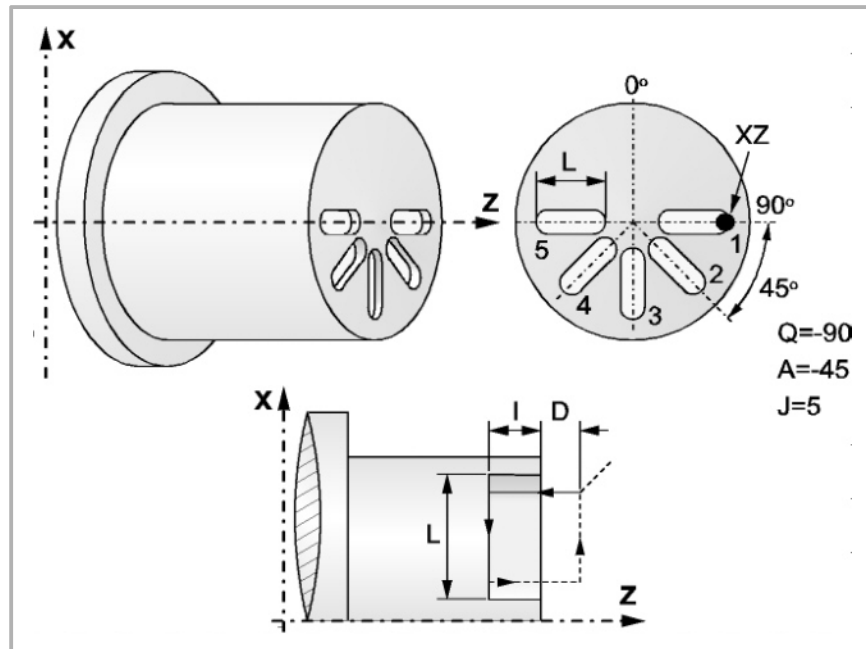
FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

1.17 G163. Radial slot milling cycle (on the face of the part)

Executing this cycle requires using a live tool. While performing this operation, the spindle is stopped and the tool is turning. This makes it possible to machine anywhere on the workpiece.



Programming format in Cartesian coordinates:

G163 X Z L I Q A J D F S N

- $X_{\pm 5.5}$ Defines the X coordinate where the cycle is to be run. It must be programmed in absolute values and in the active units (radius or diameter).
- $Z_{\pm 5.5}$ Defines the Z coordinate where the cycle will be run. It must be programmed in absolute values.
- $L_{\pm 5.5}$ Defines, in radius, the length of the slot. It is referred to the starting point (X, Z) and it will, therefore, have a positive value when machining in the negative X direction and negative when doing it in the opposite direction. "L(+)" in the example above.
- If programmed with a 0 value, the CNC will display the corresponding error message.
- $I_{\pm 5.5}$ Defines the depth of the slot. It is referred to the starting point (X, Z).
- If programmed with a 0 value, the CNC will display the corresponding error message.
- $Q_{\pm 5.5}$ Defines the angular position (in degrees) to orient the spindle to mill the first slot.
- $A_{\pm 5.5}$ Defines the angular increment between two consecutive operations. It is programmed in degrees. Positive when counter-clockwise.
- J 4 Defines the number of slots to be milled. If programmed with a 0 value, the CNC will display the corresponding error message.
- D5.5 Defines the safety distance along the Z axis and indicates how far from the starting point (Z, X) the tool will approach the part. If not programmed, a value of 0 is assumed.
- F5.5 Defines the slot milling feedrate.
- $S_{\pm 5.5}$ RPM and turning direction (sign) of the live tool.
- If not programmed, the live tool does not start up
- N Number of the spindle for the live tool. N1 for spindle S1, N2 for spindle S2 and so on.

1.

MACHINING CANNED CYCLES (ISO)
G163. Radial slot milling cycle (on the face of the part)

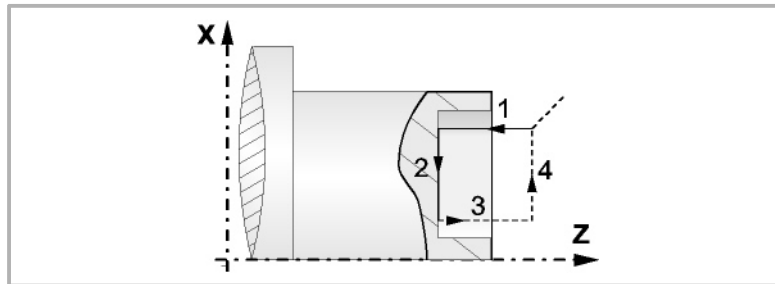
FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

Basic operation

- 1 Rapid move to the approach point located at a "D" distance from the first hole position.
- 2 The CNC turns the live tool on at the rpm and turning direction set by parameter "S".
- 3 Orients the spindle at the angular position set by "Q". If the spindle was turning, it will stop.
- 4 Milling of the slot by following these steps:



In feed at the feedrate selected prior to calling the cycle.

Milling of the slot by moving the X axis at the programmed "F" feedrate.

Rapid withdrawal to the reference point

Rapid return to the starting point

- 5 Depending on the value assigned to parameter "J" (number of slots), the spindle moves to the new position (angular increment "A") and repeats the movements indicated in step 4.
- 6 The live tool stops.

Machining considerations

- The machining conditions (feedrate, live tool rpm, etc.) must be programmed prior to the calling the cycle. Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle. Only the tool radius compensation will be canceled (G40) if it was active.
- If when executing the cycle, it is working in G95 and it has not been working in G94 before, the CNC shows "error 1039. The F has not been programmed in G94".

1.

MACHINING CANNED CYCLES (ISO)

G163. Radial slot milling cycle (on the face of the part)

FAGOR 

**CNC 8060
CNC 8065**

(REF: 1502)

1.

MACHINING CANNED CYCLES (ISO)

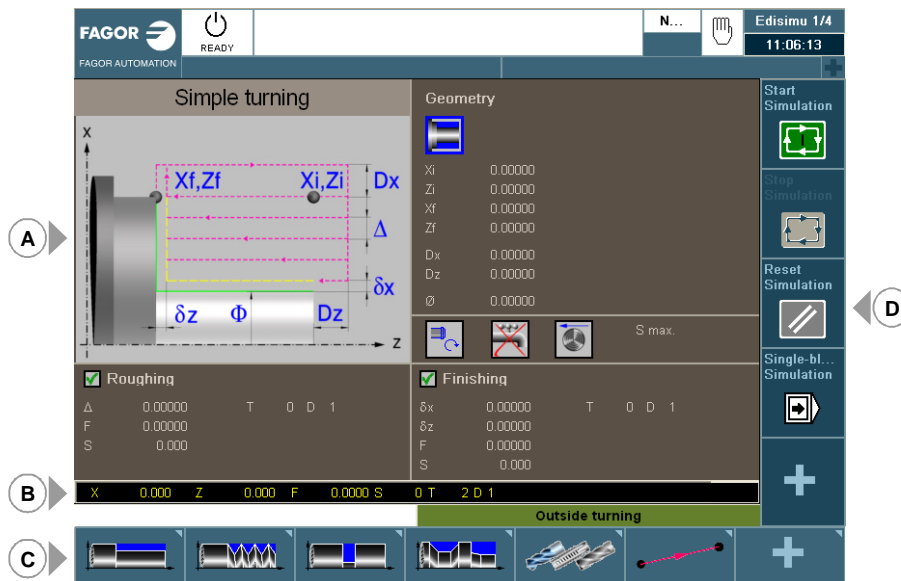
G163. Radial slot milling cycle (on the face of the part)



CNC 8060
CNC 8065

(REF: 1502)

The cycle editor is accessed from the EDISIMU mode, either directly from the softkey menu or by selecting a canned cycle in the part-program and pressing [RECALL]. When selecting a canned cycle, the editor displays the window for defining that canned cycle. Besides editing the cycles, with the cycle editor it is also possible to simulate the cycle graphically even if it is not included in the part-program.



- A Area for editing and simulating canned cycles.
- B Teach-in mode.
- C Softkey menu for selecting various cycles, activating the teach-in mode and configuring the cycle editor.
- D Softkey menu to simulate the cycle selected at the editor.

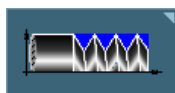
Select the machining cycles.

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.



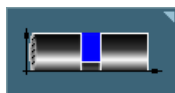
Turning.

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.

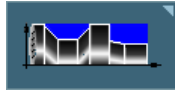


Grooving.

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

2.

CYCLE EDITOR

**Profiles.**

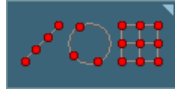
Point-to-point turning, profile turning, ZC plane profile and XC plane profile.

**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 "[2.2 Teach-in mode.](#)" on page 92.

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).

2.1 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.

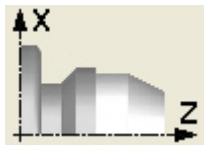
Being this option active, the editor offers in each operation of the cycle the option to edit up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.



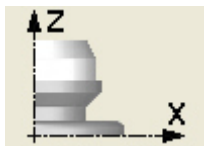
On the cycle screens, the display must be activated in order to see and define the M function data; otherwise, the data will not be displayed.

Select the graphics for a vertical lathe.

Enable the cycles for vertical lathe.

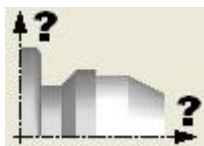


Cycle editor configured for horizontal lathe.



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.

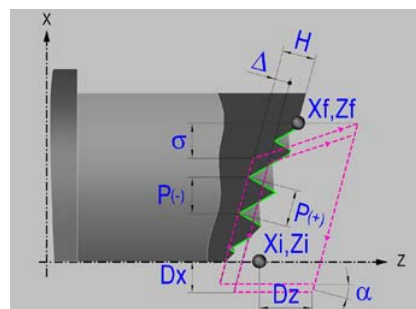
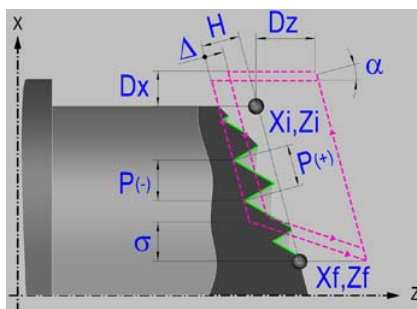
Invert the starting and end points in face threading.



Invert the starting and end points.



Do not invert the starting and end points.

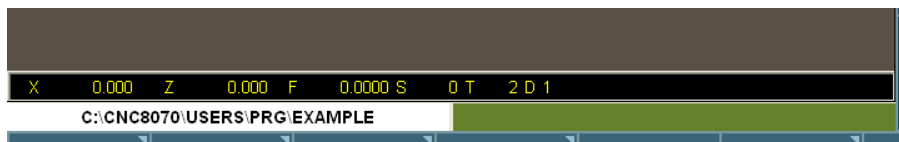


2.2 Teach-in mode.



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.

2.

CYCLE EDITOR
Teach-in mode.



CNC 8060
CNC 8065

(REF: 1502)

2.3 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.

2.

CYCLE EDITOR

Selecting data, profiles and icons

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

2.4 Simulate a canned cycle.

With the canned cycle editor, it is possible to simulate the cycle being edited without having to simulate the whole part-program. During simulation, the editor allows viewing and editing another canned cycle and also returning to the program editor.



If the cycle editor is included in the automatic operating mode, it will not be possible to simulate a cycle.

2.

CYCLE EDITOR
Simulate a canned cycle.

Simulating a cycle

Pressing the [START] icon begins the simulation of the cycle that is being edited. The simulation may be interrupted with the [STOP] icon or canceled with the [RESET] icon.



Once the simulation has started, it is maintained until the cycle is over or the [RESET] icon is pressed. Even when changing cycles or returning to the program editor during simulation, the previous cycle is still in effect during the simulation.

Cycle simulation window

The graphics window (in simulation) is activated by pressing the [START] icon and is canceled by pressing the [RESET] icon. This window is placed over the cycle help graphics; it may be expanded to full screen (or shrunk again) using the key combination [CTRL]+[G].

The lower left corner of the window indicates the name of the cycle and the simulation channel, which will be the channel of the program editor from which the cycle editor has been called.

Configuring the graphic environment

When activating or selecting the graphics window, the horizontal softkey menu shows the available graphic options. For further information on the graphic options, see the chapter on the edit-simulation mode of the operation manual.

Some graphic options can also be edited manually. The editing area is only shown when the window is expanded ([CTRL]+[G]).

The simulated graphics are maintained until erased; i.e. starting to simulate a new cycle does not erase the previous graphics.

Best area for displaying the graphics

The display area may be established from the softkey menu associated with the simulation graphics window or may be left up to the CNC to periodically calculate the best area.

While the graphics window is visible, the key combination [CTRL]+[D] activates the calculation of the best area. From that moment on and until quitting the cycle editor, the CNC periodically calculates the best display area for the graphics. When quitting the graphics, it will assume as the new display area the one calculated last.

Window for simulation and data editing

While the graphics window is selected, it may be switched to the cycle parameter area using the direct access keys. If the cycle is simulated at full screen, the cycle editor may also be accessed by pressing the [ESC] key. To select the graphics window again, use the key combination [CTRL]+[G] or [SHIFT]+[G] or [G].

The horizontal softkey menu will show the graphic options when the graphics window has the focus and those of the cycle editor if otherwise.



CNC 8060
CNC 8065

(REF: 1502)

The simulation in progress is not interrupted while editing data. If the cycle data is changed during simulation, they will be assumed for the next simulation of the cycle; i.e. after RESETTING the simulation in progress once it has finished or after a STOP and RESET to abort it.

Summary of the hotkeys while simulating a cycle.

- | | |
|--------------------|--|
| [CTRL] + [F2] | In the parameter window, it toggles between the cycle parameters and the positioning parameters. |
| [CTRL]+[G] | It selects the graphics window.
It shrinks or expands the graphics window.
It shows the dialog area for the graphics data. |
| [CTRL]+[D] | It activates the periodic calculation of the best display area. |
| [SHIFT]+[G]
[G] | It shows the graphics window when a simulation is running and the parameter editing window is active. |
| [ESC] | If the graphics are shown at full screen, it shows the cycle editor screen. |

2.

CYCLE EDITOR
Simulate a canned cycle.



CNC 8060
CNC 8065

(REF: 1502)

2.

CYCLE EDITOR

Simulate a canned cycle.



CNC 8060
CNC 8065

(REF: 1502)

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

3

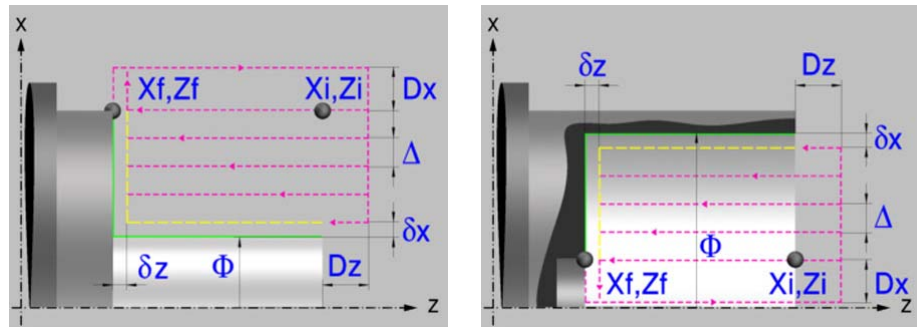
When pressing the softkey, the editor shows the cycle used last. When pressing the same softkey again, the menu shows all the cycles of the group.



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

3.1 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 .

The X axis coordinates must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

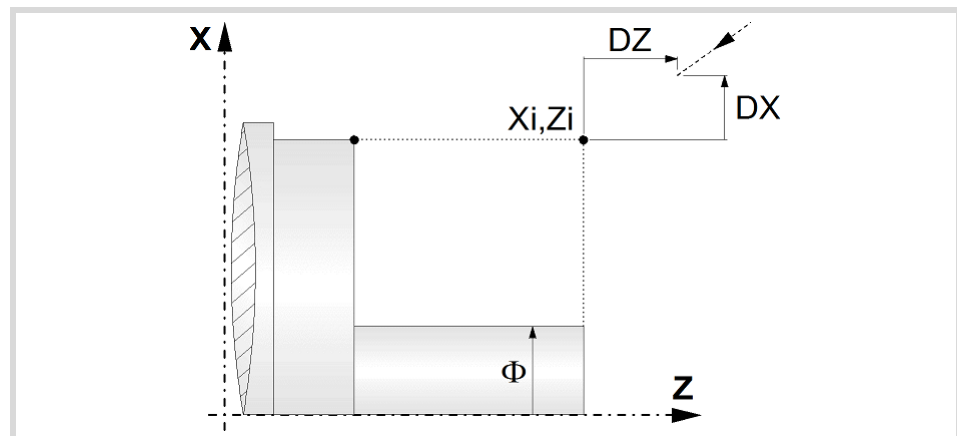
When the surface to be machined is not fully cylindrical, the CNC analyzes the coordinates on X of the starting and end points and takes as starting point on X the coordinate farthest from the final diameter.

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.



3.

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



CNC 8060
CNC 8065

(REF: 1502)

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.

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.

Machining feedrate.

F Machining feedrate.

3.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Simple turning cycle.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

3.

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 (δx , δz).

Finishing stocks.

δx , δz 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.

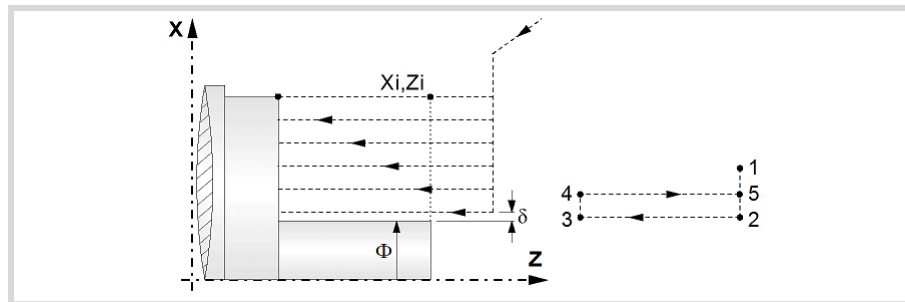
3.1.1 Basic operation.

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.

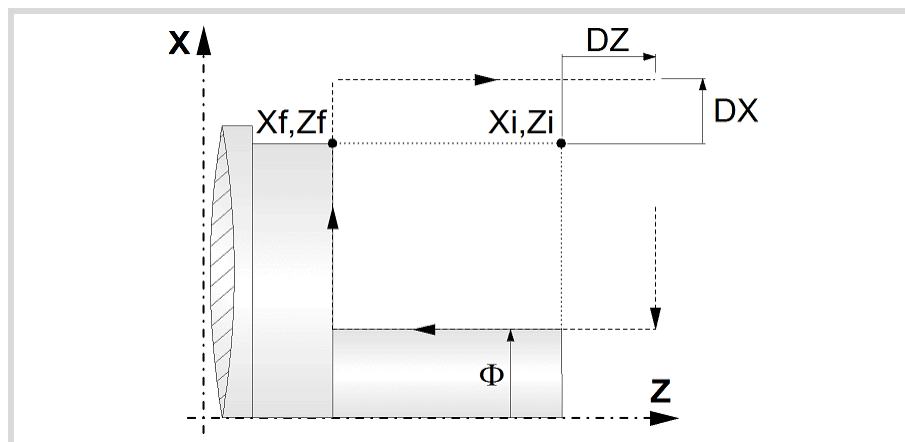
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



- 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).

3.

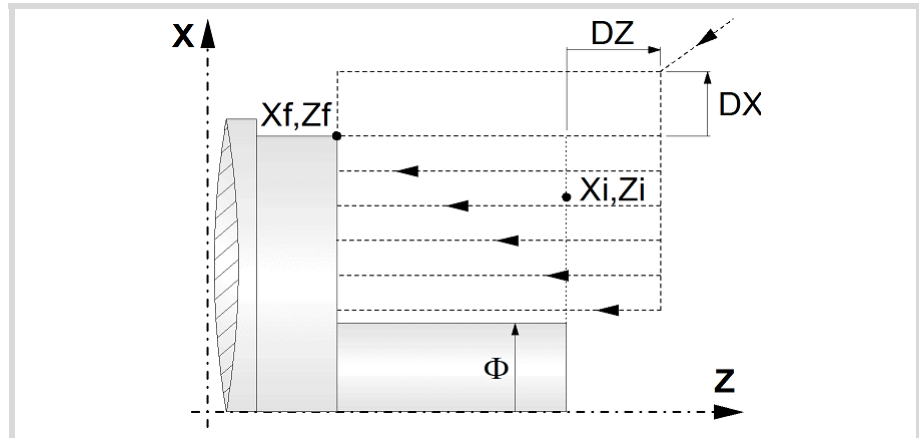
CANNED CYCLES OF THE EDITOR. TURNING AND FACING.
Simple turning cycle.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

- 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.

3.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

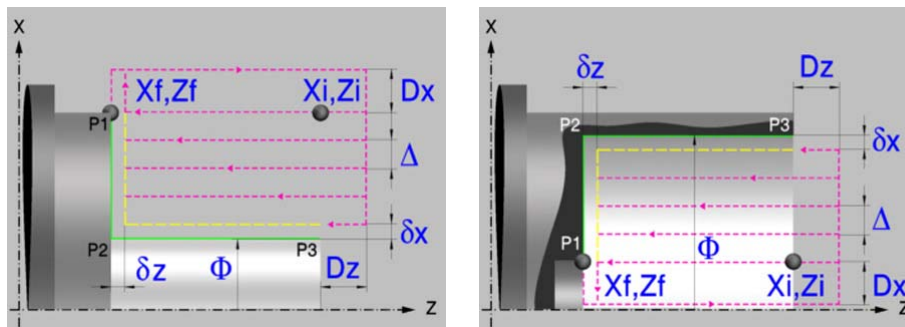
Simple turning cycle.

CNC 8060
CNC 8065

(REF: 1502)

3.2 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 .

The X axis coordinates must be programmed in the active units, radius or diameter. 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.

3.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

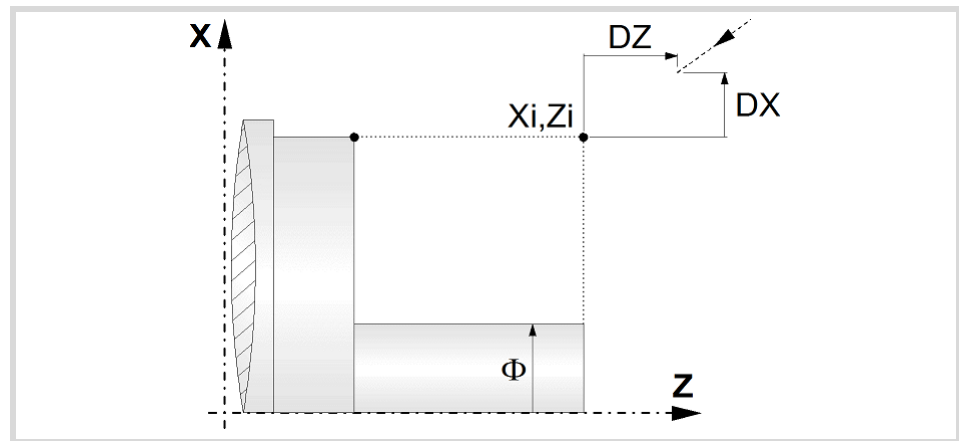
Turning cycle with vertex rounding.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

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



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.

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.

3.

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

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

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.

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 (δx , δz).

Finishing stocks.

δx , δz 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.

3.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Turning cycle with vertex rounding.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

- 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.

3.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Turning cycle with vertex rounding.



CNC 8060
CNC 8065

(REF: 1502)

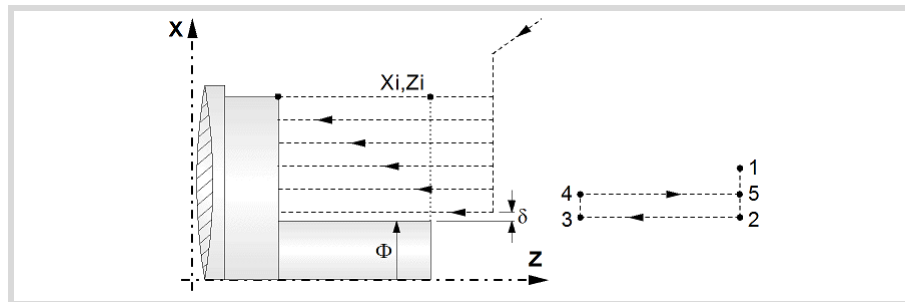
3.2.1 Basic operation.

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.

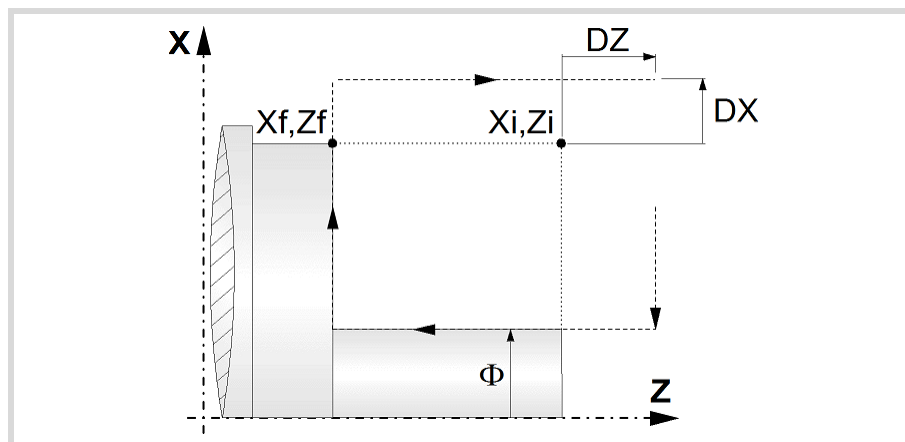
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



- 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).

3.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

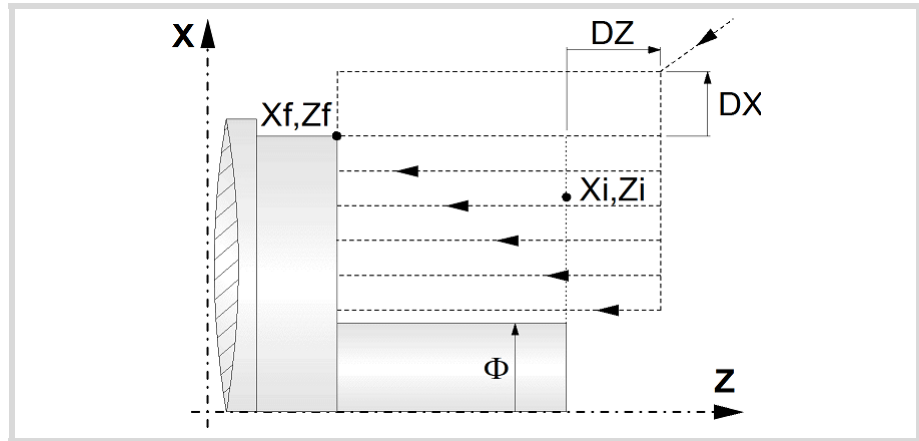
Turning cycle with vertex rounding.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

- 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.

3.

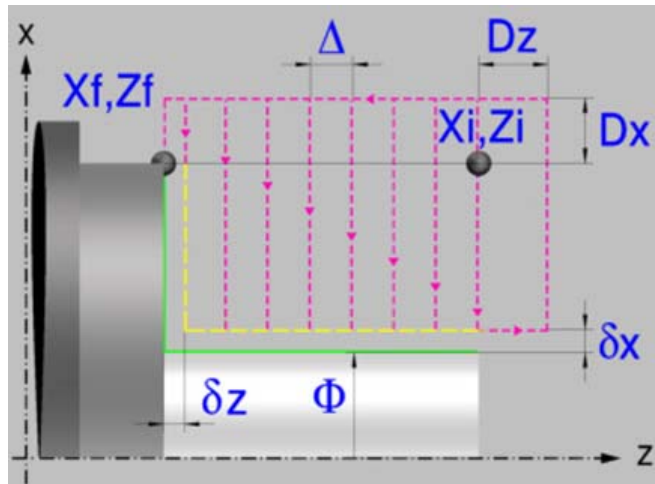
CANNED CYCLES OF THE EDITOR. TURNING AND FACING.
Turning cycle with vertex rounding.



CNC 8060
CNC 8065

(REF: 1502)

3.3 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 .

The X axis coordinates must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

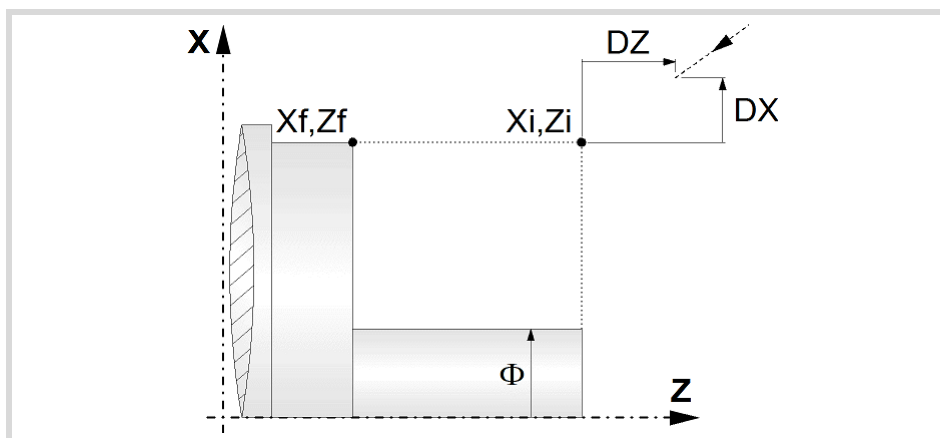
When the surface to be machined is not fully cylindrical, the CNC analyzes the coordinates on X of the starting and end points and takes as starting point on X the coordinate farthest from the final diameter.

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.

Φ Final diameter.

3.

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

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

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 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.

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.

3.

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



CNC 8060
CNC 8065

(REF: 1502)

- 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 (δx , δz).

Finishing stocks.

δx , δz 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.

3.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Simple facing cycle.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

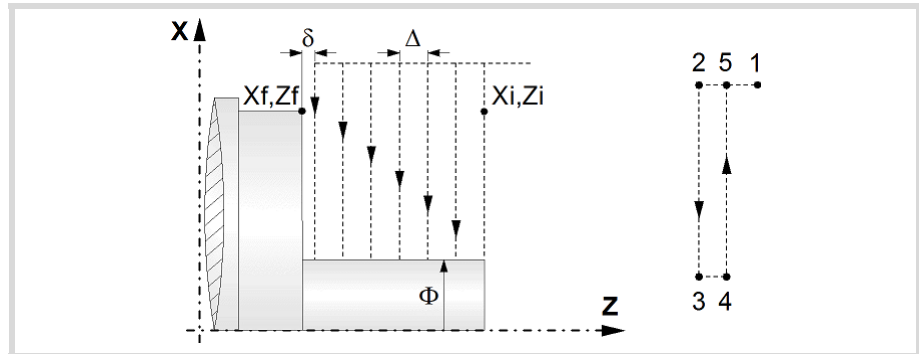
3.3.1 Basic operation.

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.

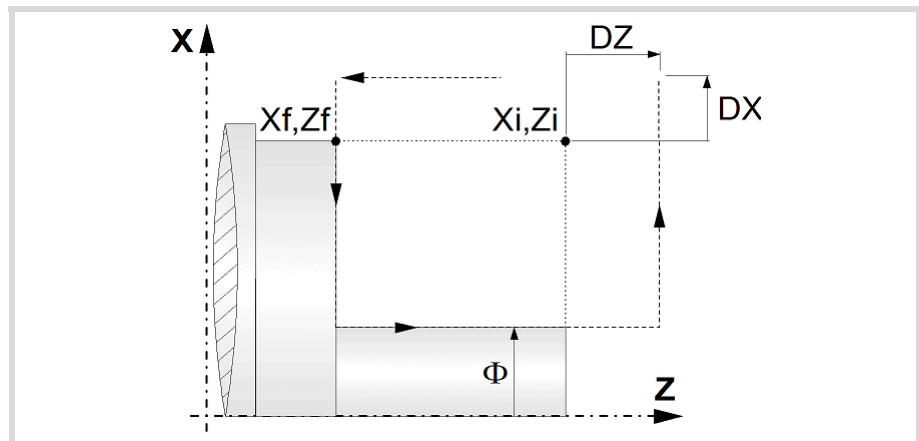
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



- 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).

3.

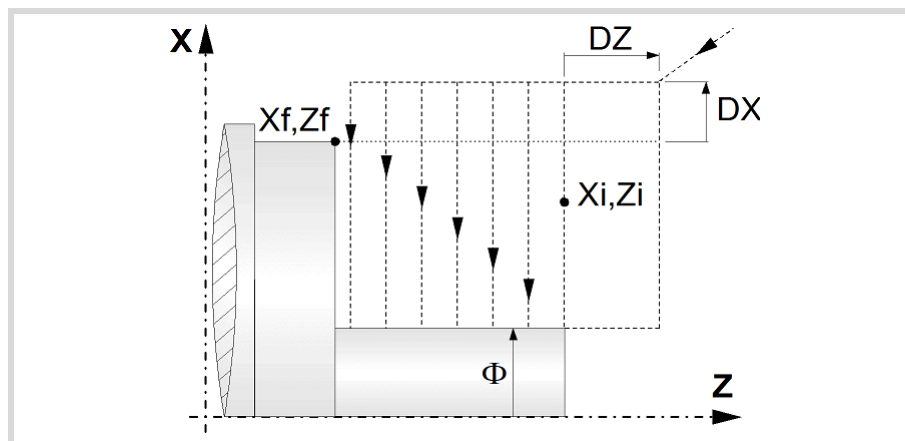
CANNED CYCLES OF THE EDITOR. TURNING AND FACING.
Simple facing cycle.



CNC 8060
CNC 8065

(REF: 1502)

- 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.

3.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Simple facing cycle.



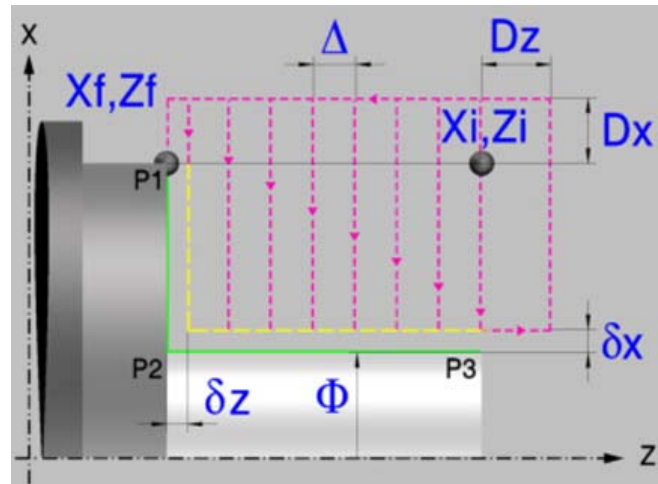
CNC 8060
CNC 8065

(REF: 1502)

3.4 Facing cycle with vertex rounding.

3.

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



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 .

The X axis coordinates must be programmed in the active units, radius or diameter. 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).

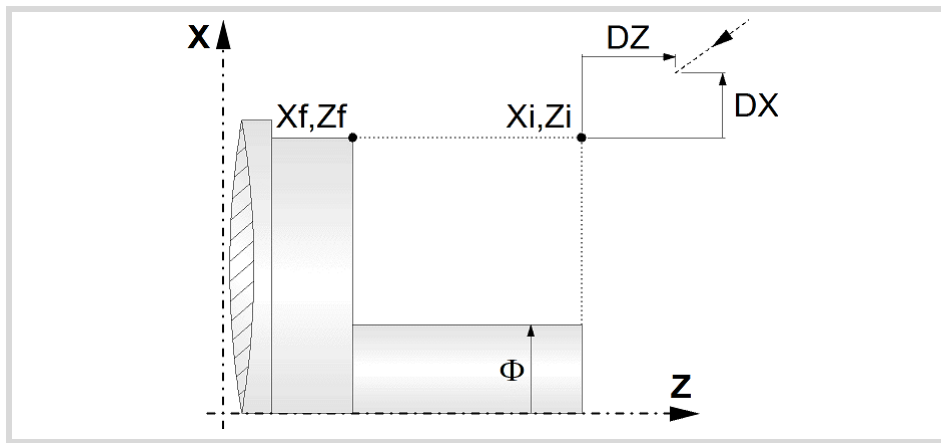
D_x, D_z Safety distance.



CNC 8060
CNC 8065

(REF: 1502)

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



3.

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.

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.



CNC 8060
CNC 8065

(REF: 1502)

3.

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.

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 (δx , δz).

Finishing stocks.

δx , δz 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.

3.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Facing cycle with vertex rounding.



CNC 8060
CNC 8065

(REF: 1502)

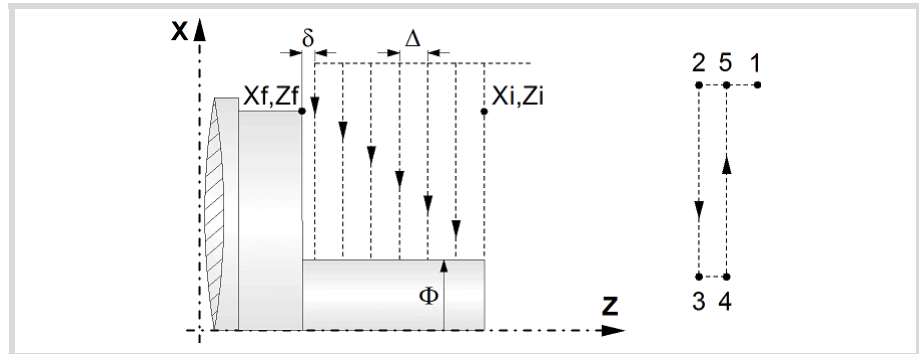
3.4.1 Basic operation.

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.

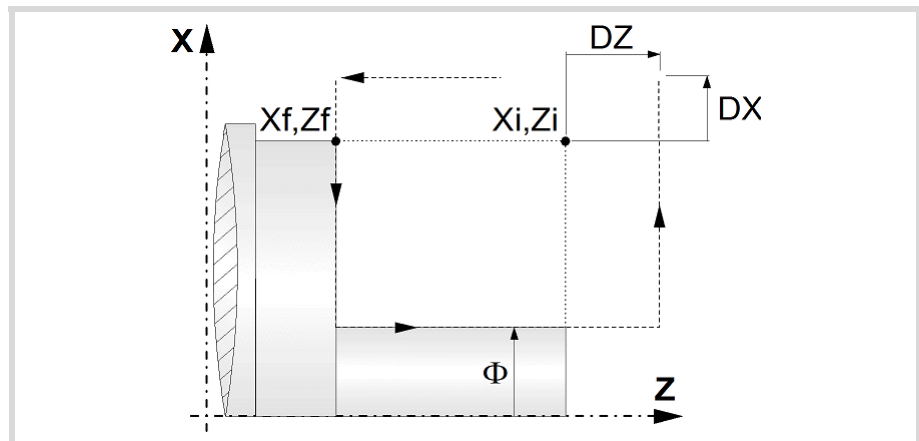
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



- 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).

3.

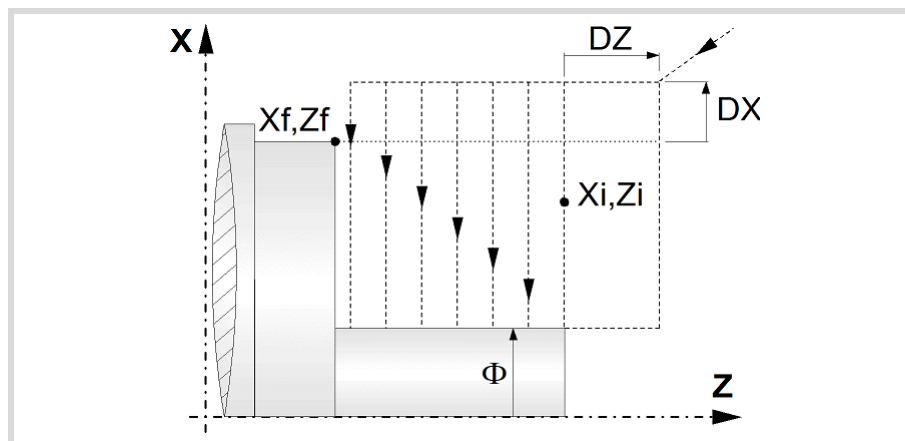
CANNED CYCLES OF THE EDITOR. TURNING AND FACING.
Facing cycle with vertex rounding.



CNC 8060
CNC 8065

(REF: 1502)

- 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.

3.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Facing cycle with vertex rounding.

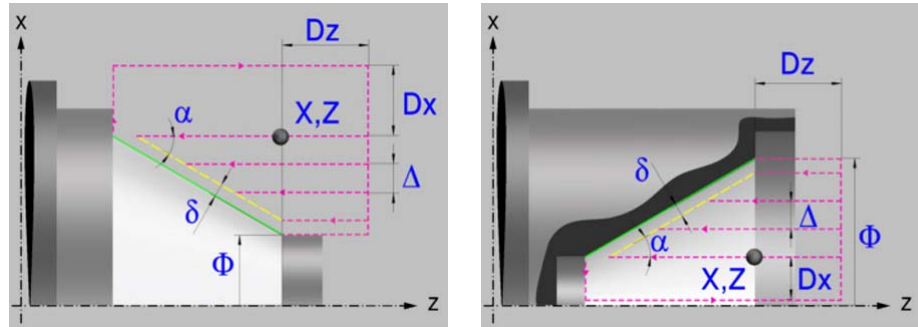


CNC 8060
CNC 8065

(REF: 1502)

3.5 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.

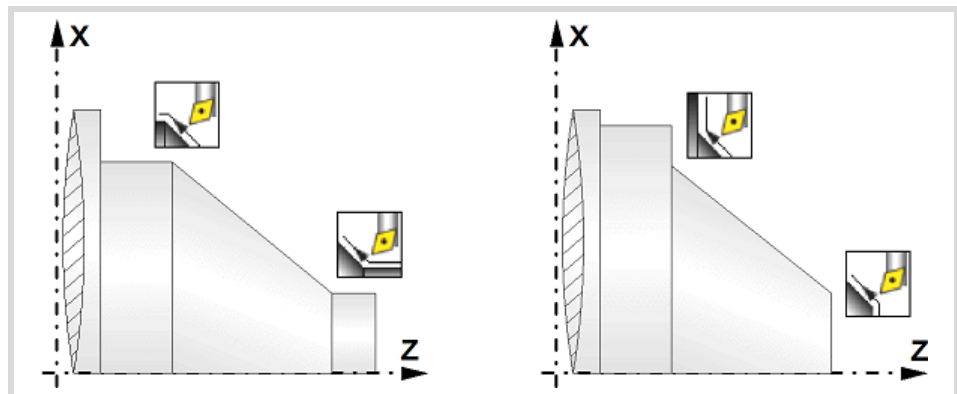
Shape of the part after the tapered segment.



Final concave corner.



Final convex corner.



3.

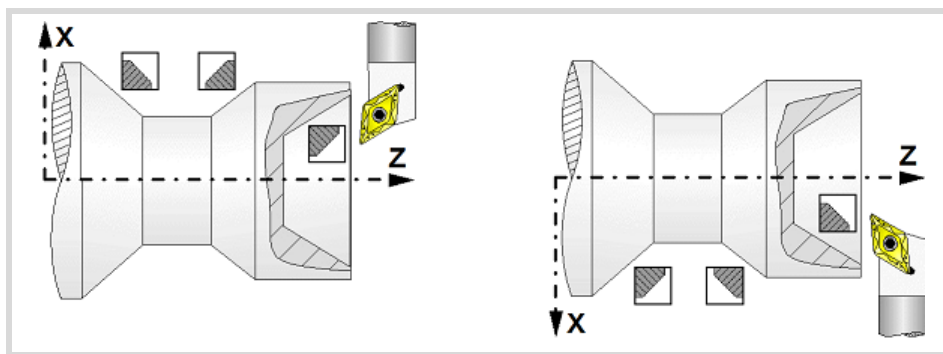
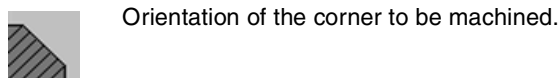
CANNED CYCLES OF THE EDITOR. TURNING AND FACING.
Vertex chamfering cycle.



CNC 8060
CNC 8065

(REF: 1502)

Work quadrant.



Coordinates of the starting point

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

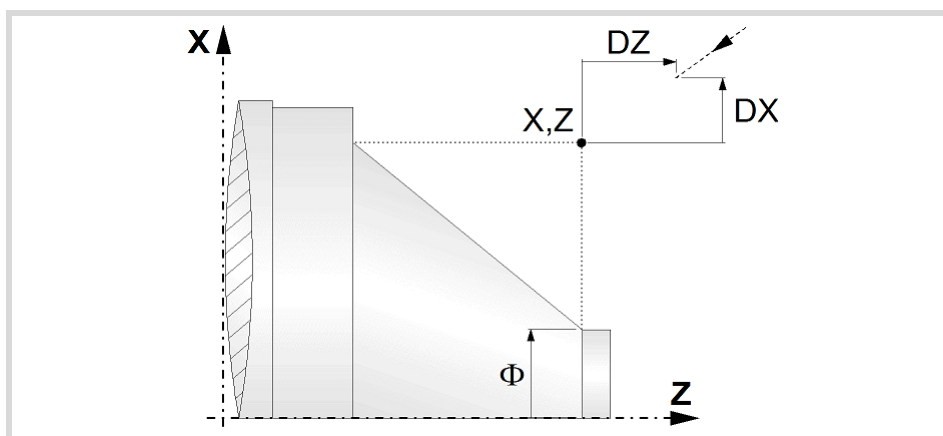
The X axis coordinate must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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.

Φ Final diameter.

Taper angle.

α 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.

3.

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



CNC 8060
CNC 8065

(REF: 1502)

3.

Coolant status.

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



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

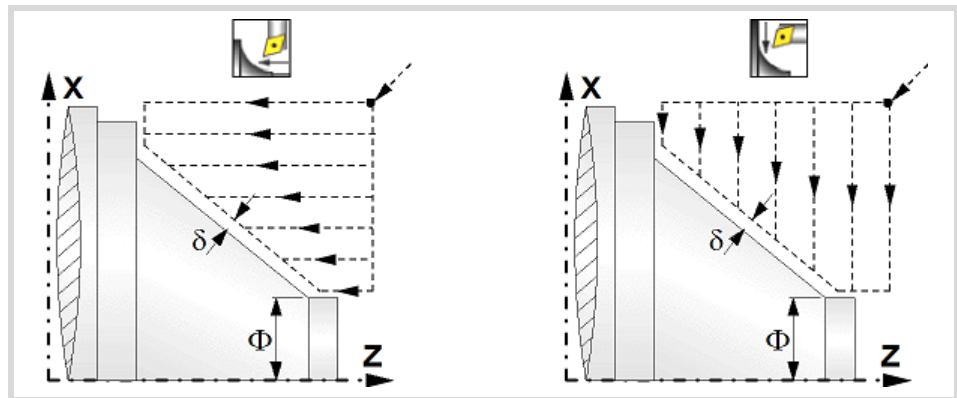
Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

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.

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 (δx , δz).

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.

δ Finishing stock depending on the cutter edge.

δx , δz 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.

3.

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

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

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.

3.

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



CNC 8060
CNC 8065

(REF: 1502)

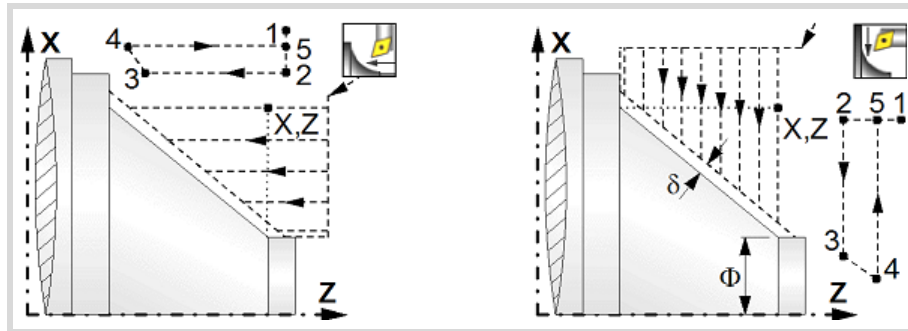
3.5.1 Basic operation.

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.

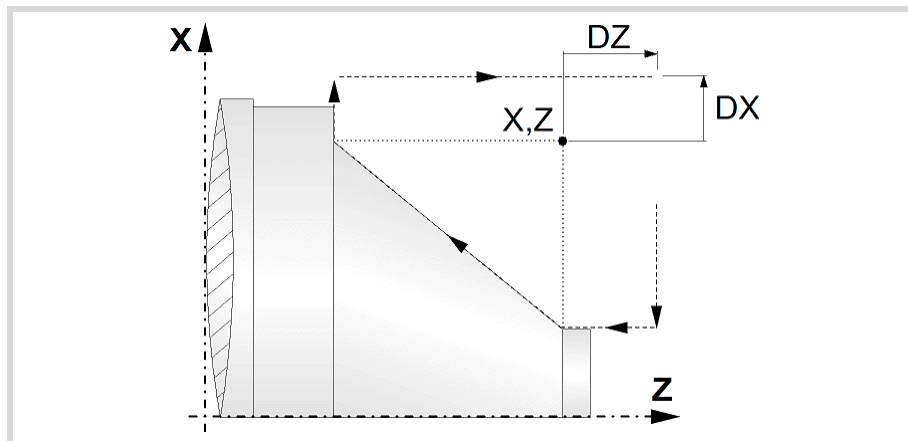
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



- 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).

3.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Vertex chamfering cycle.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

- 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.

3.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Vertex chamfering cycle.

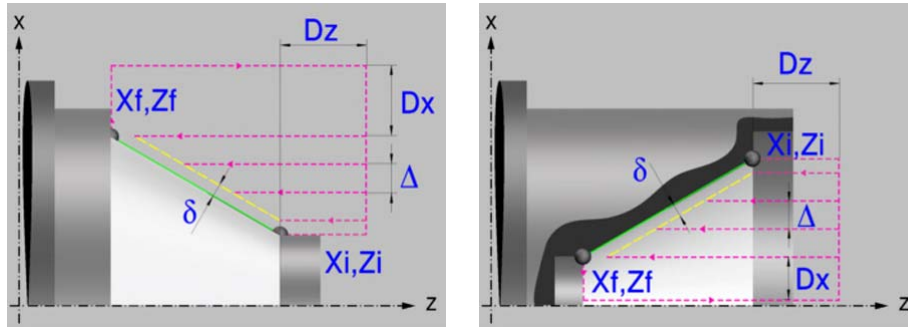


CNC 8060
CNC 8065

(REF: 1502)

3.6 "Chamfering between points" cycle.

The cycle allows outside and inside machining.



3.

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

Shape of the part before the tapered segment.



Initial concave corner.



Initial convex corner.

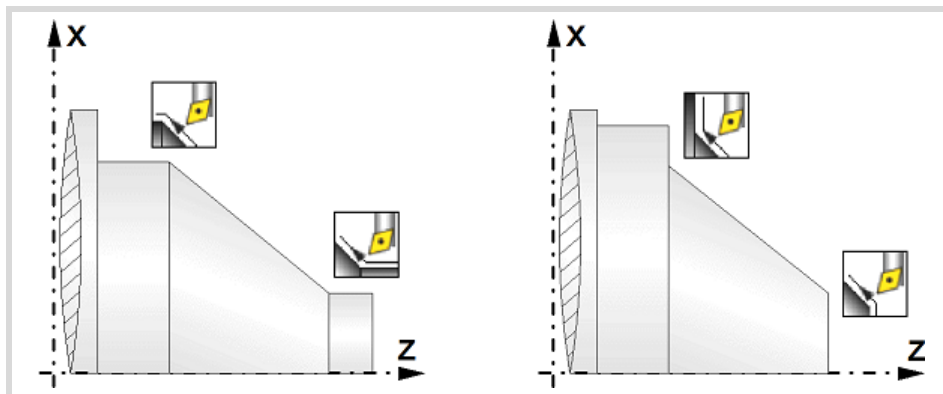
Shape of the part after the tapered segment.



Final concave corner.



Final convex corner.

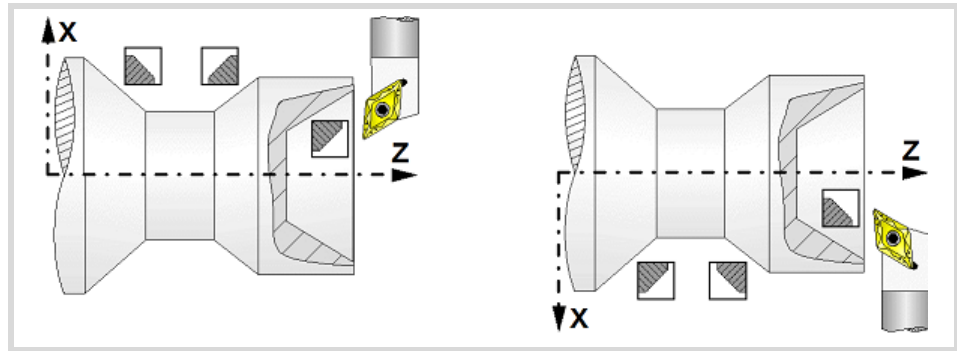


CNC 8060
CNC 8065

(REF: 1502)

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.

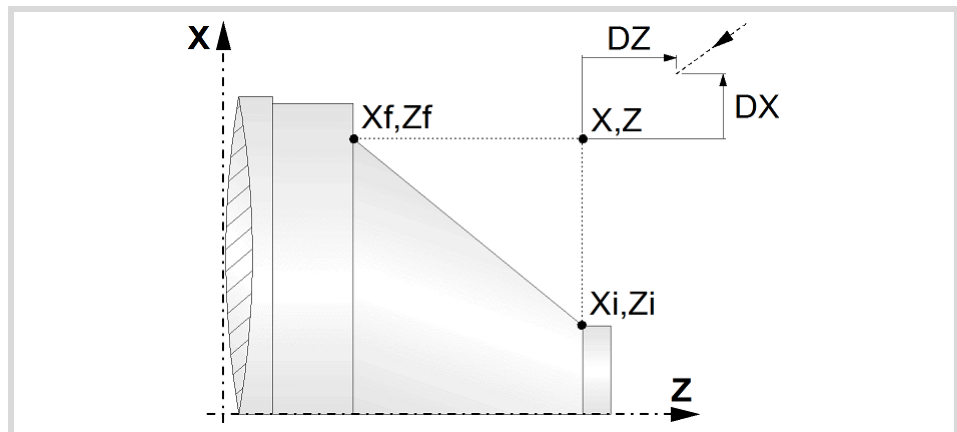
The X axis coordinates must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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.

3.

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



CNC 8060
CNC 8065

(REF: 1502)

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.

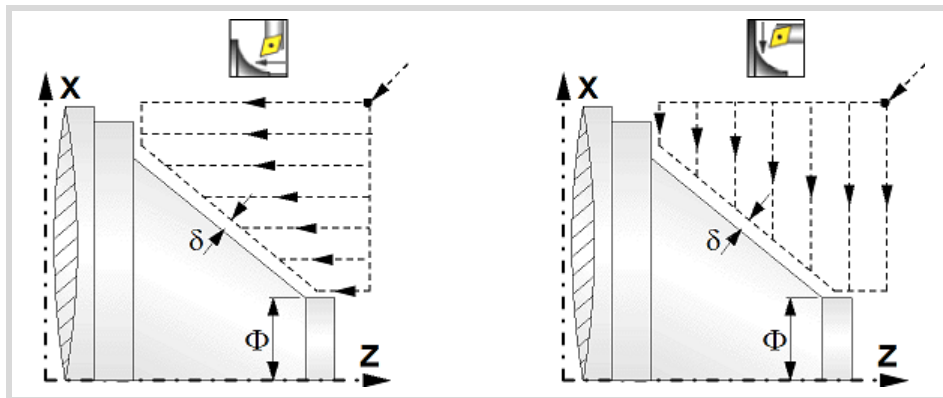
Machining direction.



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.

3.

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



CNC 8060
CNC 8065

(REF: 1502)

3.

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 (δx , δz).**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.

 δ Finishing stock depending on the cutter edge. δx , δz 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.

3.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

"Chamfering between points" cycle.



CNC 8060
CNC 8065

(REF: 1502)

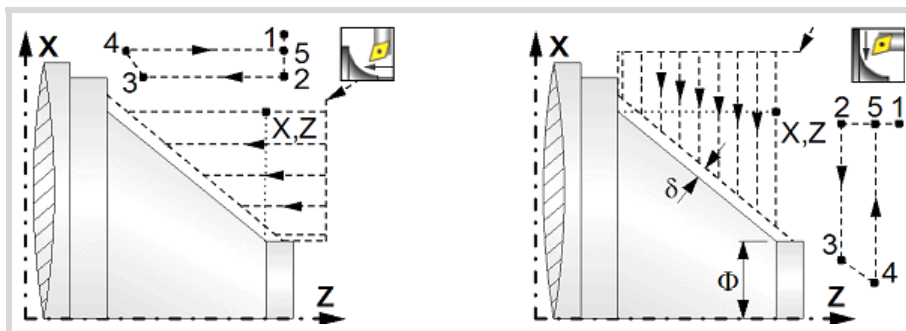
3.6.1 Basic operation.

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.

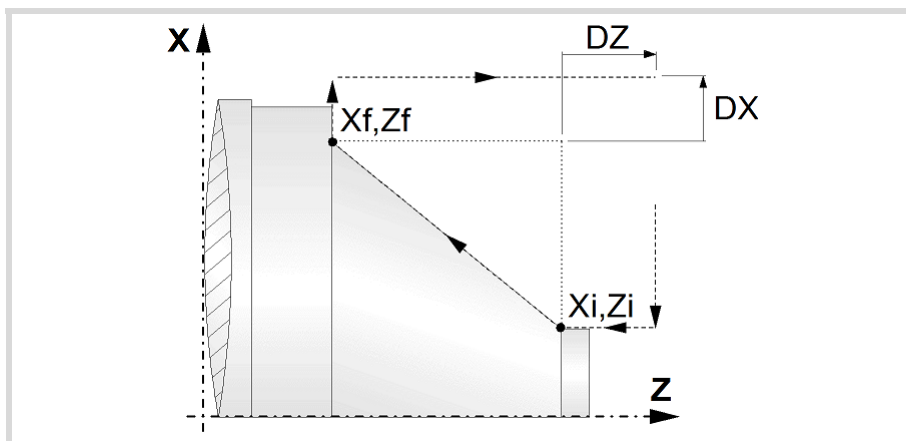
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



- 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).

3.

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

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

Canned cycles (.T. model)

- 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.

3.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

"Chamfering between points" cycle.

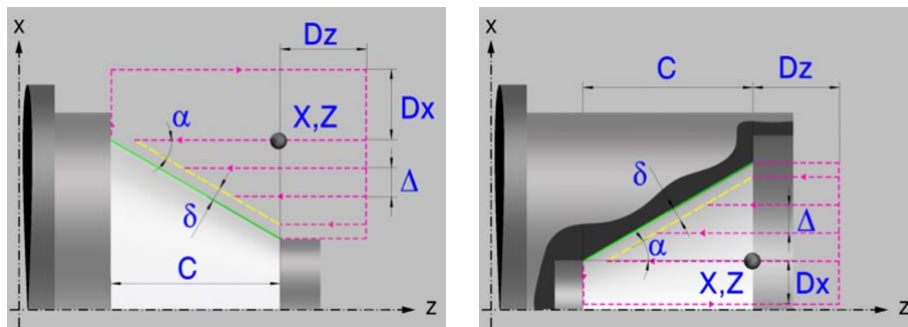
FAGOR 

**CNC 8060
CNC 8065**

(REF: 1502)

3.7 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.

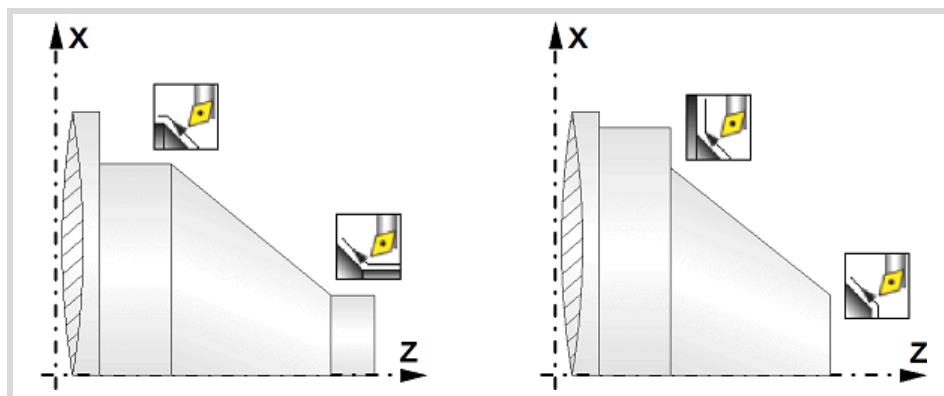
Shape of the part after the tapered segment.



Final concave corner.



Final convex corner.



3.

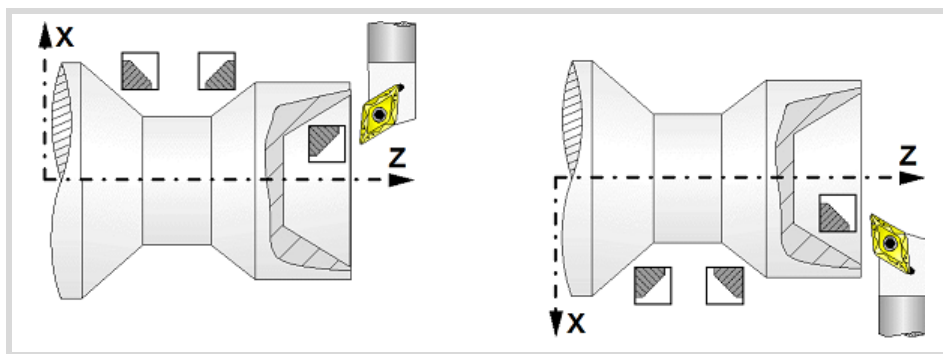
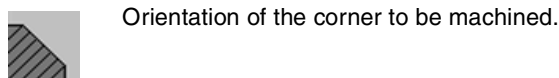
CANNED CYCLES OF THE EDITOR. TURNING AND FACING.
Vertex chamfering cycle 2.



CNC 8060
CNC 8065

(REF: 1502)

Work quadrant.



Coordinates of the starting point

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

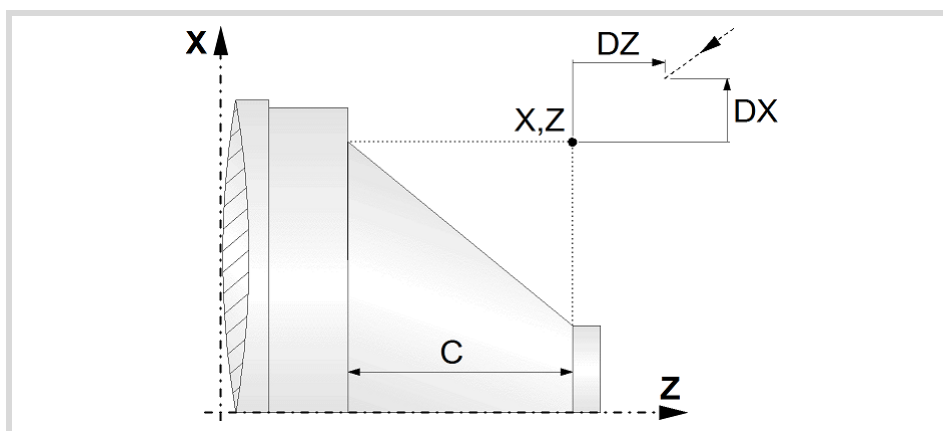
The X axis coordinate must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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.

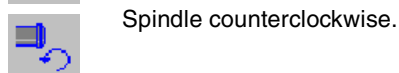
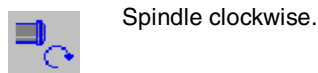
Taper angle.

α Taper angle with respect to the abscissa axis.

Spindle parameters.

Spindle turning direction

The cycle lets select the spindle turning direction.



3.

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



CNC 8060
CNC 8065

(REF: 1502)

3.

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

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



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

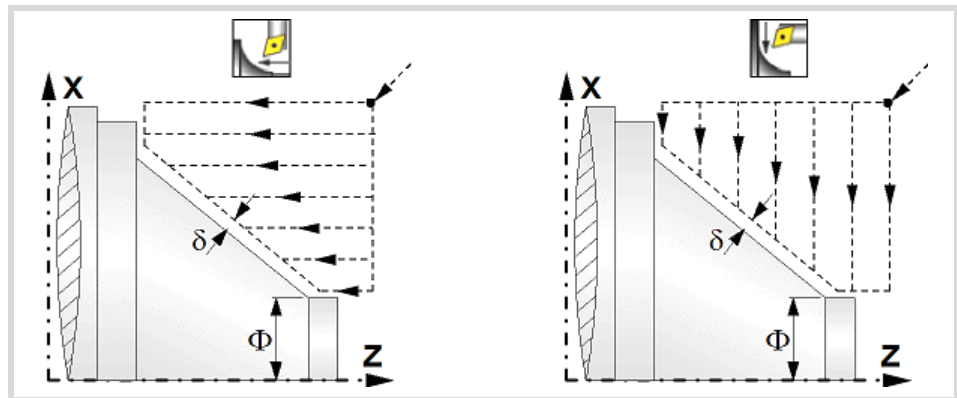
Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

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.

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 (δx , δz).

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.

δ Finishing stock depending on the cutter edge.

δx , δz 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.

3.

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

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

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.

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

Vertex chamfering cycle 2.

**CNC 8060
CNC 8065**

(REF: 1502)

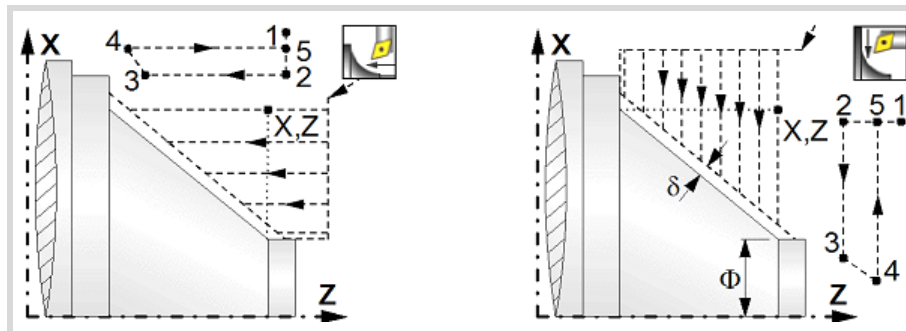
3.7.1 Basic operation.

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.

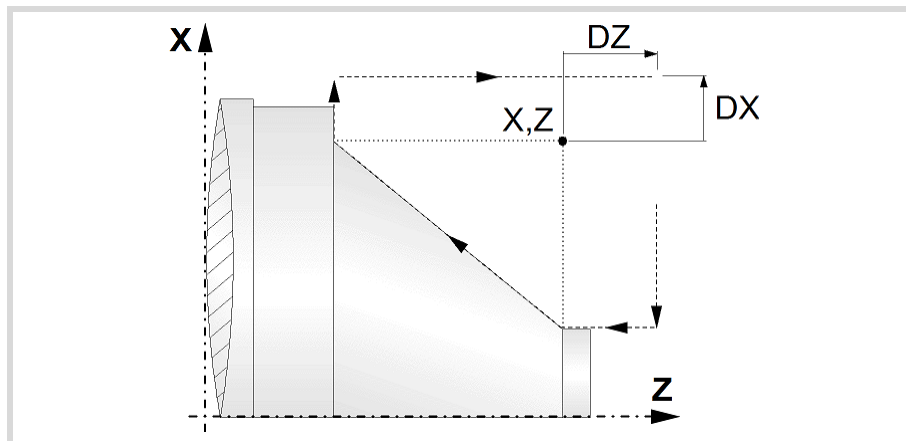
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



- 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).

3.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Vertex chamfering cycle 2.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

- 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.

3.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Vertex chamfering cycle 2.

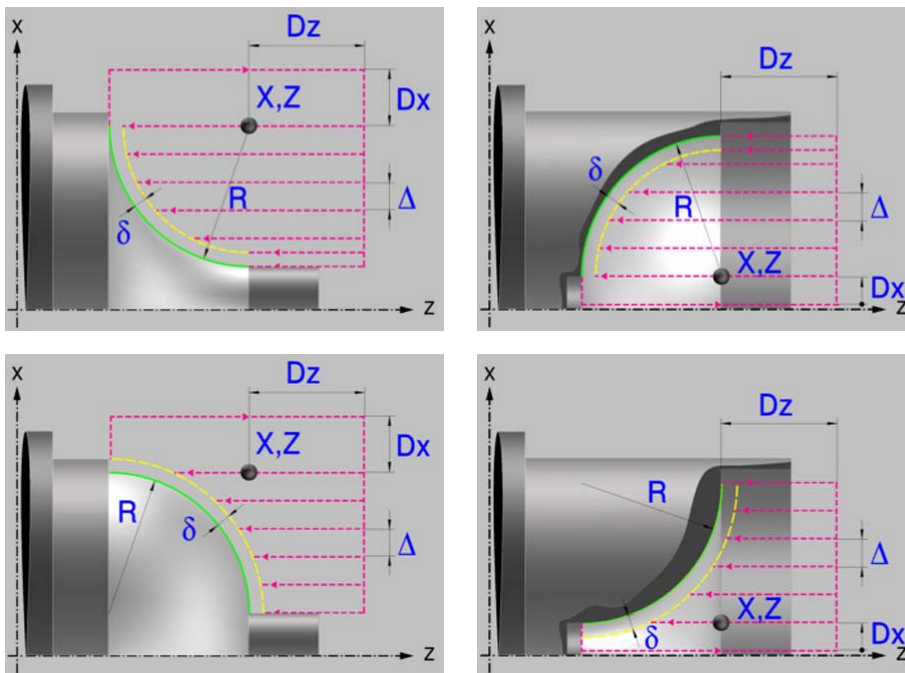


CNC 8060
CNC 8065

(REF: 1502)

3.8 Vertex rounding cycle.

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



3.

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.

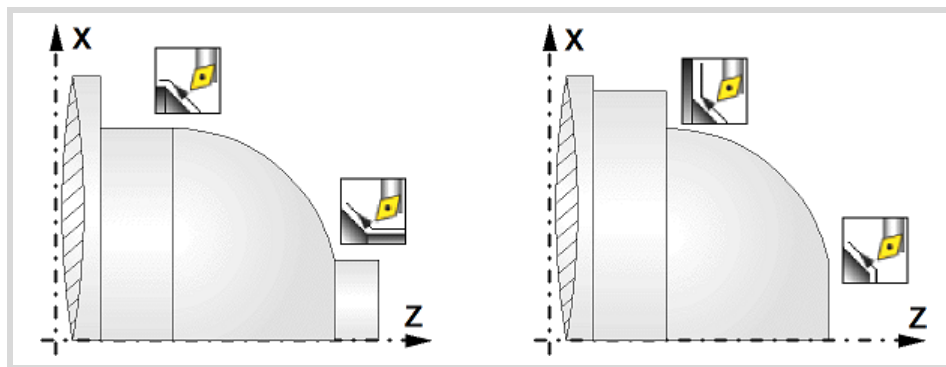


CNC 8060
CNC 8065


(REF: 1502)

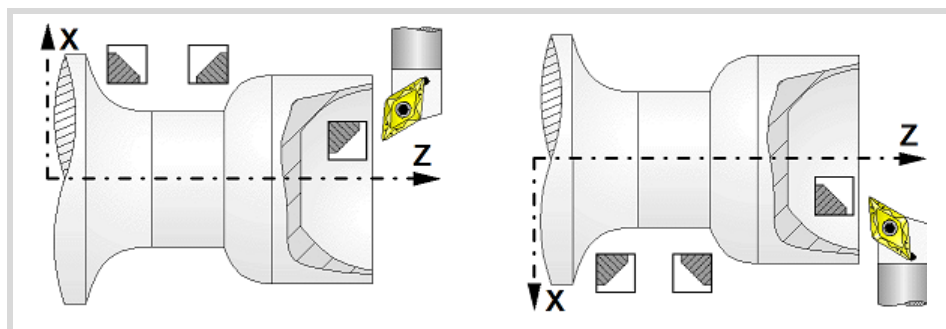
3.

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.

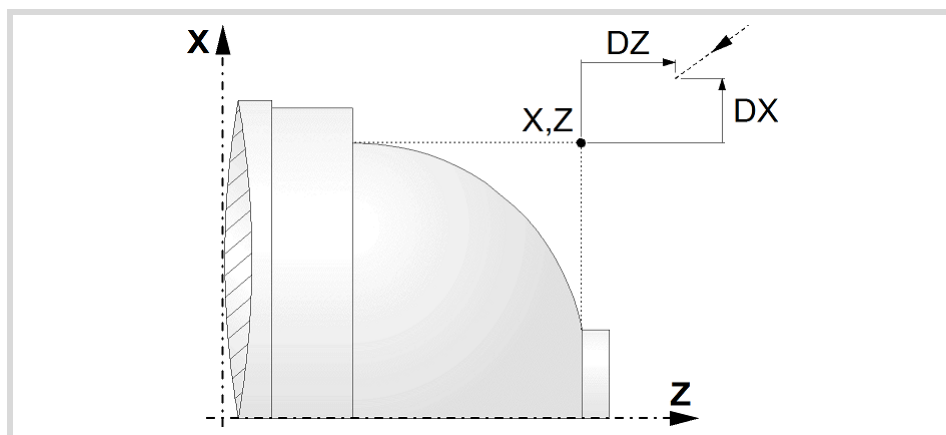
The X axis coordinate must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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.



CNC 8060
CNC 8065

(REF: 1502)

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.

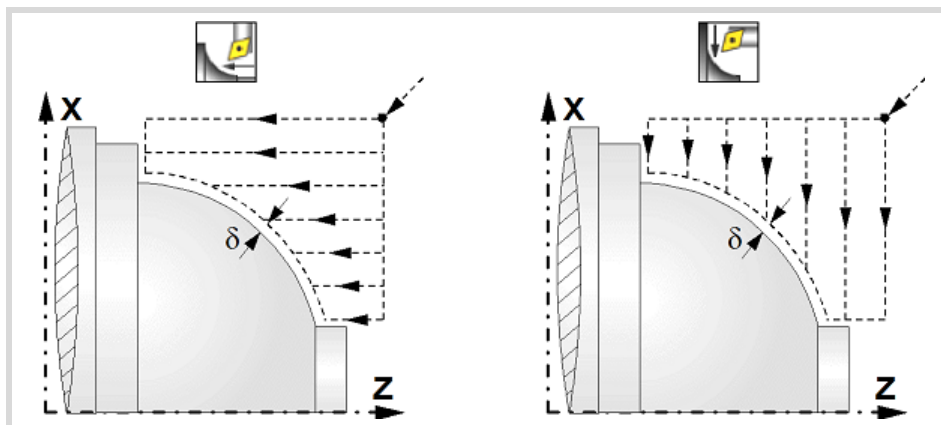
Machining direction.



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.

3.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Vertex rounding cycle.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

3.

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.

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 (δx , δz).

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.

δ Finishing stock depending on the cutter edge.

δx , δz 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.

3.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Vertex rounding cycle.

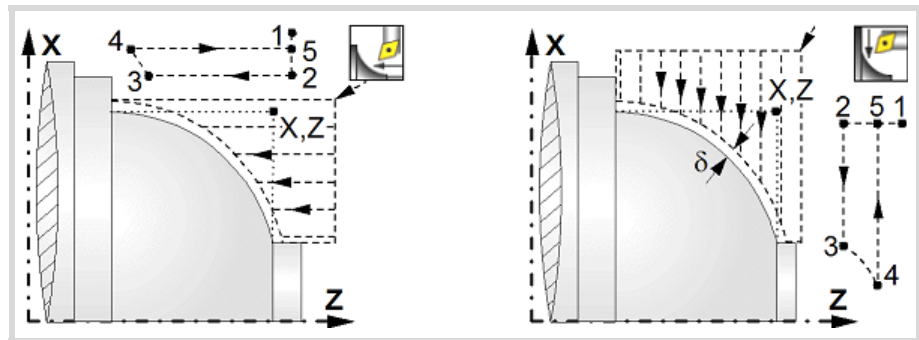
3.8.1 Basic operation.

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.

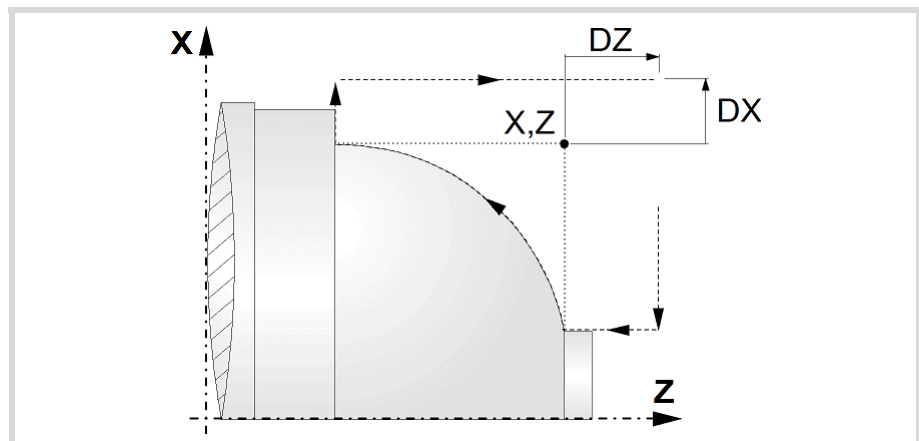
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



- 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).

3.

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

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

Canned cycles (.T. model)

- 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.

3.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

Vertex rounding cycle.

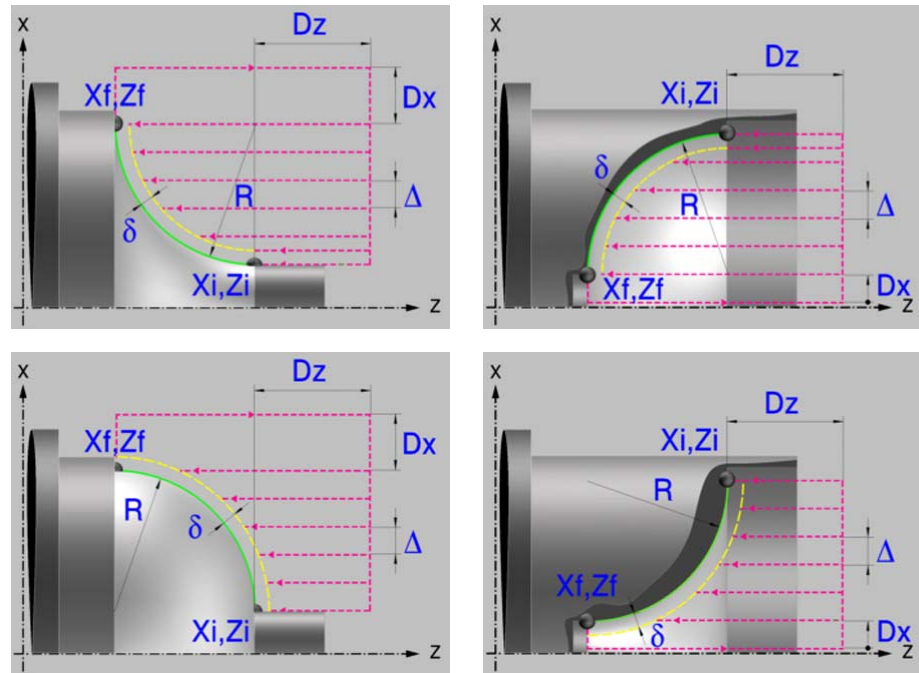
FAGOR 

**CNC 8060
CNC 8065**

(REF: 1502)

3.9 "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.

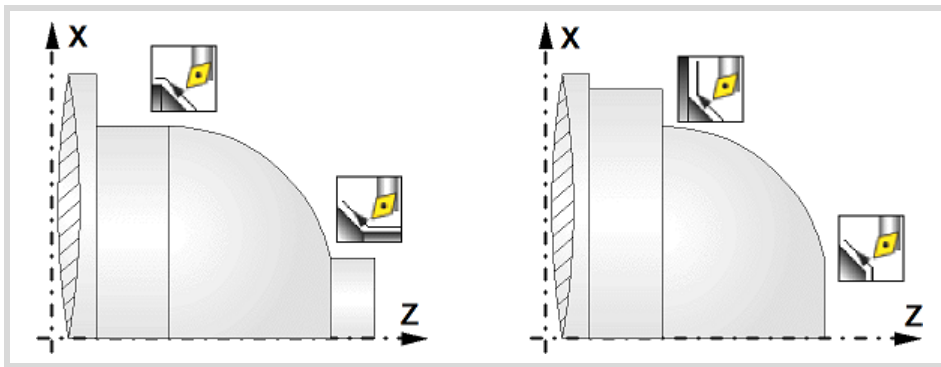
3.

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

FAGOR 

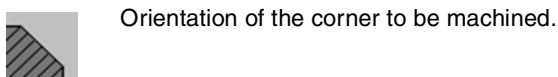
CNC 8060
CNC 8065

(REF: 1502)

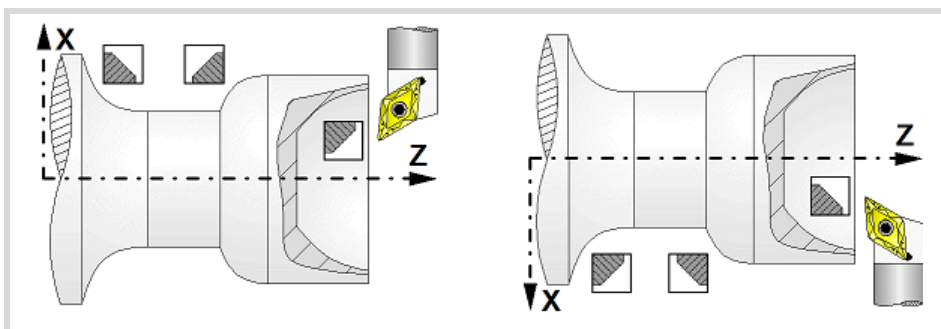


3.

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.

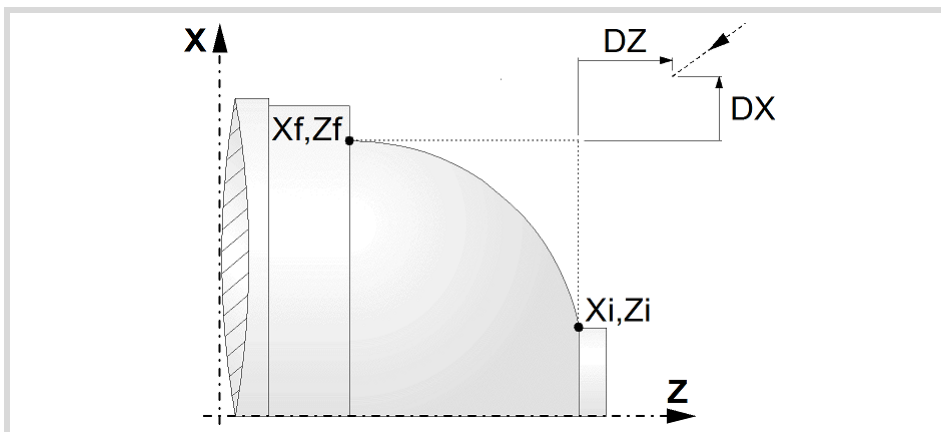
The X axis coordinates must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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.

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



CNC 8060
CNC 8065

(REF: 1502)

3.

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.

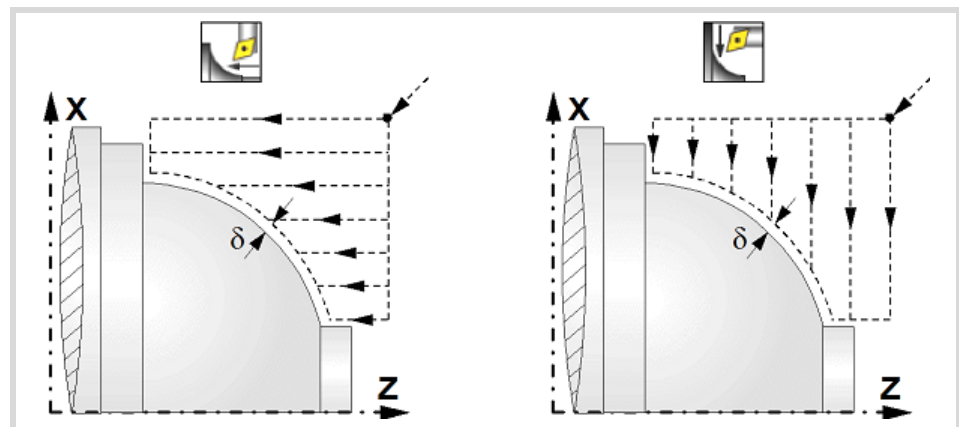
Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

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.

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.

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 (δx , δz).

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.

δ Finishing stock depending on the cutter edge.

δx , δz Finishing stock on each axis.

3.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

"Rounding between points" cycle.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

3.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

"Rounding 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 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.

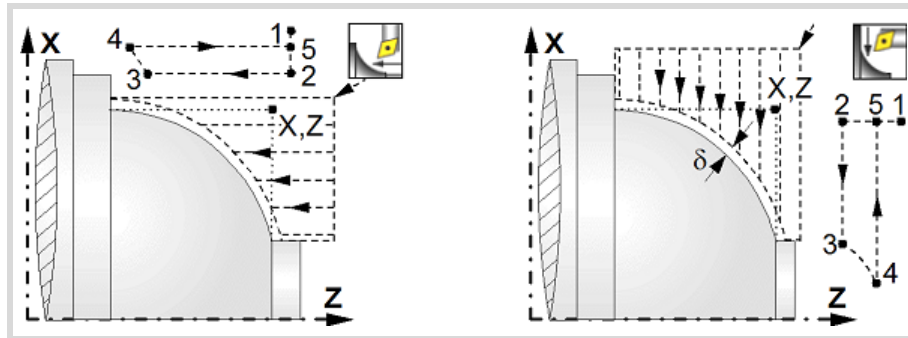
3.9.1 Basic operation.

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.

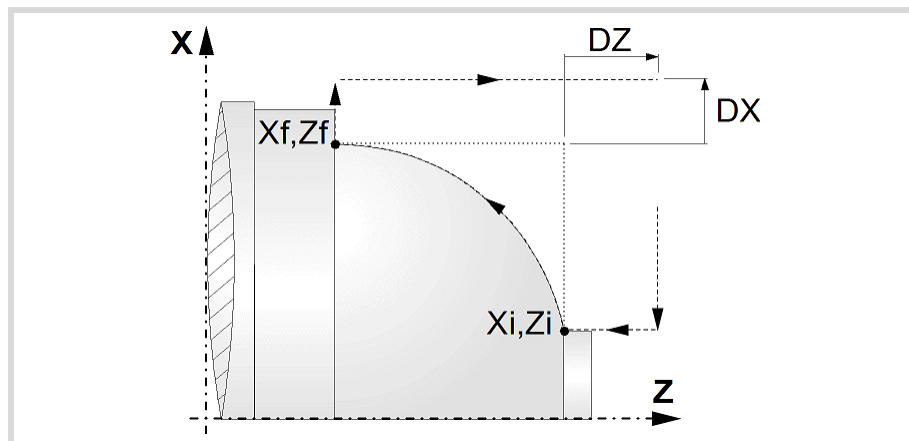
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



- 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).

3.

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

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

- 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.

3.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

"Rounding between points" cycle.



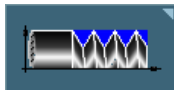
CNC 8060
CNC 8065

(REF: 1502)

CANNED CYCLES OF THE EDITOR. TAPPING.

4

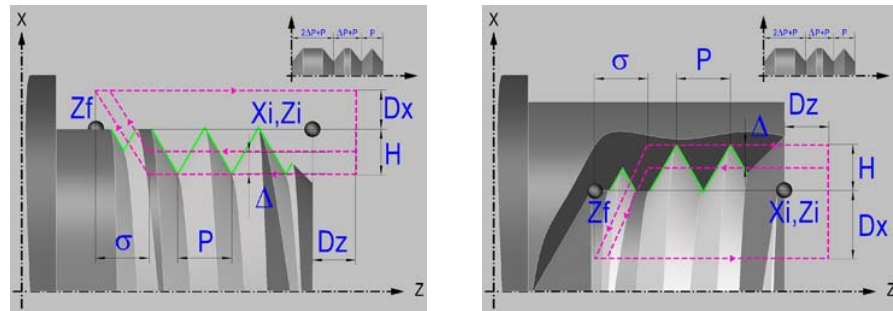
When pressing the softkey, the editor shows the cycle used last. When pressing the same softkey again, the menu shows all the cycles of the group.



- Longitudinal threading.
- Taper threading.
- Face threading.
- Thread repair.
- Multi-entry (start) threading.

4.1 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.

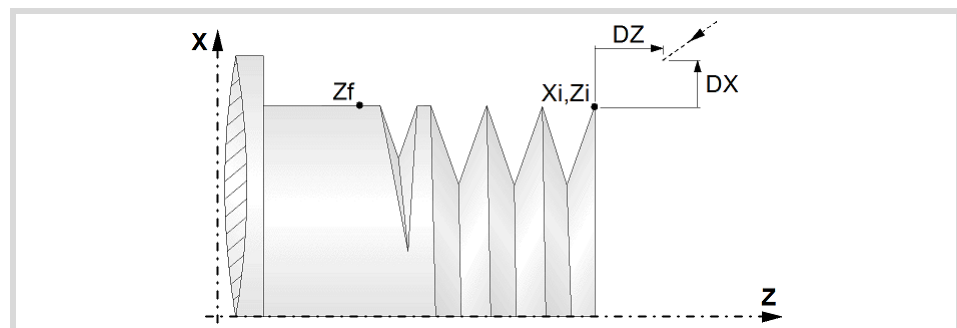
The X axis coordinate must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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.

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.

4.

CANNED CYCLES OF THE EDITOR. TAPPING.
Longitudinal threading cycle.



CNC 8060
CNC 8065

(REF: 1502)

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.

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.

ΔP 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.

ϕ Thread diameter (except in API threads).



Units used to enter the thread data (mm or inches).

4.

CANNED CYCLES OF THE EDITOR. TAPPING.
Longitudinal threading cycle.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

4.

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.**

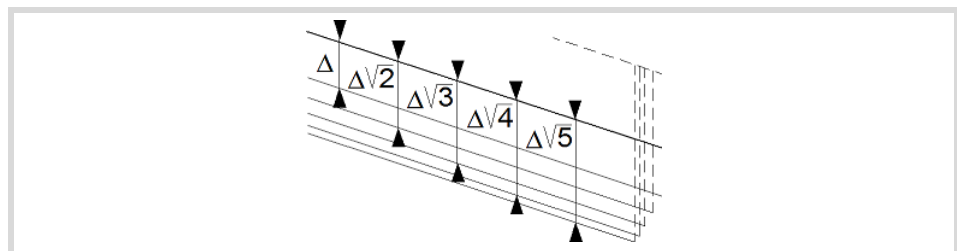
S Spindle speed.

Depth of pass.

Δ Maximum penetration step.

δ 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 Δ , $\Delta\sqrt{2}$, $\Delta\sqrt{3}$, $\Delta\sqrt{4}$...

**Repeat the last pass.**

The cycle repeats the last pass.



The cycle does not repeat the last pass.

Normal thread or blind thread.

Normal thread. Parameter σ 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.

4.

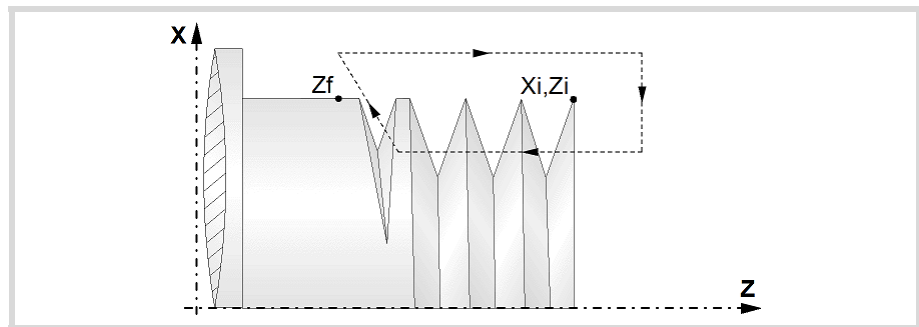
CANNED CYCLES OF THE EDITOR. TAPPING.

Longitudinal threading cycle.

4.1.1 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).

4.

CANNED CYCLES OF THE EDITOR. TAPPING.
Longitudinal threading cycle.

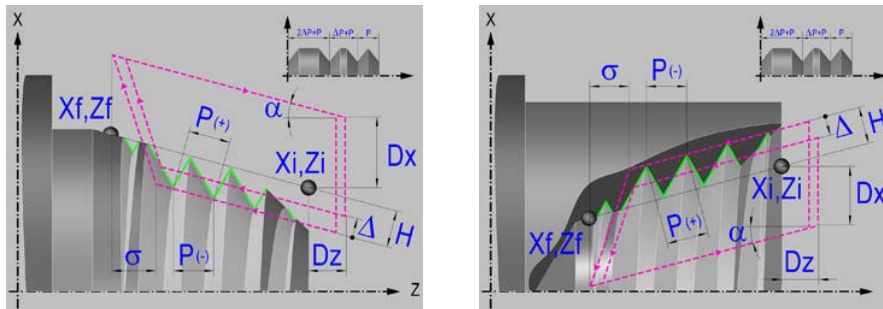


CNC 8060
CNC 8065

(REF: 1502)

4.2 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 (Xf, Zf).



Angle and length of the thread (α , ΔZ).



Angle of the thread and end coordinate (α , Zf).

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 .

α Thread angle.

Δz Thread length.

The X axis coordinates must be programmed in the active units, radius or diameter. 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.

4.

CANNED CYCLES OF THE EDITOR. TAPPING.

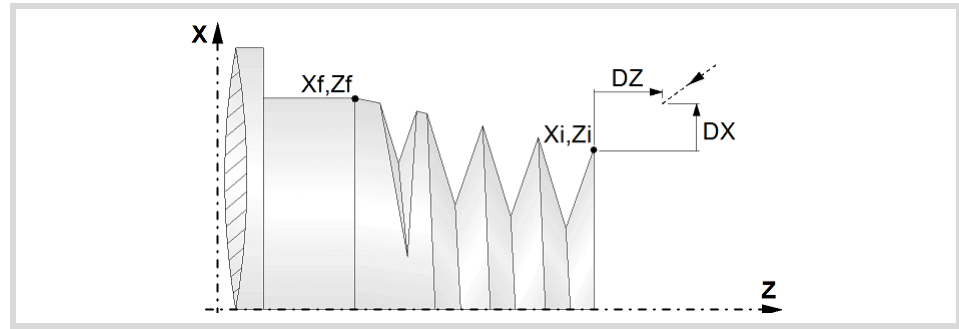
Taper threading cycle.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

4.

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.

ΔP 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.

ϕ 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.

Δ Maximum penetration step.



Constant depth of pass, with a smaller or same value as the maximum programmed.

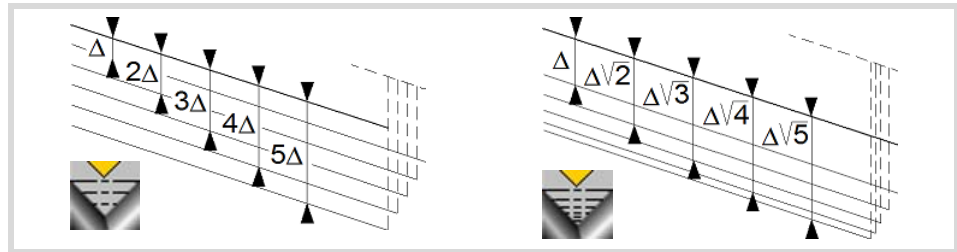
4.

CANNED CYCLES OF THE EDITOR. TAPPING.
Taper threading cycle.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)



4.

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 Δ , $\Delta\sqrt{2}$, $\Delta\sqrt{3}$, $\Delta\sqrt{4}$...

δ 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 σ 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 (α).



Penetration in zigzag, alternates between radial and starting flank. The CNC will ask for the cutter's penetration angle (α).



Penetration by final flank. The CNC will ask for the cutter's penetration angle (α).



Penetration in zigzag, alternates between radial and final flank. The CNC will ask for the cutter's penetration angle (α).

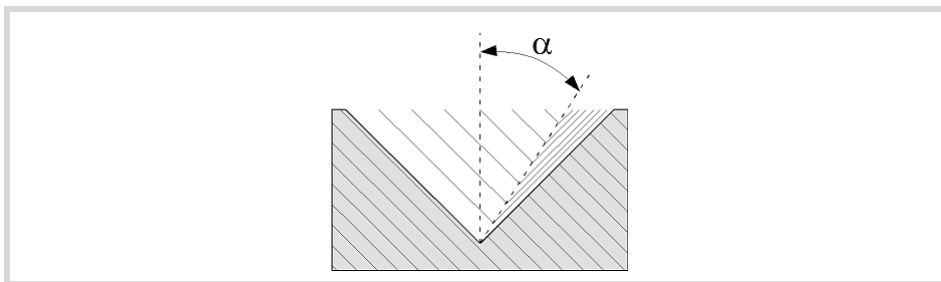


Penetration in zigzag, alternates between starting and final flank. The CNC will ask for the cutter's penetration angle (α) (leadscrew threading).

α 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.

4.

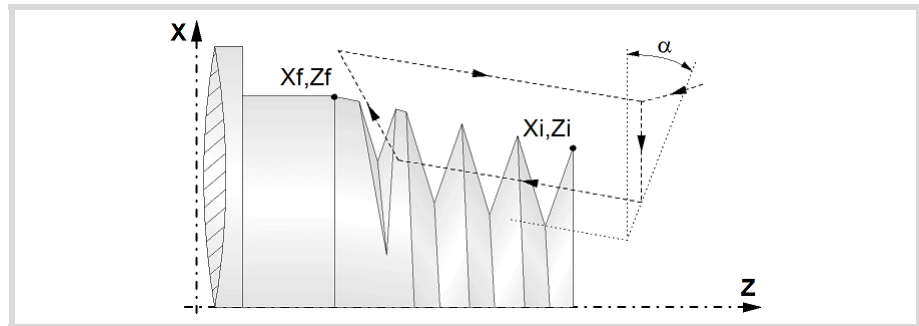
CANNED CYCLES OF THE EDITOR. TAPPING.

Taper threading cycle.

4.2.1 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 (α). 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).

4.

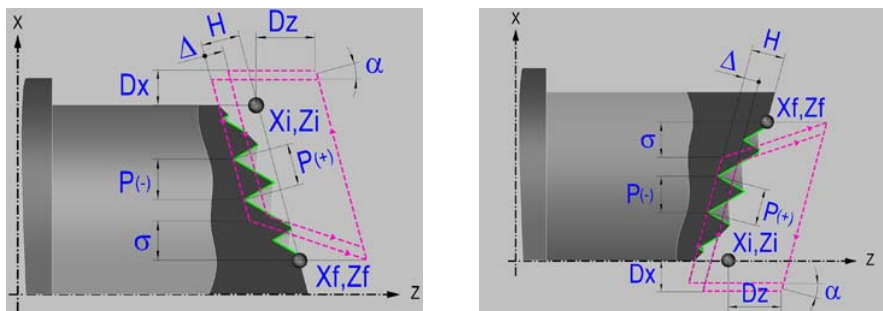
CANNED CYCLES OF THE EDITOR. TAPPING.
Taper threading cycle.



CNC 8060
CNC 8065

(REF: 1502)

4.3 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 .

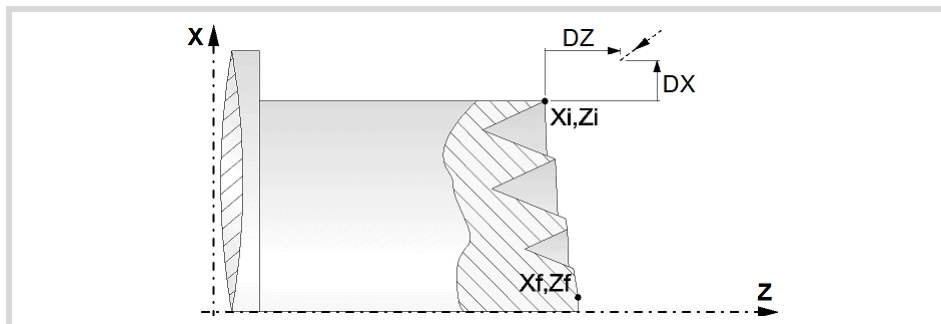
The X axis coordinates must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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.

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.

4.

CANNED CYCLES OF THE EDITOR. TAPPING.
Face threading cycle.



CNC 8060
CNC 8065

(REF: 1502)

4.

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.

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.**

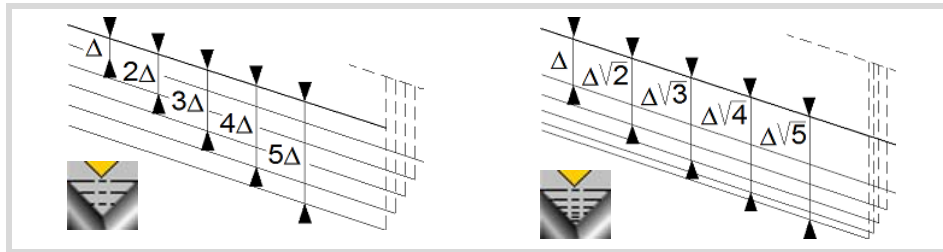
S Spindle speed.

Depth of pass.

Δ Maximum penetration step.



Constant depth of pass, with a smaller or same value as the maximum programmed.



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 Δ , $\Delta\sqrt{2}$, $\Delta\sqrt{3}$, $\Delta\sqrt{4}$...

δ 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 σ 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 (α).



Penetration in zigzag, alternates between radial and starting flank. The CNC will ask for the cutter's penetration angle (α).



Penetration by final flank. The CNC will ask for the cutter's penetration angle (α).



Penetration in zigzag, alternates between radial and final flank. The CNC will ask for the cutter's penetration angle (α).



Penetration in zigzag, alternates between starting and final flank. The CNC will ask for the cutter's penetration angle (α) (leadscrew threading).

α 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.

4.

CANNED CYCLES OF THE EDITOR. TAPPING.
Face threading cycle.



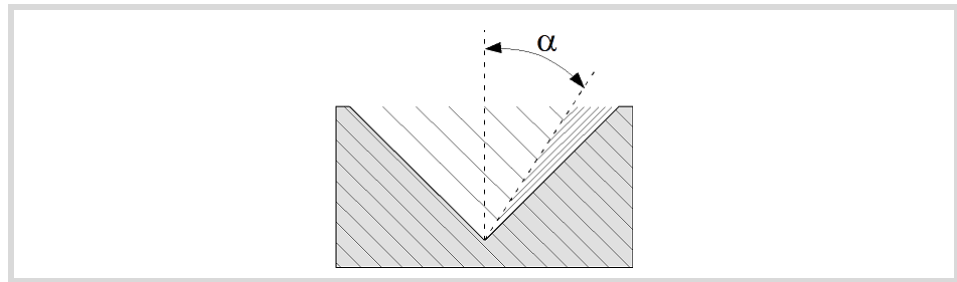
CNC 8060
CNC 8065

(REF: 1502)

4.

CANNED CYCLES OF THE EDITOR. TAPPING.

Face threading cycle.

**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.



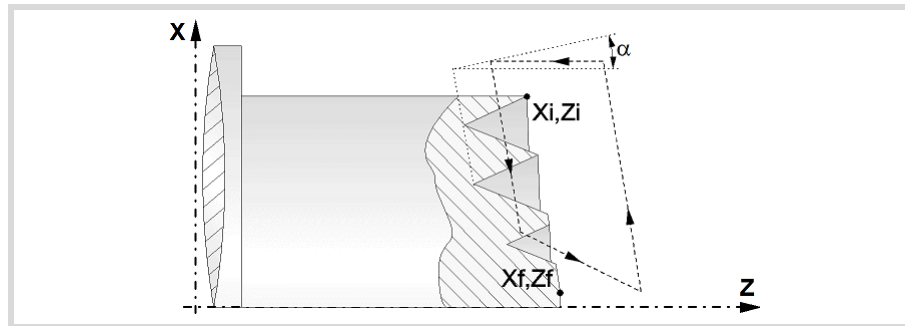
Deactivating the execution of M functions before machining.

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

4.3.1 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 (α). 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).

4.

CANNED CYCLES OF THE EDITOR. TAPPING.

Face threading cycle.

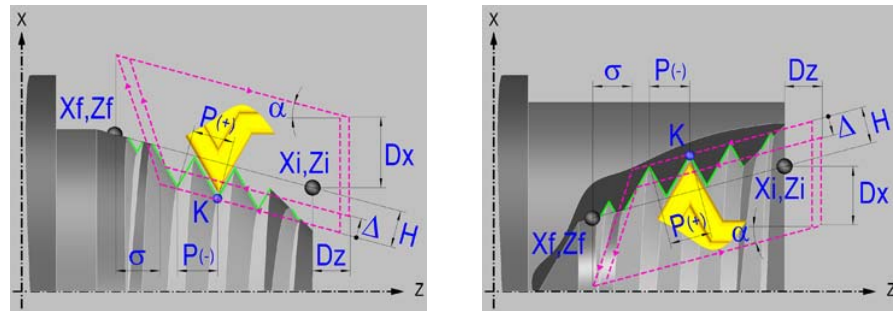
FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

4.4 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 (α, 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 .

α Thread angle.

ΔZ Thread length.

The X axis coordinates must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

4.

CANNED CYCLES OF THE EDITOR. TAPPING.
Thread repair cycle.



CNC 8060
CNC 8065

(REF: 1502)

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.

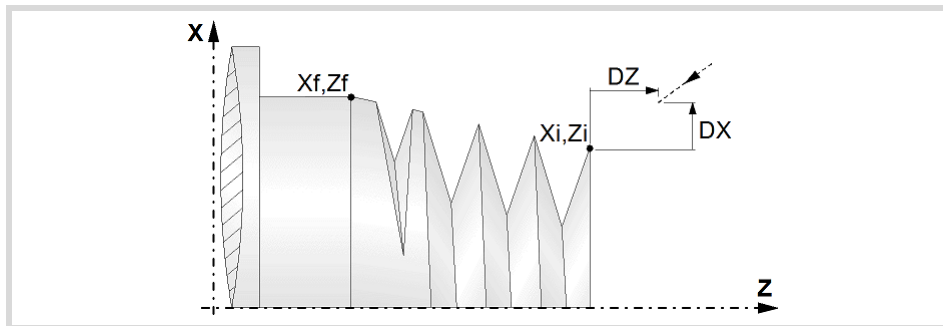
In partial thread repair, the starting point and the final (end) point correspond to the segment to be machined. The entry to the thread (start) is calculated automatically.

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.

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.

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.

4.

CANNED CYCLES OF THE EDITOR. TAPPING.
Thread repair cycle.



CNC 8060
CNC 8065

(REF: 1502)

4.

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).

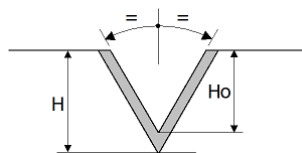
- If P-H (free-pitch thread) is selected, the pitch and the depth of the thread are selected directly by the user.
- When choosing a standard thread, the user only selects the diameter of the metrics, the pitch and the depth of that thread are calculated automatically. The standard threads are single 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).

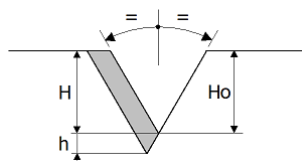
On P-H threads (free-pitch threads), first one must choose between defining the pitch of the thread or the number of threads per inch.

- | | |
|---|--|
| 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. <ul style="list-style-type: none"> • $P > 0$; 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.

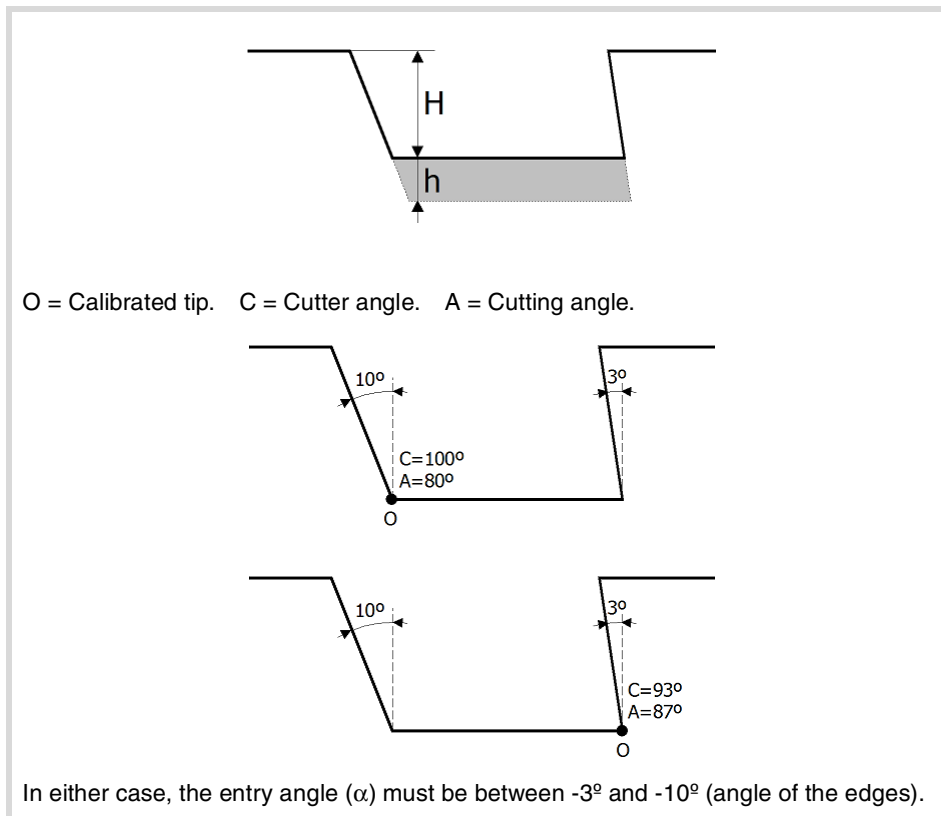


When programming a depth (H) greater than the initial one (H_0), the cycle increases the depth of the thread ignoring the existing flanks.



When programming the original depth ($H=H_0$) plus an increment (h), the cycle increases the depth but maintaining the entry (start) flank.

- 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.



4.

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.

ϕ 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.



CNC 8060
CNC 8065

(REF: 1502)

4.

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.**

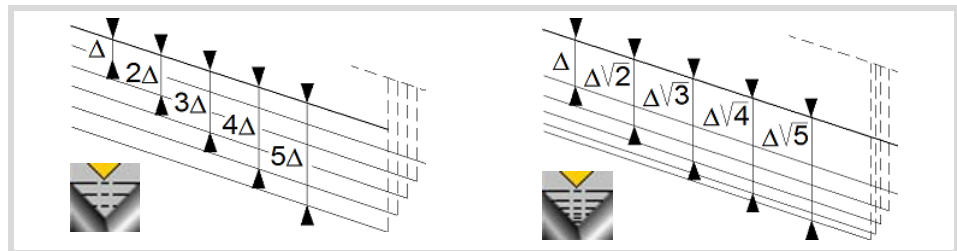
S Spindle speed.

Depth of pass.

Δ Maximum penetration step.



Constant depth of pass, with a smaller or same value as the maximum programmed.



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 $\Delta, \Delta\sqrt{2}, \Delta\sqrt{3}, \Delta\sqrt{4}...$

δ 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 σ 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 (α).



Penetration in zigzag, alternates between radial and starting flank. The CNC will ask for the cutter's penetration angle (α).



Penetration by final flank. The CNC will ask for the cutter's penetration angle (α).



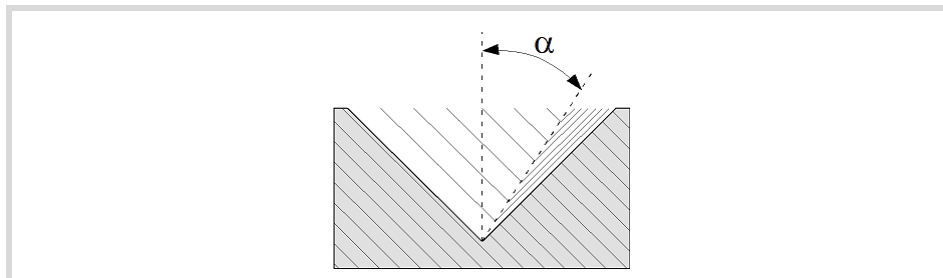
Penetration in zigzag, alternates between radial and final flank. The CNC will ask for the cutter's penetration angle (α).



Penetration in zigzag, alternates between starting and final flank. The CNC will ask for the cutter's penetration angle (α) (leadscrew threading).

α 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.

4.

CANNED CYCLES OF THE EDITOR. TAPPING.

Thread repair cycle.

FAGOR 

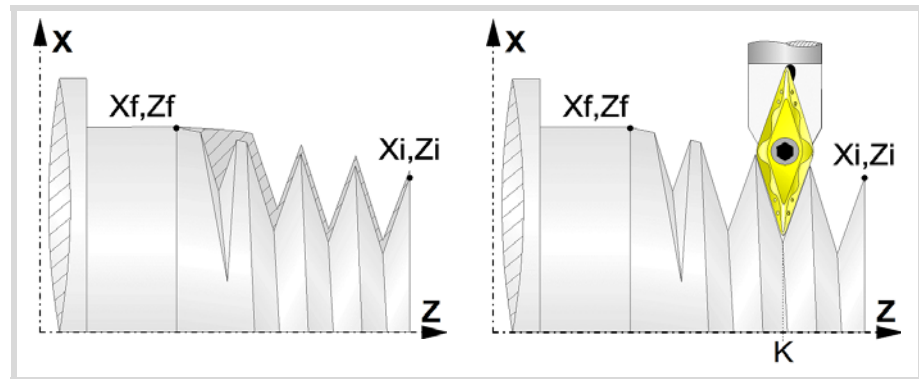
**CNC 8060
CNC 8065**

(REF: 1502)

4.4.1 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.



4.

CANNED CYCLES OF THE EDITOR. TAPPING.
Thread repair cycle.

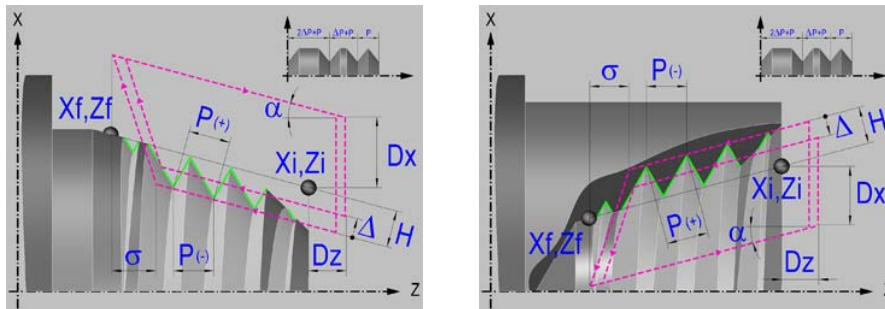


CNC 8060
CNC 8065

(REF: 1502)

4.5 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 (Xf, Zf).



Angle and length of the thread (α , ΔZ).



Angle of the thread and end coordinate (α , Zf).

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 .

α Thread angle.

Δz Thread length.

The X axis coordinates must be programmed in the active units, radius or diameter. 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.

4.

CANNED CYCLES OF THE EDITOR. TAPPING.

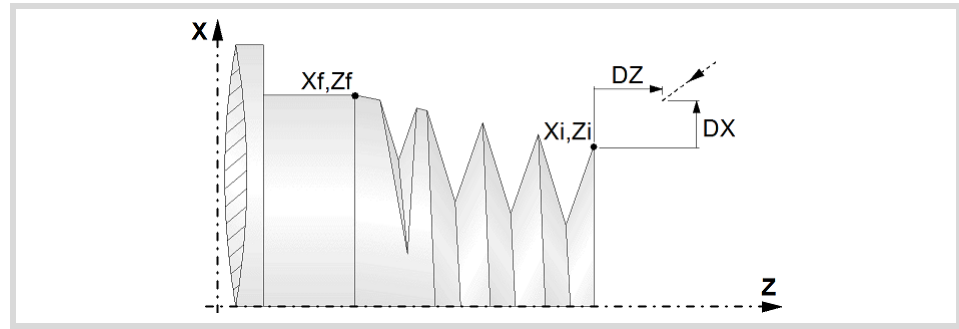
Multi-entry threading cycle.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

4.

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.

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.

ΔP 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.

φ 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.

Δ Maximum penetration step.

4.

CANNED CYCLES OF THE EDITOR. TAPPING.

Multi-entry threading cycle.



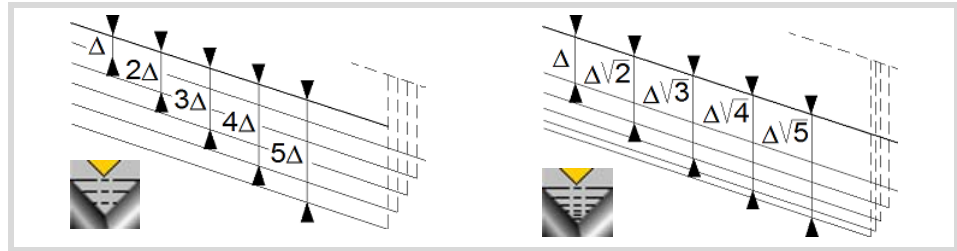
CNC 8060
CNC 8065

(REF: 1502)

4.



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 $\Delta, \Delta\sqrt{2}, \Delta\sqrt{3}, \Delta\sqrt{4}...$

δ 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 σ 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 (α).



Penetration in zigzag, alternates between radial and starting flank. The CNC will ask for the cutter's penetration angle (α).



Penetration by final flank. The CNC will ask for the cutter's penetration angle (α).



Penetration in zigzag, alternates between radial and final flank. The CNC will ask for the cutter's penetration angle (α).

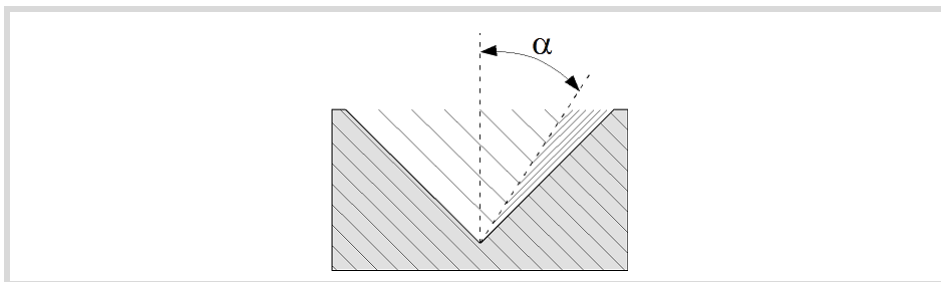


Penetration in zigzag, alternates between starting and final flank. The CNC will ask for the cutter's penetration angle (α) (leadscrew threading).

α 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.

4.

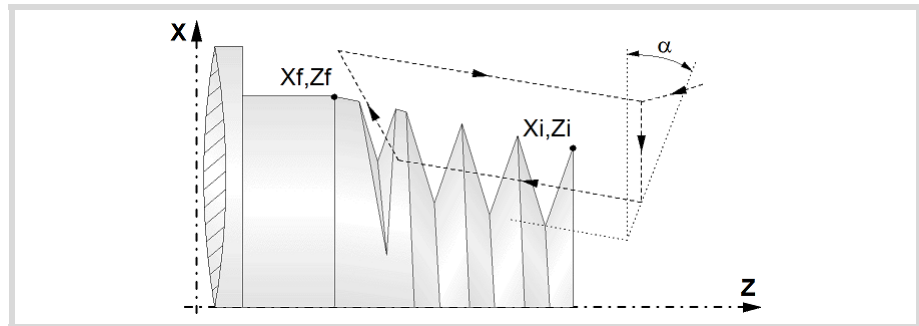
CANNED CYCLES OF THE EDITOR. TAPPING.

Multi-entry threading cycle.

4.5.1 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 (α). 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).

4.

CANNED CYCLES OF THE EDITOR. TAPPING.
Multi-entry threading cycle.



CNC 8060
CNC 8065

(REF: 1502)

CANNED CYCLES OF THE EDITOR. GROOVING.

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.



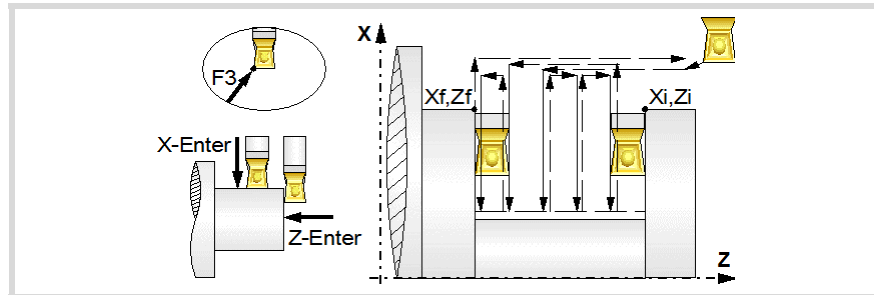
- Simple longitudinal grooving.
- Simple face grooving.
- Inclined (angled) longitudinal grooving.
- Inclined (angled) face grooving.
- Cut-off.

5.1 Calibrate the grooving tool (location code, shape).

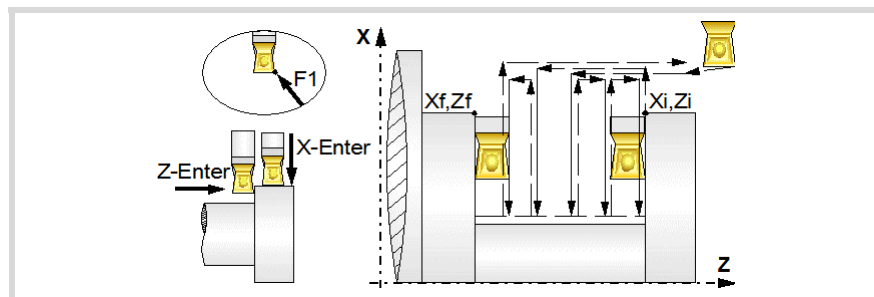
5.1.1 Outside longitudinal grooving.

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:

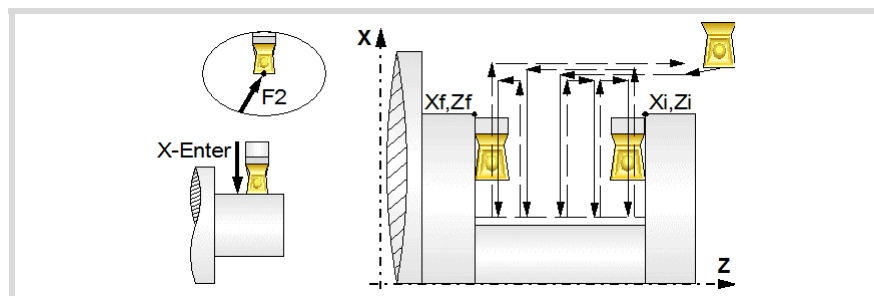
- The lower left-hand corner of the cutter is calibrated. Location code F3.



- 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.



5.

CANNED CYCLES OF THE EDITOR. GROOVING.
Calibrate the grooving tool (location code, shape).

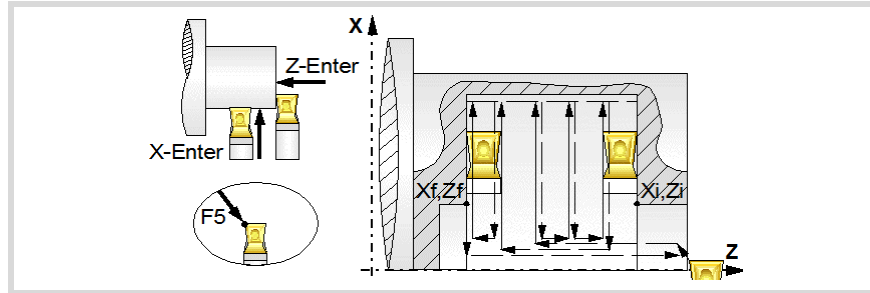


CNC 8060
CNC 8065

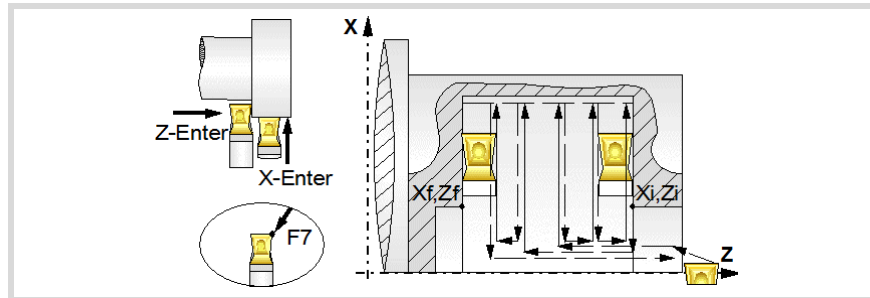
(REF: 1502)

5.1.2 Inside longitudinal grooving.

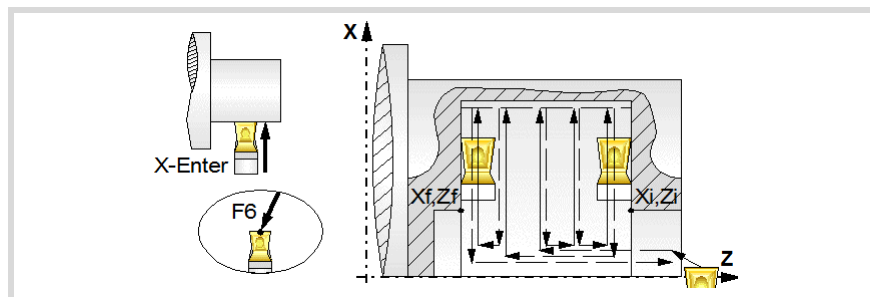
- The upper left-hand corner of the cutter is calibrated. Location code F5.



- 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.



5.

CANNED CYCLES OF THE EDITOR. GROOVING.
 Calibrate the grooving tool (location code, shape).



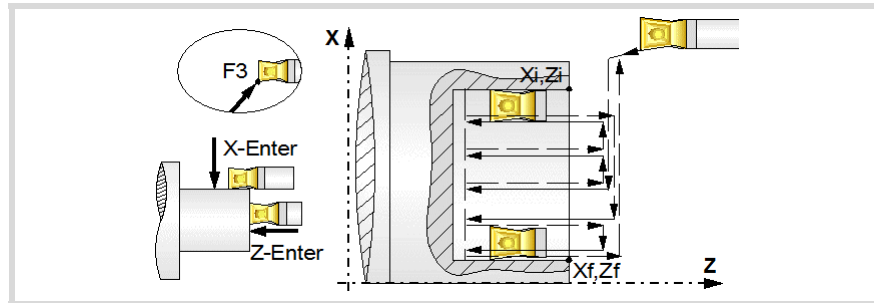
CNC 8060
 CNC 8065

(REF: 1502)

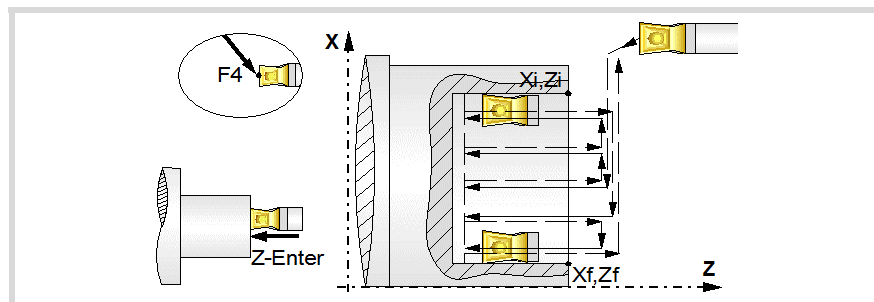
5.1.3 Face grooving.

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:

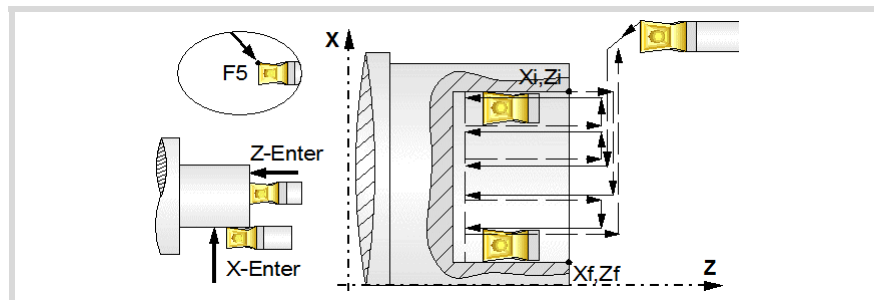
- The lower left-hand corner of the cutter is calibrated. Location code F3.



- 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.



5.

CANNED CYCLES OF THE EDITOR. GROOVING.
Calibrate the grooving tool (location code, shape).

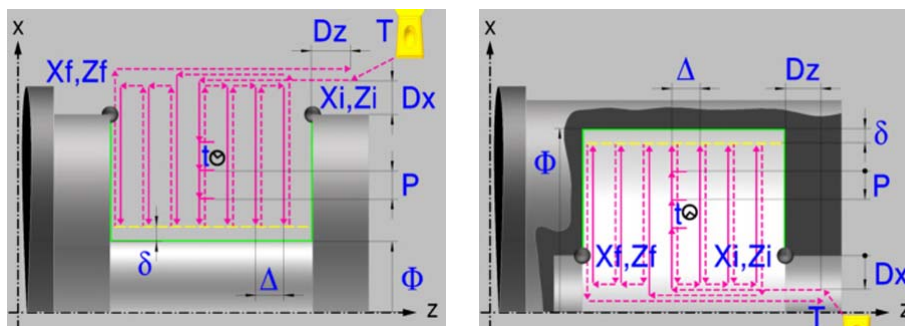


CNC 8060
CNC 8065

(REF: 1502)

5.2 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 ["5.1 Calibrate the grooving tool \(location code, shape\)."](#) on page 186.

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 .

The X axis coordinates must be programmed in the active units, radius or diameter. 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.

5.

CANNED CYCLES OF THE EDITOR. GROOVING.

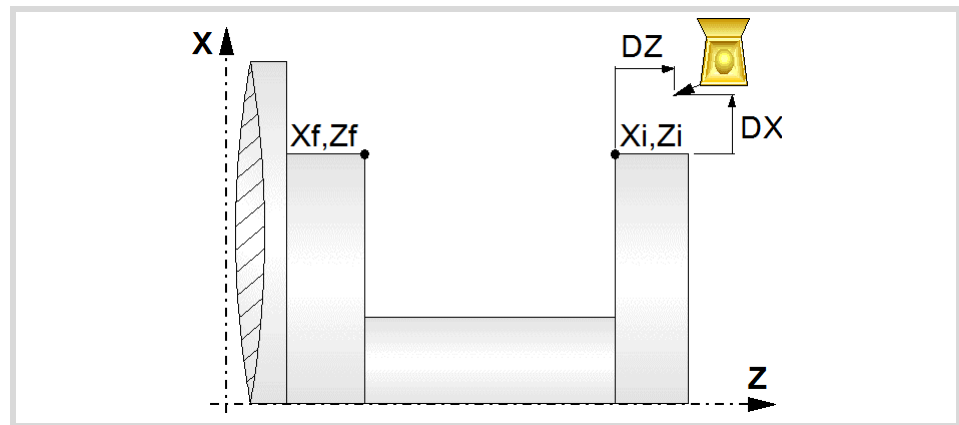
Simple longitudinal grooving cycle.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

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



Final diameter.

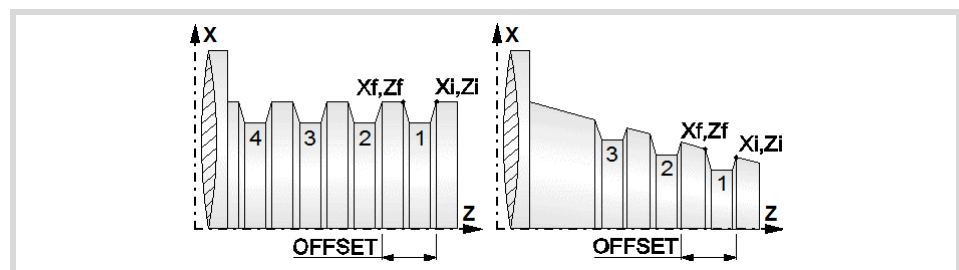
Φ Final diameter.

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 (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.

5.

CANNED CYCLES OF THE EDITOR. GROOVING.
Simple longitudinal grooving cycle.



CNC 8060
CNC 8065

(REF: 1502)

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.

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.

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.

5.

CANNED CYCLES OF THE EDITOR. GROOVING.

Simple longitudinal grooving cycle.

FAGOR CNC 8060
CNC 8065

(REF: 1502)

5.

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 (δx , δz).

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.

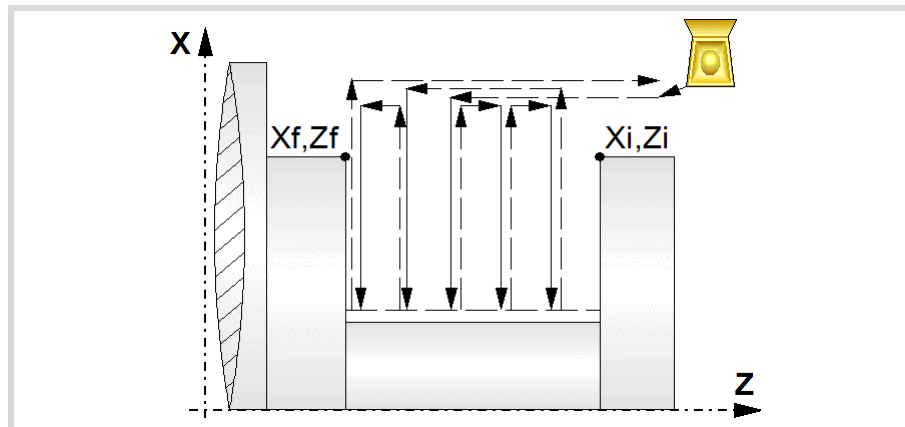
5.2.1 Basic operation

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.

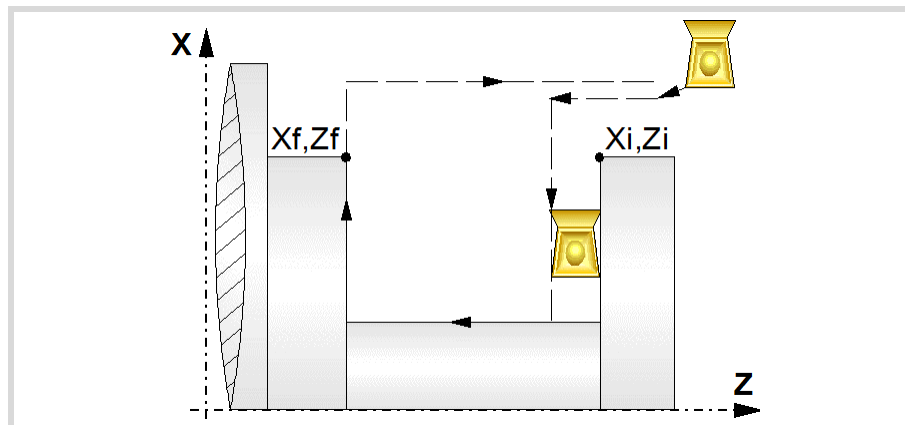
The first penetration pass may be done in two ways; either continuously or in intervals depending on the programmed pass and with a dwell at the bottom of each pass.



- 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).

5.

CANNED CYCLES OF THE EDITOR. GROOVING.

Simple longitudinal grooving cycle.

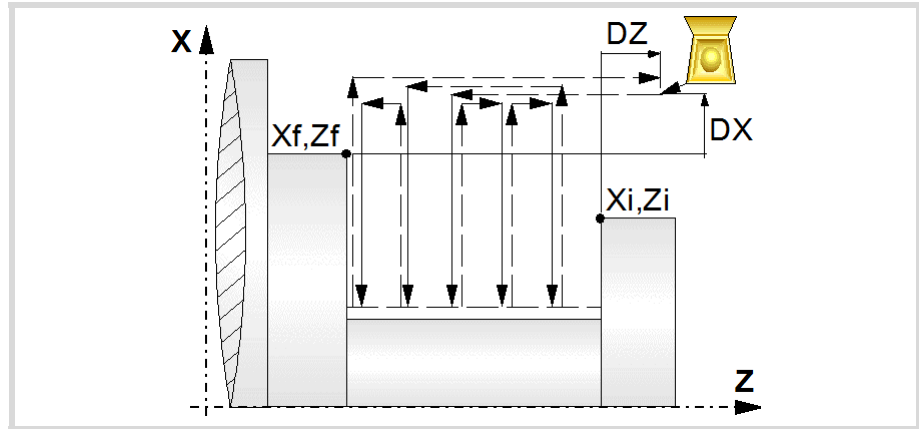
FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

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.



5.

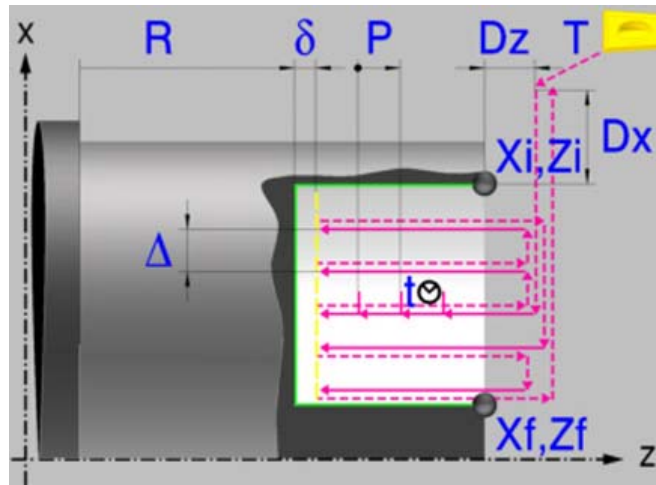
CANNED CYCLES OF THE EDITOR. GROOVING.
Simple longitudinal grooving cycle.



CNC 8060
CNC 8065

(REF: 1502)

5.3 Simple face grooving cycle



5.

CANNED CYCLES OF THE EDITOR. GROOVING.
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 "5.1 Calibrate the grooving tool (location code, shape)." on page 186.

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 .

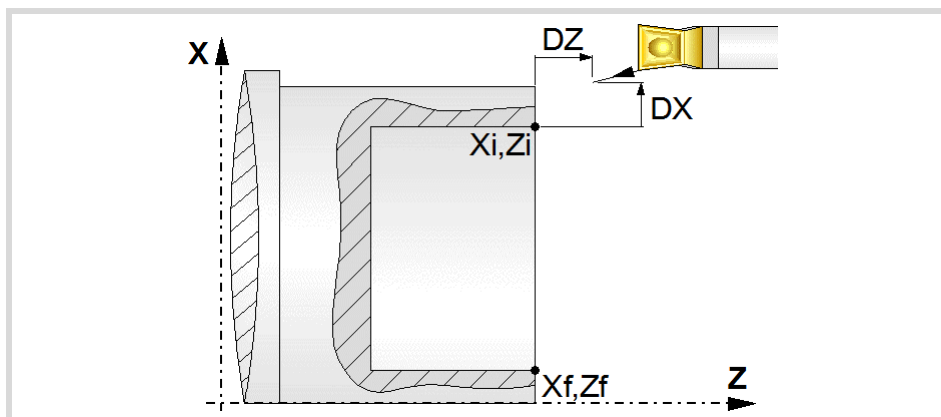
The X axis coordinates must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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.



CNC 8060
CNC 8065

(REF: 1502)

Coordinate of the bottom of the groove.

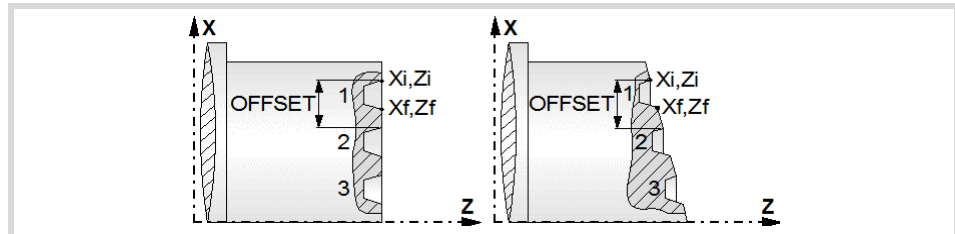
R Coordinate of the bottom of the groove

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.

5.

CANNED CYCLES OF THE EDITOR. GROOVING.
Simple face grooving cycle



FAGOR 

CNC 8060
CNC 8065




(REF: 1502)

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.

- 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.

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 (δx , δz).

5.

CANNED CYCLES OF THE EDITOR. GROOVING.

Simple face grooving cycle

FAGOR CNC 8060
CNC 8065

(REF: 1502)

5.

CANNED CYCLES OF THE EDITOR. GROOVING.

Simple face grooving cycle

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.

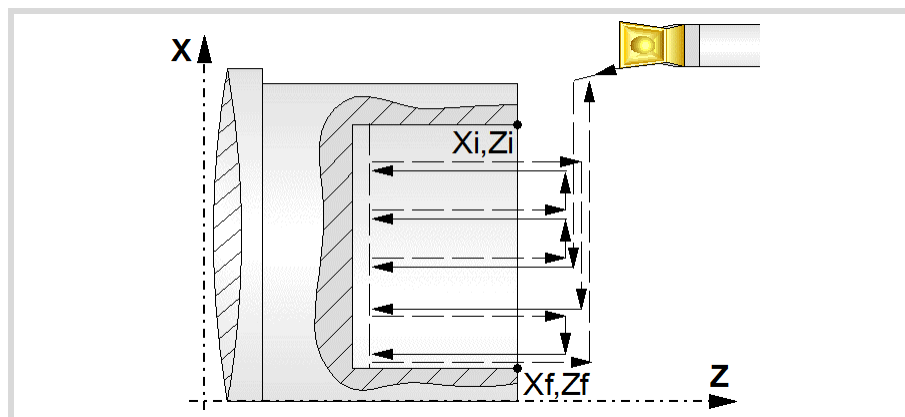
5.3.1 Basic operation

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.

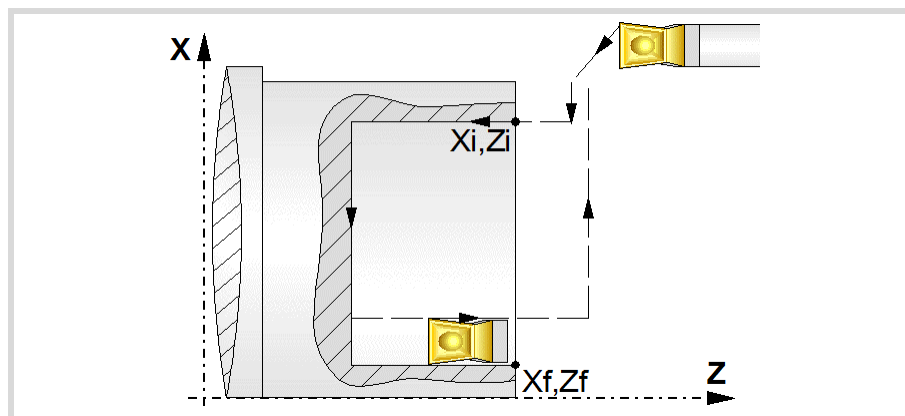
The first penetration pass may be done in two ways; either continuously or in intervals depending on the programmed pass and with a dwell at the bottom of each pass.



- 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).

5.

CANNED CYCLES OF THE EDITOR. GROOVING.
Simple face grooving cycle

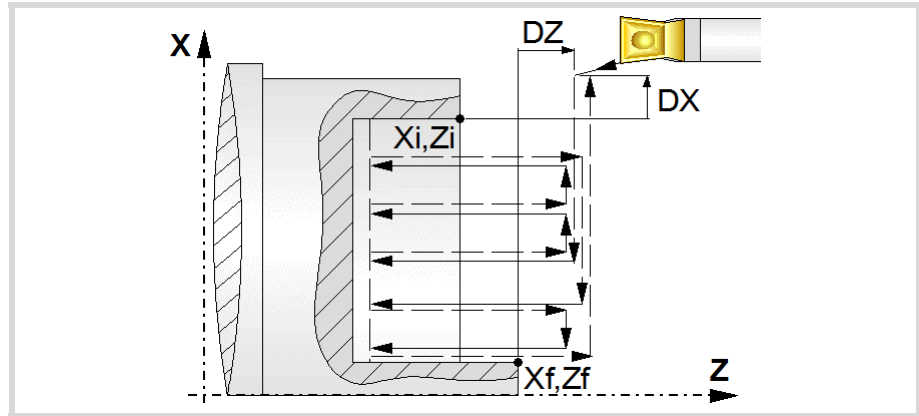
FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

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.



5.

CANNED CYCLES OF THE EDITOR. GROOVING.
Simple face grooving cycle

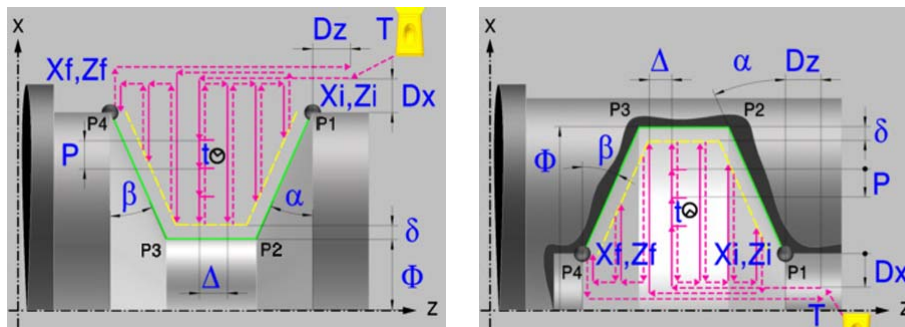


CNC 8060
CNC 8065

(REF: 1502)

5.4 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 "5.1 Calibrate the grooving tool (location code, shape)." on page 186.

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 .

The X axis coordinates must be programmed in the active units, radius or diameter. 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.

5.

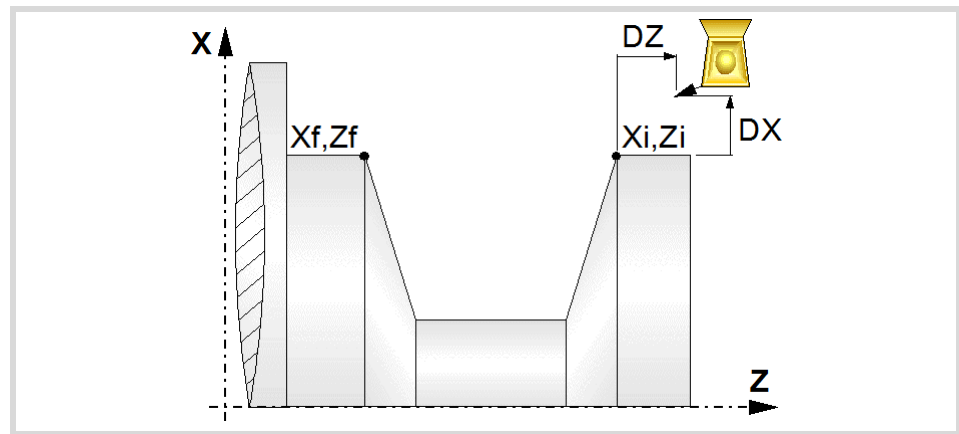
CANNED CYCLES OF THE EDITOR. GROOVING.
Inclined longitudinal grooving cycle.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

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



Final diameter.

Φ 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.

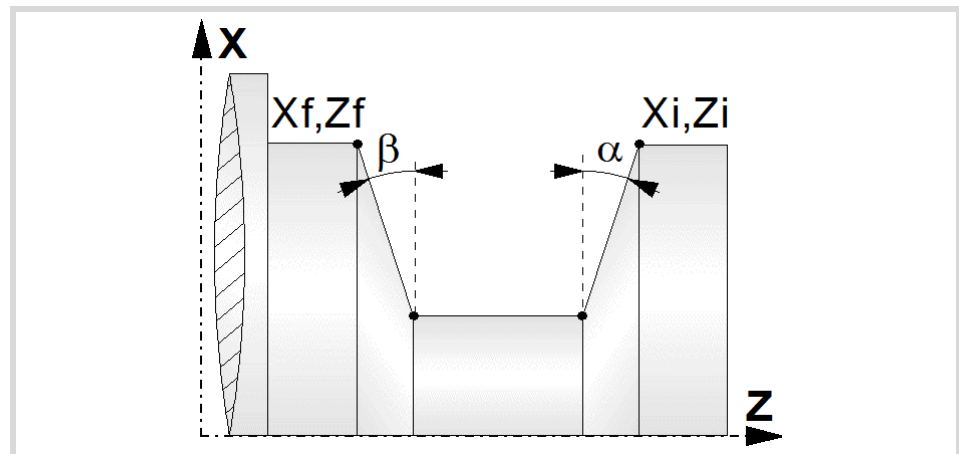
r Rounding radius of the pocket corners.

c Size of the chamfer of the pocket corners.

Inclination angles.

α Inclination angle of the starting wall.

β 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.

5.

CANNED CYCLES OF THE EDITOR. GROOVING.

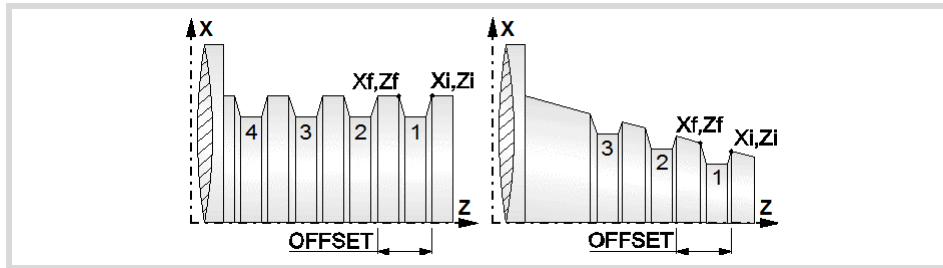
Inclined longitudinal grooving cycle.



CNC 8060
CNC 8065

(REF: 1502)

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.



5.

CANNED CYCLES OF THE EDITOR. GROOVING.
Inclined longitudinal grooving 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.

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.

5.

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.

- 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.

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 (δx , δz).

Finishing stock.

δ Finishing stock.

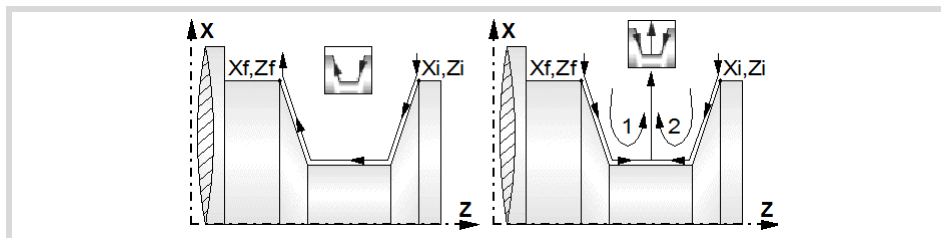
Type of finishing pass.



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.

5.

CANNED CYCLES OF THE EDITOR. GROOVING.

Inclined longitudinal grooving cycle.



CNC 8060
CNC 8065

(REF: 1502)

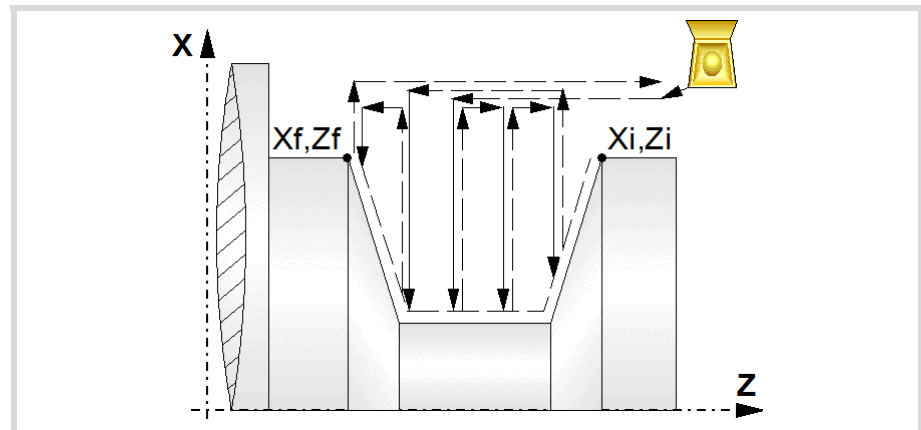
5.4.1 Basic operation

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.

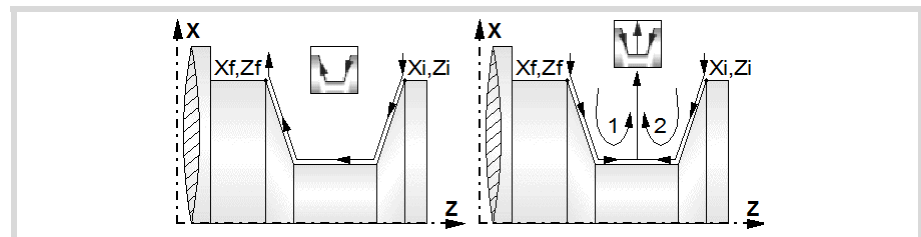
The first penetration pass may be done in two ways; either continuously or in intervals depending on the programmed pass and with a dwell at the bottom of each pass.



- 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).

5.

CANNED CYCLES OF THE EDITOR. GROOVING.
Inclined longitudinal grooving cycle.

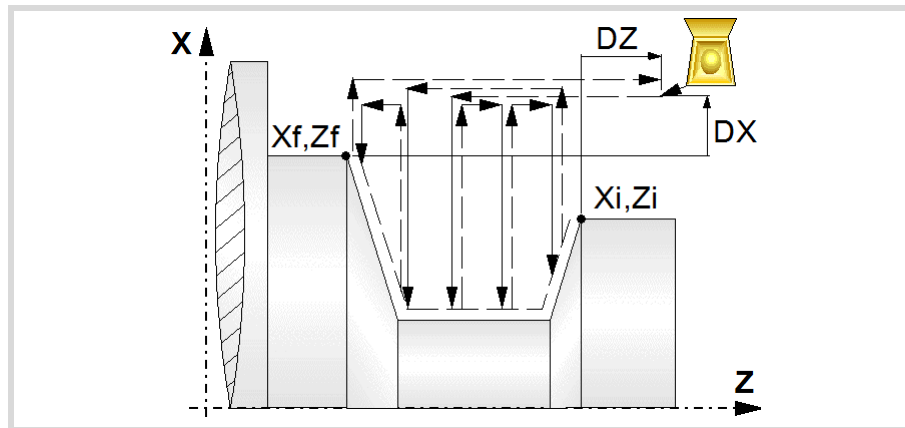


CNC 8060
CNC 8065

(REF: 1502)

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.

5.

CANNED CYCLES OF THE EDITOR. GROOVING.

Inclined longitudinal grooving cycle.

FAGOR 

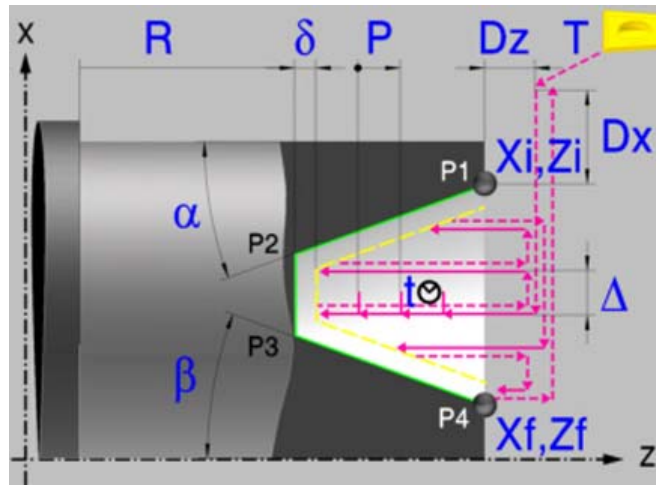
CNC 8060
CNC 8065

(REF: 1502)

5.5 Inclined face grooving cycle.

5.

CANNED CYCLES OF THE EDITOR. GROOVING.
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 "[5.1 Calibrate the grooving tool \(location code, shape\).](#)" on page 186.

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 .

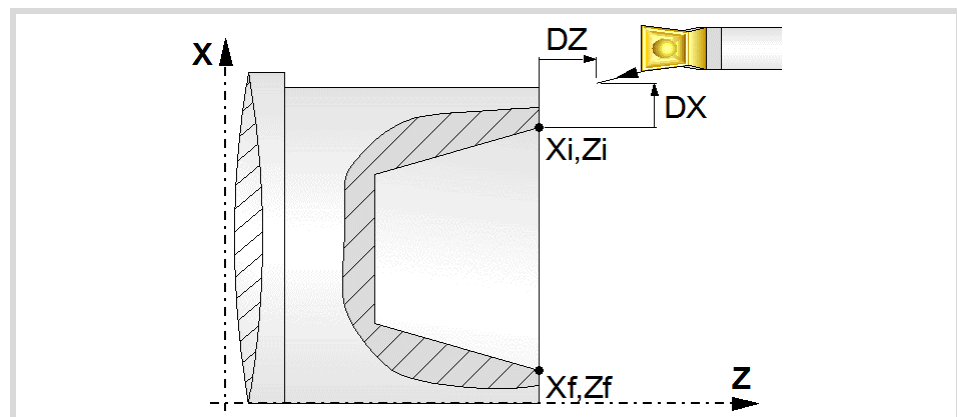
The X axis coordinates must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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.



CNC 8060
CNC 8065

(REF: 1502)

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.

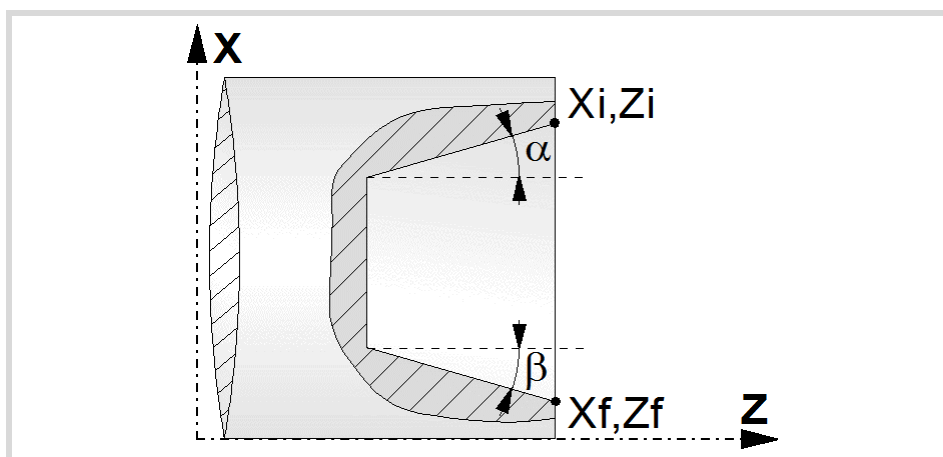
r Rounding radius of the pocket corners.

c Size of the chamfer of the pocket corners.

Inclination angles.

α Inclination angle of the starting wall.

β Inclination angle of the final wall.

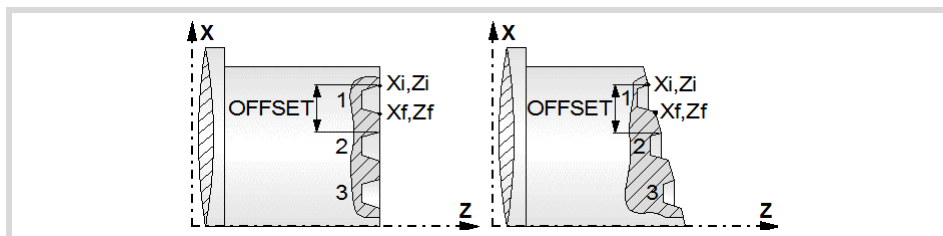


Groove repetition.

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.

5.

CANNED CYCLES OF THE EDITOR. GROOVING.

Inclined face grooving cycle.



CNC 8060
CNC 8065

(REF: 1502)

5.

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.

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.

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 (δx , δz).

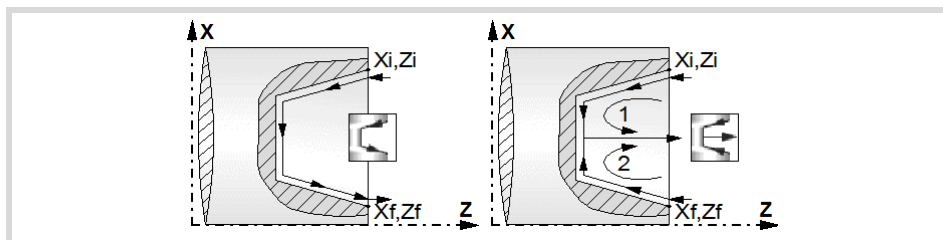
Finishing stock.

δ Finishing stock.

Type of finishing pass.

 Groove finishing following the profile.

 Descending finishing of the groove.



5.

CANNED CYCLES OF THE EDITOR. GROOVING.
Inclined face grooving cycle.



CNC 8060
CNC 8065

(REF: 1502)

5.

CANNED CYCLES OF THE EDITOR. GROOVING.

Inclined face grooving 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 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.

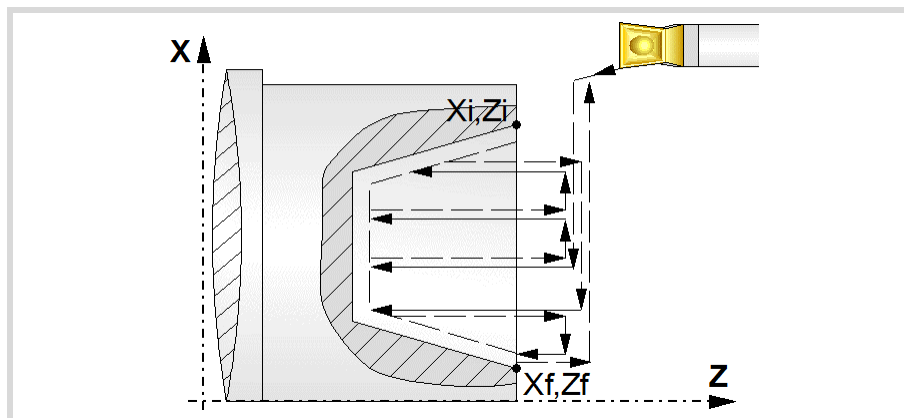
5.5.1 Basic operation

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.

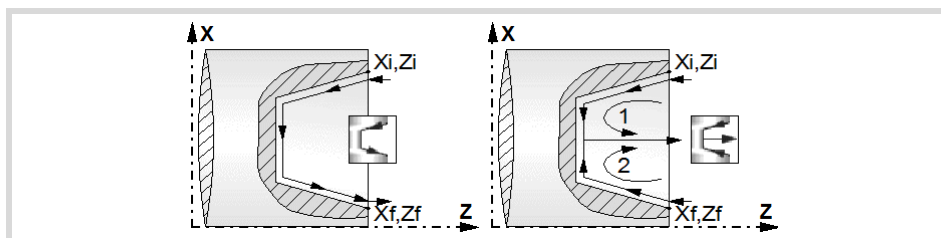
The first penetration pass may be done in two ways; either continuously or in intervals depending on the programmed pass and with a dwell at the bottom of each pass.



- 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).

5.

CANNED CYCLES OF THE EDITOR. GROOVING.

Inclined face grooving cycle.

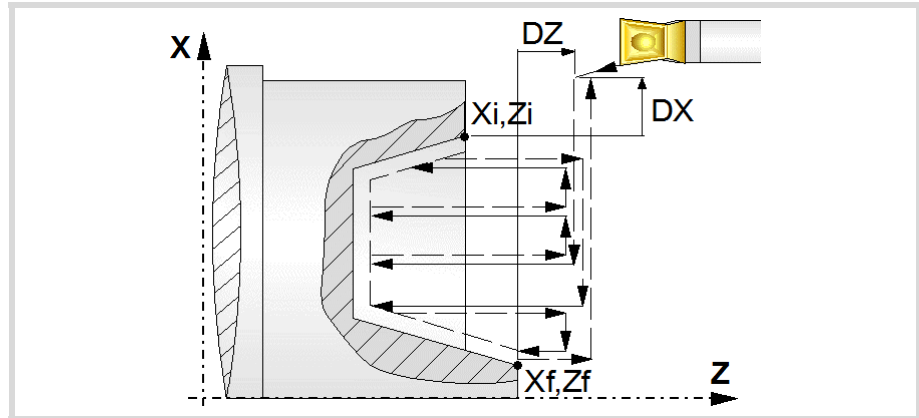
FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

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.

5.

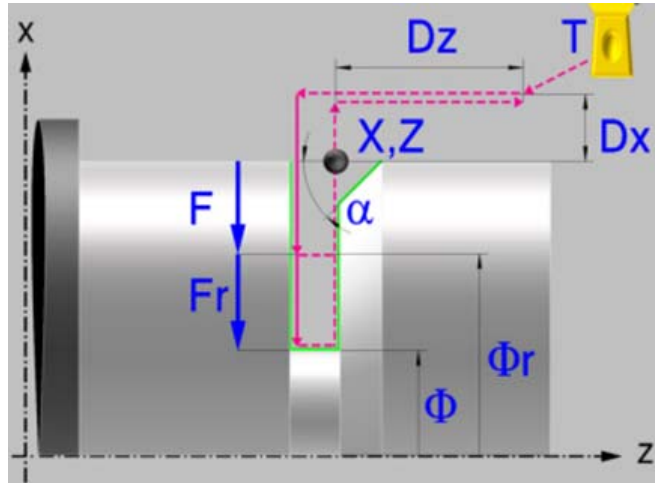
CANNED CYCLES OF THE EDITOR. GROOVING.
Inclined face grooving cycle.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

5.6 Cut-off cycle.



5.

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 "5.1 Calibrate the grooving tool (location code, shape)." on page 186.

Geometric parameters.

Coordinates of the profile's starting and final points.

X_i, Z_i Coordinates of the starting point

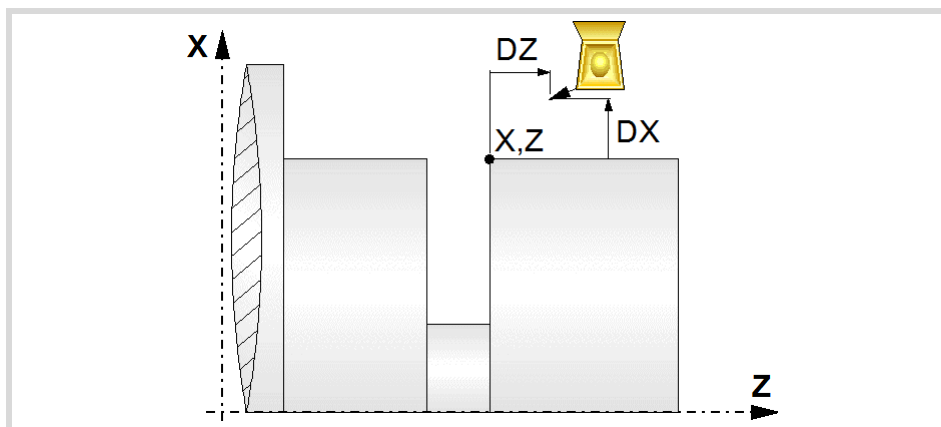
The X axis coordinate must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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 

CNC 8060
CNC 8065

(REF: 1502)

5.

CANNED CYCLES OF THE EDITOR. GROOVING. Cut-off cycle.

Final diameter.

Φ 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.

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 (Φr) 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.
 Φr 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.

5.

CANNED CYCLES OF THE EDITOR. GROOVING.
 Cut-off cycle.

FAGOR 

**CNC 8060
 CNC 8065**

(REF: 1502)

5.6.1 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 Φ .
- 5 The cycle begins machining at a feedrate F until reaching the diameter Φ_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).

5.

CANNED CYCLES OF THE EDITOR. GROOVING.

Cut-off cycle.



CNC 8060
CNC 8065

(REF: 1502)

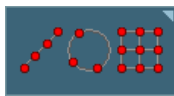
CANNED CYCLES OF THE EDITOR. PROFILES.

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.



- 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.



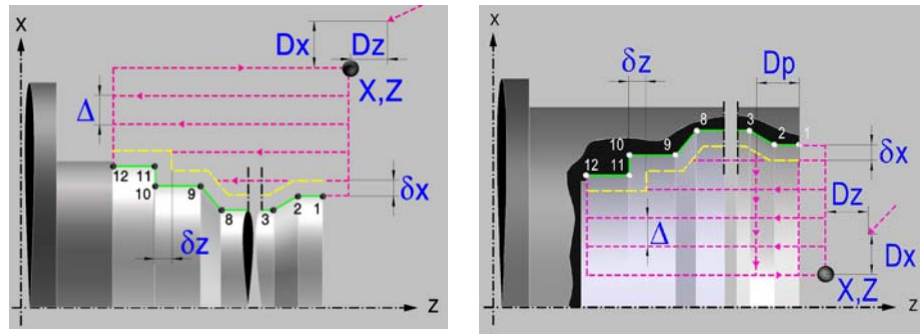
Multiple machining may be added to pocket cycles so the cycle is repeated in several points. The available multiple machining operations are the following. See chapter "[9 Canned cycles of the editor. Multiple positioning.](#)".

- 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.

6.1 Point-to-point turning cycle.

The cycle allows outside and inside machining.



Geometric parameters.

Inside or outside machining.



Outside machining.



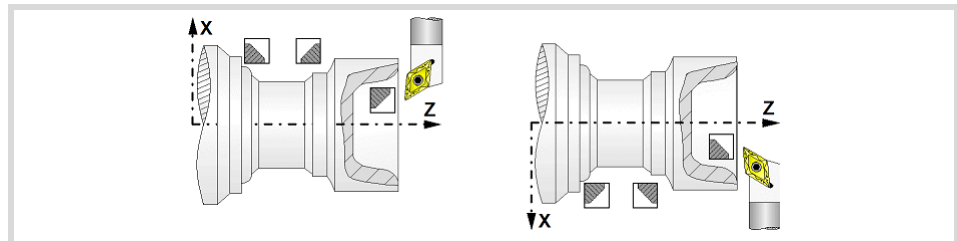
Inside machining.

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. The X axis coordinate must be programmed in the active units, radius or diameter.

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.

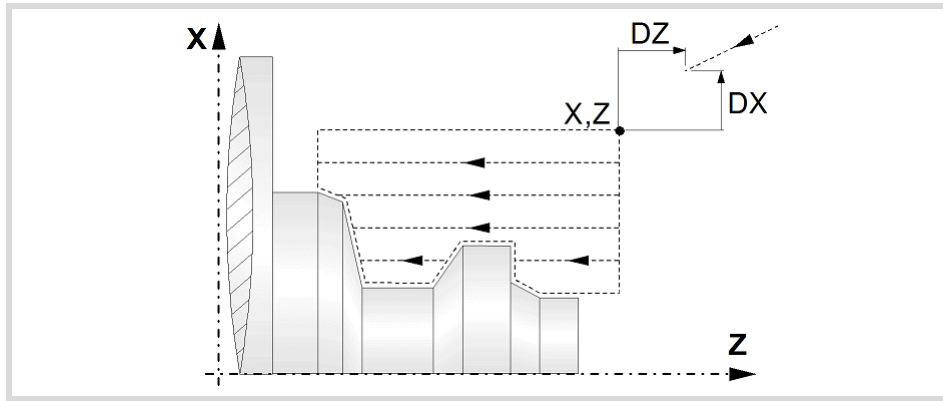
6.

CANNED CYCLES OF THE EDITOR. PROFILES.
Point-to-point turning cycle.



CNC 8060
CNC 8065

(REF: 1502)

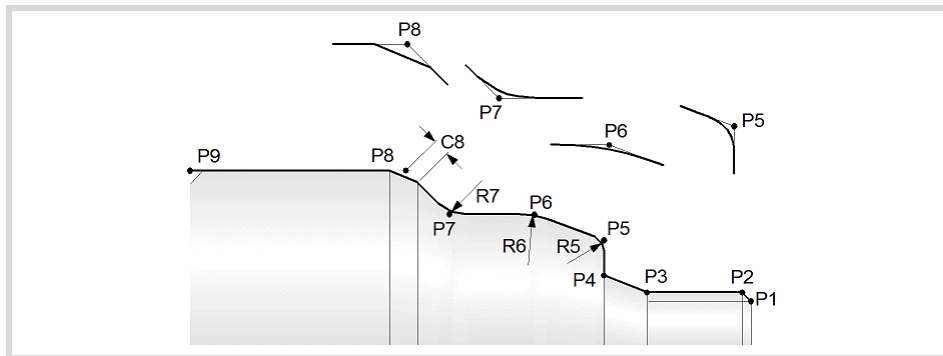


6.

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.



CNC 8060
CNC 8065

(REF: 1502)

6.

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.

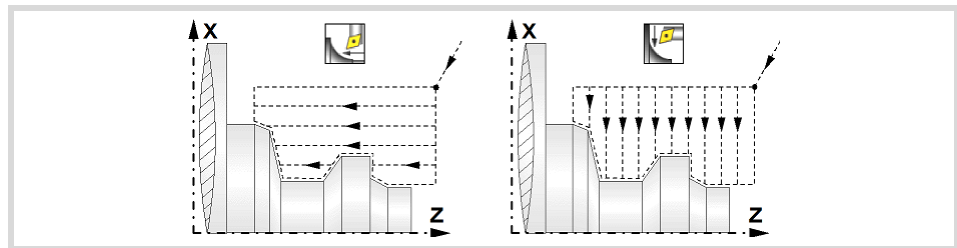
Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

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.

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.

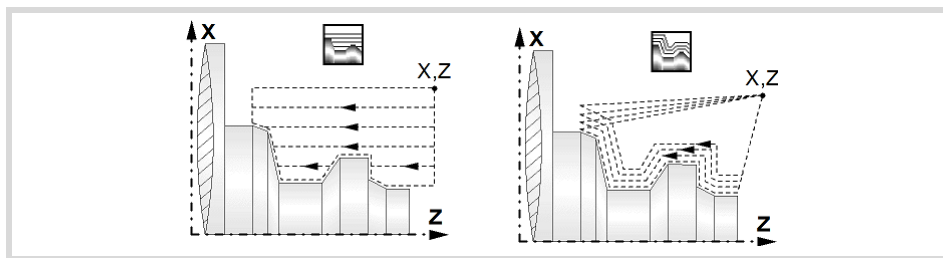


Paraxial machining. The tool penetration feedrate (Fp) must be defined for the roots (valleys).



Machining by pattern repeat (following the profile). The amount of material required to be removed from the origin part (ϵ) must be defined. That value is defined in radius.

- Fp Penetration feedrate in the roots for paraxial machining.
- ϵ 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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.

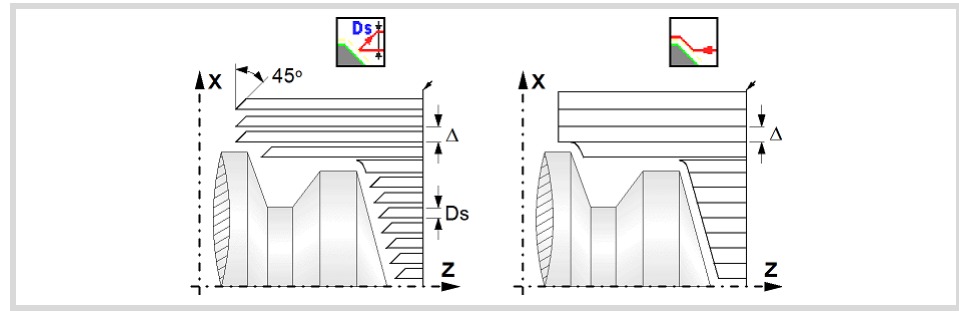
Point-to-point turning cycle.



CNC 8060
CNC 8065

(REF: 1502)

6.

**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.

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 (δ , δx , δz).

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.

δ Finishing stock depending on the cutter edge.

δx , δz 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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.

Point-to-point turning cycle.

6.1.1 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).

6.

CANNED CYCLES OF THE EDITOR. PROFILES.

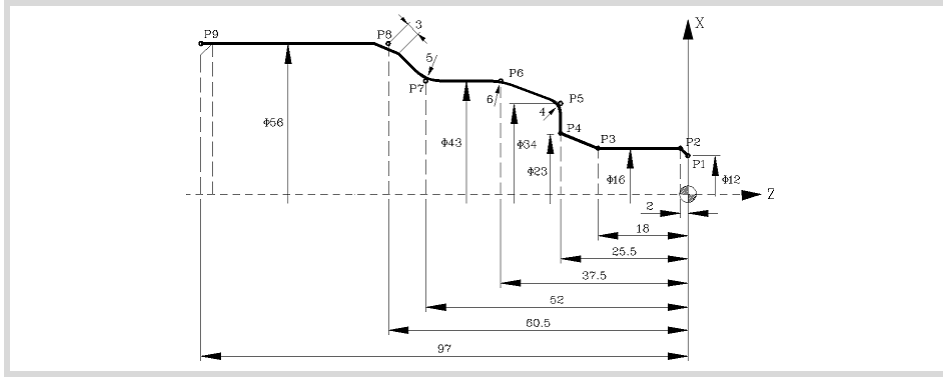
Point-to-point turning cycle.



CNC 8060
CNC 8065

(REF: 1502)

6.1.2 Programming example.



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.

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
P6	ABS	X	43.0000	Z	- 37.5000
P7	ABS	X	43.0000	Z	- 52.0000
P8	ABS	X	56.0000	Z	- 60.5000
P9	ABS	X	56.0000	Z	- 97.0000
P10	ABS	X	56.0000	Z	- 97.0000



R 4.0000
R 6.0000
R 5.0000
C 3.0000

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

6.

CANNED CYCLES OF THE EDITOR. PROFILES.

Point-to-point turning cycle.

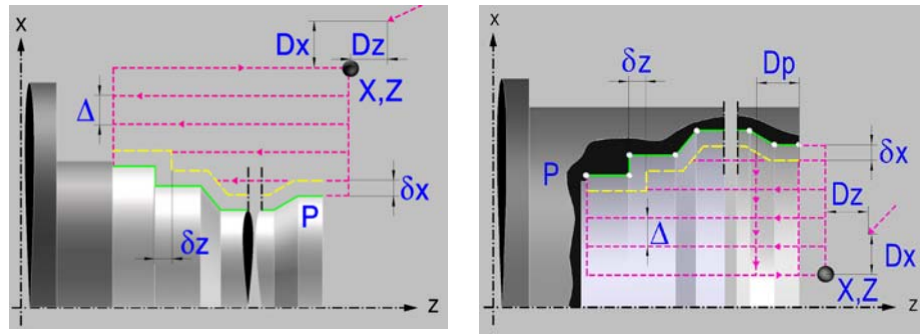
FAGOR

CNC 8060
CNC 8065

(REF: 1502)

6.2 Profile turning cycle.

The cycle allows outside and inside machining.



Geometric parameters.

Inside or outside machining.



Outside machining.



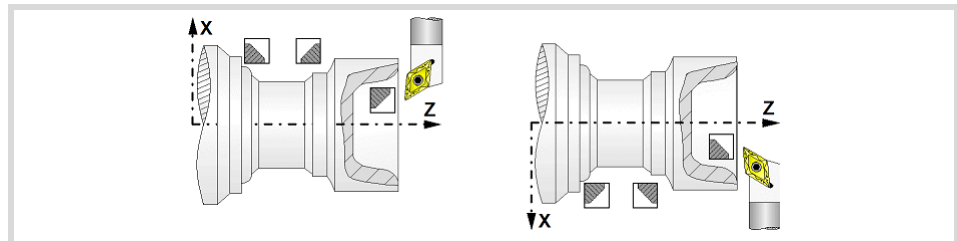
Inside machining.

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. The X axis coordinate must be programmed in the active units, radius or diameter.

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.

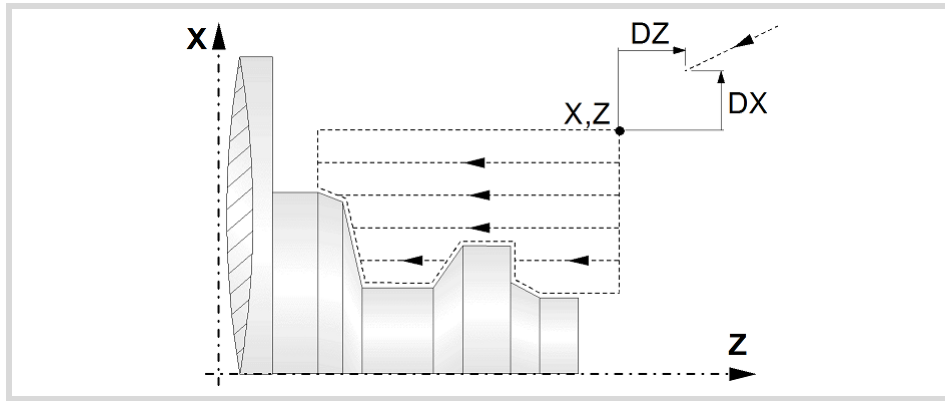
6.

CANNED CYCLES OF THE EDITOR. PROFILES.
Profile turning cycle.



CNC 8060
CNC 8065

(REF: 1502)



6.

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.

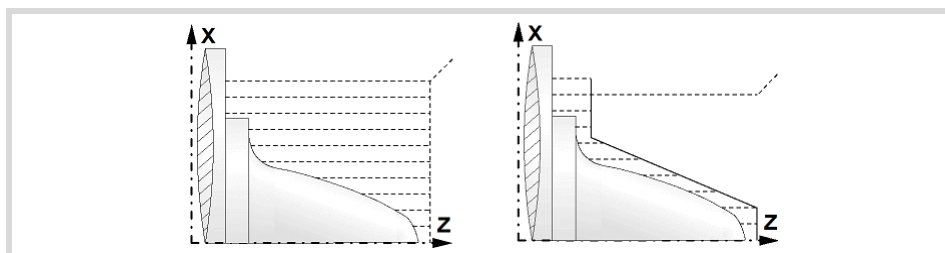
Modify an existing profile.

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

CNC 8060
CNC 8065

(REF: 1502)

6.

- 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.

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.

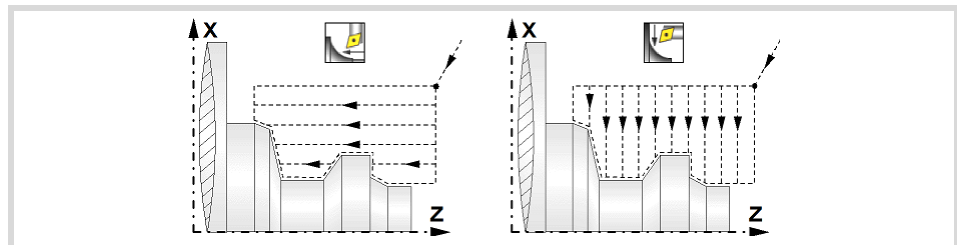
Machining direction.



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.

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.

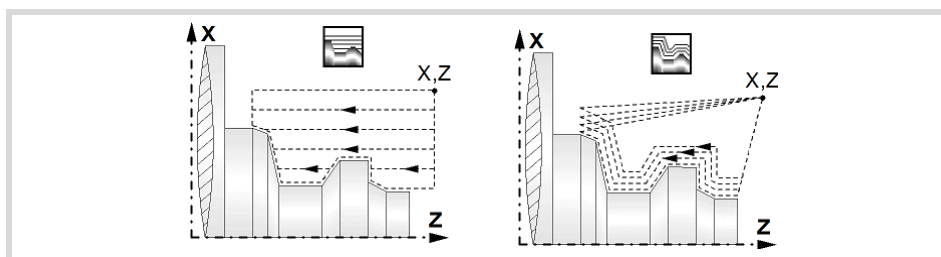


Paraxial machining. The tool penetration feedrate (Fp) must be defined for the roots (valleys).



Machining by pattern repeat (following the profile). The amount of material required to be removed from the origin part (ϵ) must be defined. That value is defined in radius.

- Fp Penetration feedrate in the roots for paraxial machining.
- ϵ 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.

6.

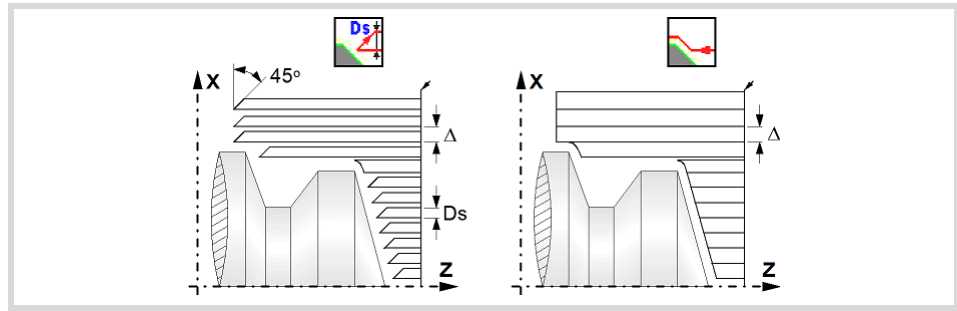
CANNED CYCLES OF THE EDITOR. PROFILES.
Profile turning cycle.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

6.

**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.

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 (δ , δx , δz).

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.

δ Finishing stock depending on the cutter edge.

δx , δz 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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.

Profile turning cycle.



CNC 8060
CNC 8065

(REF: 1502)

6.2.1 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).

6.

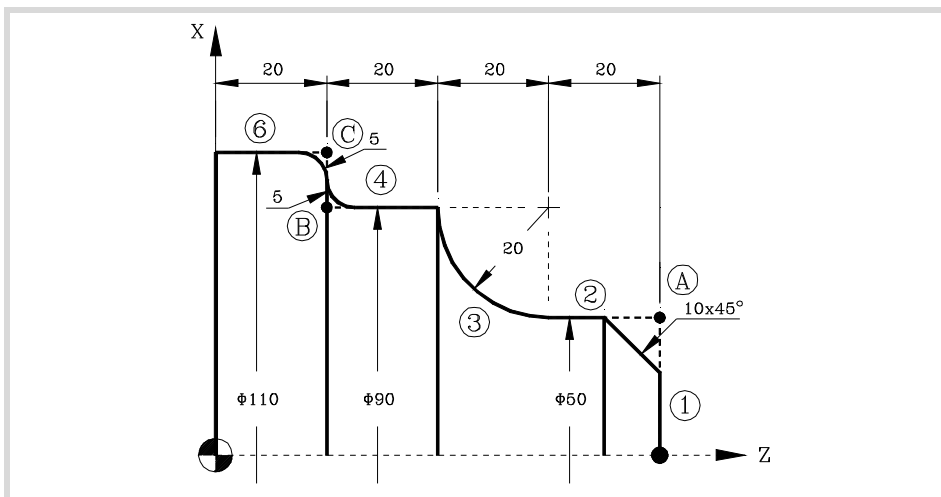
CANNED CYCLES OF THE EDITOR. PROFILES.
Profile turning cycle.



CNC 8060
CNC 8065

(REF: 1502)

6.2.2 Programming example.



Geometric parameters.



Outside machining.



Work quadrant.

X 120.0000
Z 90.0000

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 = 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.



Δ 2 F 0.8 S 1000 T 3

Machining parameters. Finishing.



δ 0.25 F 0.4 S 1000 T 3

6.

CANNED CYCLES OF THE EDITOR. PROFILES.
Profile turning cycle.

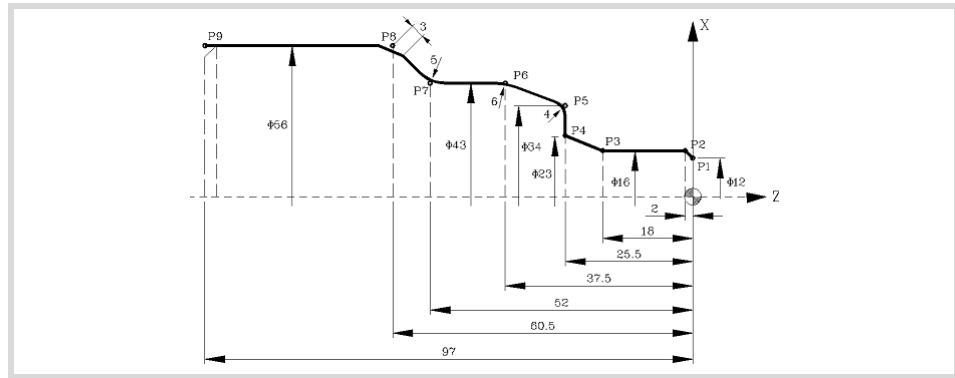


CNC 8060
CNC 8065

(REF: 1502)

6.

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.

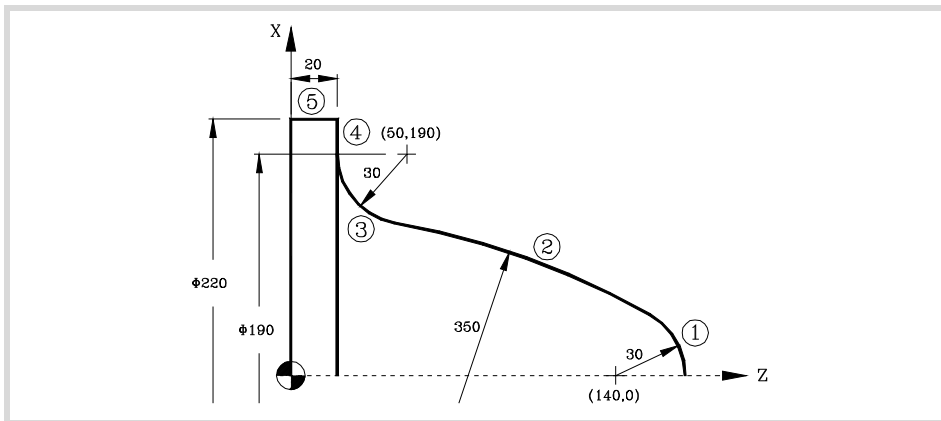


δ 0.25 F 0.4 S 1000 T 3



CNC 8060
CNC 8065

(REF: 1502)



6.

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		

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

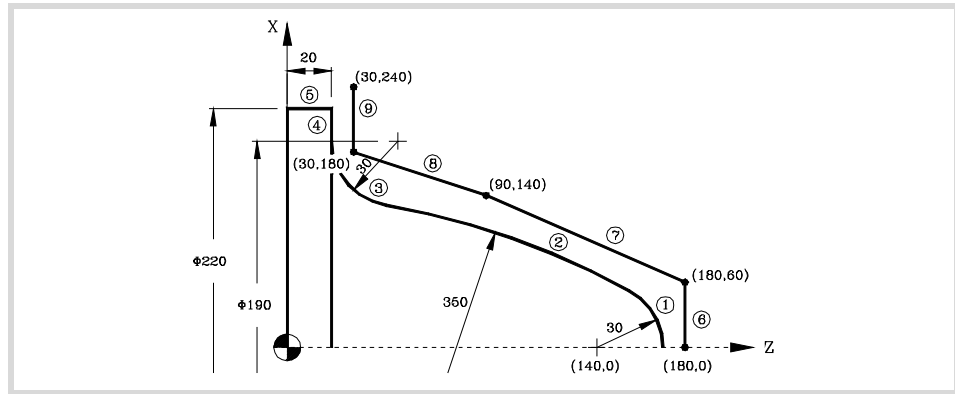


CNC 8060
CNC 8065

(REF: 1502)

6.

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.

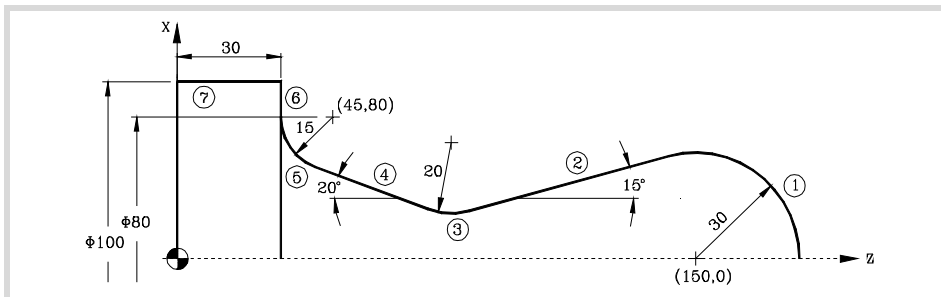


δ 0.25 F 0.4 S 1000 T 3



CNC 8060
CNC 8065

(REF: 1502)



6.

CANNED CYCLES OF THE EDITOR. PROFILES.
Profile turning cycle.

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

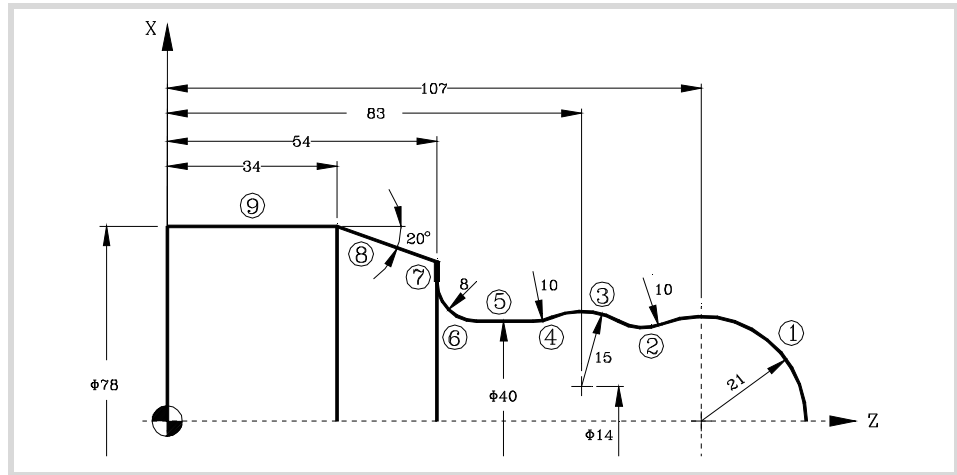


CNC 8060
CNC 8065

(REF: 1502)

6.

CANNED CYCLES OF THE EDITOR. PROFILES.
Profile turning cycle.



Geometric parameters.



Outside machining.



Work quadrant.

X 85.0000
Z 135.0000

Starting point.

Dx 1.0000
Dz 1.0000

Safety distance.

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.



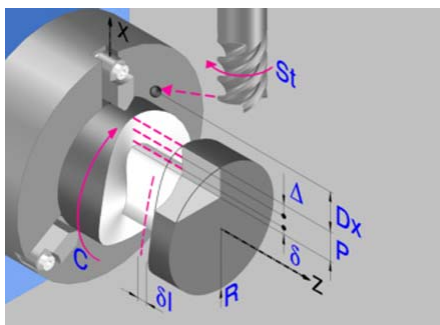
δ 0.25 F 0.4 S 1000 T 3



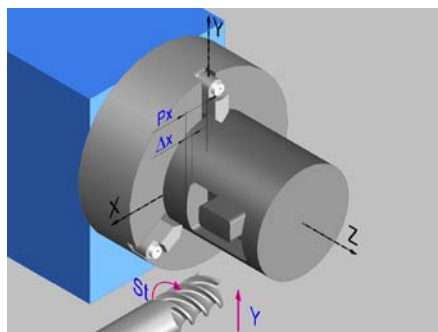
CNC 8060
CNC 8065

(REF: 1502)

6.3 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. The coordinate must always be programmed in radius.

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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.
ZC/YZ plane profiling cycle.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

6.

Spindle angular position.

ω 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.

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 , δ) (δ , δ).

6.

CANNED CYCLES OF THE EDITOR. PROFILES.
ZC/YZ plane profiling cycle.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

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.

δl Side finishing stock. This parameter is only available when tool radius compensation is active.

δ 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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.
ZC/YZ plane profiling cycle.



CNC 8060
CNC 8065

(REF: 1502)

6.3.1 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).

6.

CANNED CYCLES OF THE EDITOR. PROFILES.

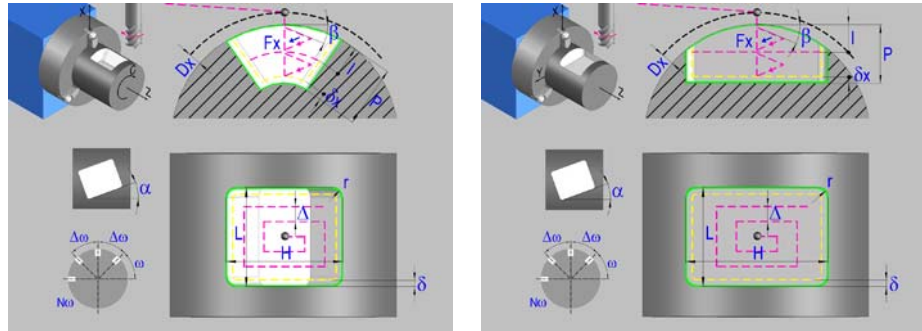
ZC/YZ plane profiling cycle.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

6.4 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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.
Rectangular pocket cycle in the ZC/YZ plane.



CNC 8060
CNC 8065

(REF: 1502)

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.

α 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. The coordinate must always be programmed in radius.

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.

ω 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 ω 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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.
Rectangular pocket cycle in the ZC/YZ plane.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

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.

β Sideways penetration angle.

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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.
Rectangular pocket cycle in the ZC/NZ 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 (δ , δ_l) (δ , δ_l).

Sideways penetration angle.

θ Sideways penetration angle.

Finishing stocks.

δ Side finishing stock.

δ_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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.

Rectangular pocket cycle in the ZC/YZ plane.

FAGOR CNC 8060
CNC 8065

(REF: 1502)

- 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. PROFILES.
Rectangular pocket cycle in the ZC/YZ plane.



CNC 8060
CNC 8065

(REF: 1502)

6.4.1 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 (δ , δx), 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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.

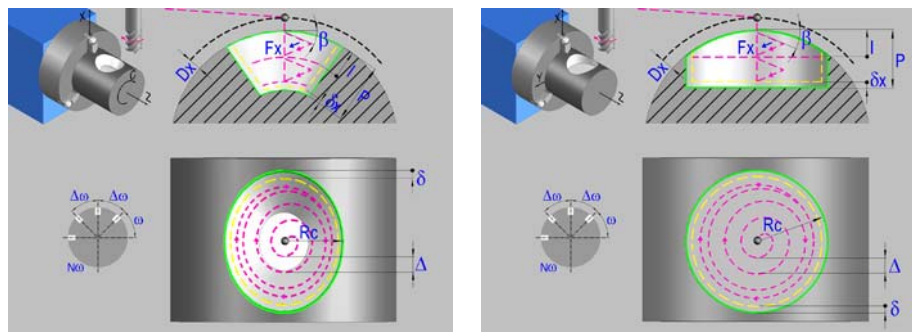
Rectangular pocket cycle in the ZC/NZ plane.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

6.5 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. The coordinate must always be programmed in radius.

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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.
Circular pocket cycle in the ZC/YZ plane.



CNC 8060
CNC 8065

(REF: 1502)

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.

ω 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.

β Sideways penetration angle.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.

Circular pocket cycle in the ZC/NZ plane.

FAGOR CNC 8060
CNC 8065

(REF: 1502)

6.

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.

Finishing stocks.

δ Side finishing stock.

δx 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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.

Circular pocket cycle in the ZC/NZ plane.

FAGOR CNC 8060
CNC 8065

(REF: 1502)

6.5.1 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 (δ , δx), 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.

6.

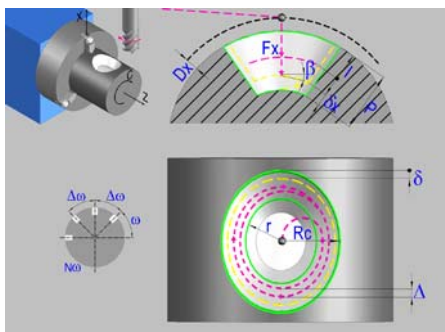
CANNED CYCLES OF THE EDITOR. PROFILES.
Circular pocket cycle in the ZC/NZ plane.



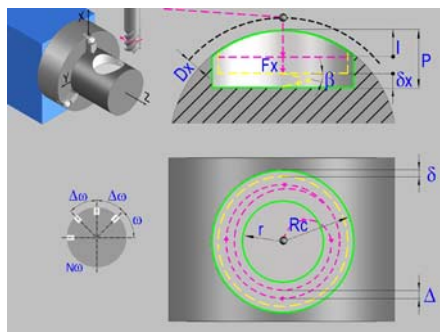
CNC 8060
CNC 8065

(REF: 1502)

6.6 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. The coordinate must always be programmed in radius.

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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.
Circular pre-empted pocket cycle in the ZC/YZ plane.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

6.

CANNED CYCLES OF THE EDITOR. PROFILES.
Circular pre-empted pocket cycle in the ZC/NZ 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.

ω 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.

β 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.

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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.

Circular pre-empted pocket cycle in the ZC/NZ plane.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

6.

CANNED CYCLES OF THE EDITOR. PROFILES.
Circular pre-empted pocket cycle in the ZC/YZ plane.



Finishing stocks.

- δ Side finishing stock.
- δx 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.

6.6.1 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 (δ , δx), 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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.

Circular pre-empted pocket cycle in the ZC/NZ plane.

FAGOR 

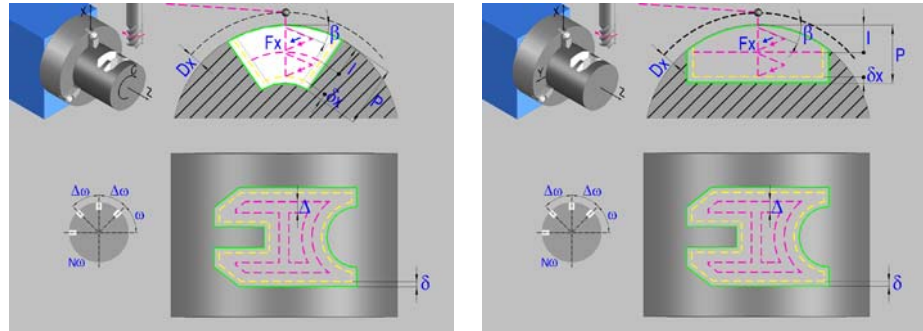
CNC 8060
CNC 8065

(REF: 1502)

6.7 2D profile pocket cycle in the ZC/YZ plane.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.
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. The coordinate must always be programmed in radius.

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.



CNC 8060
CNC 8065

(REF: 1502)

Angular position of the machining operations.






- ω 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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.
2D profile pocket cycle in the ZC/NZ plane.

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.

β Sideways penetration angle.

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.

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.

Finishing stocks.

δ Side finishing stock.

δx 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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.

2D profile pocket cycle in the ZC/NZ plane.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

6.7.1 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 (δ , δx), 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.

6.

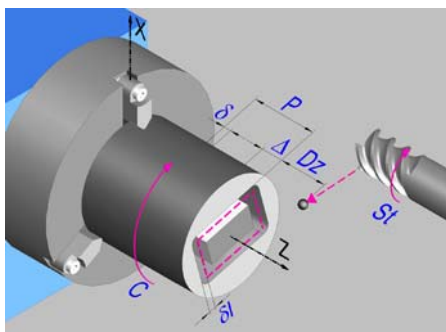
CANNED CYCLES OF THE EDITOR. PROFILES.
2D profile pocket cycle in the ZC/NZ plane.



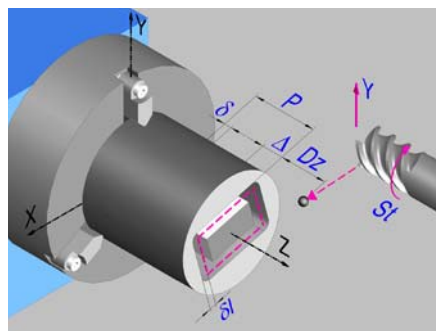
CNC 8060
CNC 8065

(REF: 1502)

6.8 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.

ω Angular position of the spindle, in degrees.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.
XC/XY plane profiling cycle.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

6.

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.

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 , δ l) (δ , δ l).

6.

CANNED CYCLES OF THE EDITOR. PROFILES.

XC/XY plane profiling cycle.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

6.

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.


δl Side finishing stock. This parameter is only available when tool radius compensation is active.


δ 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.

6.8.1 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).

6.

CANNED CYCLES OF THE EDITOR. PROFILES.

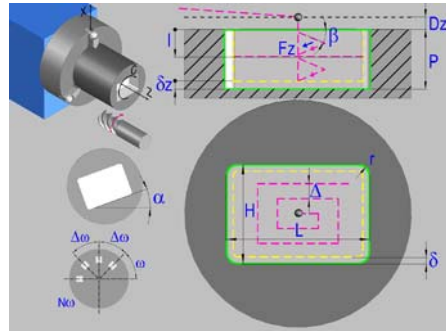
XC/XY plane profiling cycle.

FAGOR 

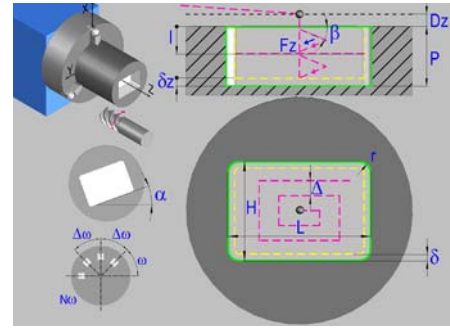
CNC 8060
CNC 8065

(REF: 1502)

6.9 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). The X axis coordinate must always be programmed in radius.

X, Y Coordinates of the starting point (if XY plane). The X axis coordinate must always be programmed in radius.

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

6.

CANNED CYCLES OF THE EDITOR. PROFILES.
Rectangular pocket cycle in the XC/XY plane.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

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.

α 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.

ω 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 ω 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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.

Rectangular pocket cycle in the XC/XY plane.

FAGOR CNC 8060
CNC 8065

(REF: 1502)

6.

CANNED CYCLES OF THE EDITOR. PROFILES.
Rectangular pocket cycle in the XC/XY plane.

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. β Sideways penetration angle.**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 , δ_l) (δ , δ_l).

Sideways penetration angle.

θ Sideways penetration angle.

Finishing stocks.

δ Side finishing stock.

δ_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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.

Rectangular pocket cycle in the Xc/Yc plane.

FAGOR

CNC 8060
CNC 8065

(REF: 1502)

- 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. PROFILES.
Rectangular pocket cycle in the XC/XY plane.



CNC 8060
CNC 8065

(REF: 1502)

6.9.1 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 (δ , δx), 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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.

Rectangular pocket cycle in the Xc/Yc plane.

FAGOR 

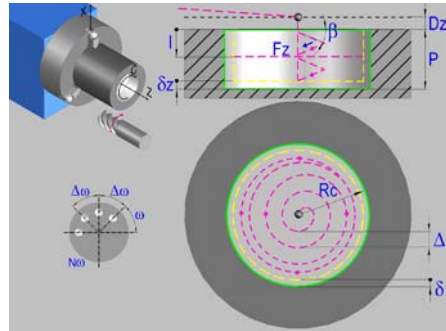
CNC 8060
CNC 8065

(REF: 1502)

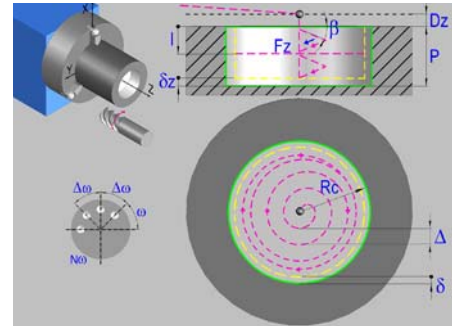
6.10 Circular pocket cycle in the XC/XY plane.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.
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.

X_c, C_c Coordinates of the center of the pocket (if XC plane). The X axis coordinate must always be programmed in radius.

X_c, Y_c Coordinates of the center of the pocket (if XY plane). The X axis coordinate must always be programmed in radius.

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).

Y_f 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.

R_c 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.



CNC 8060
CNC 8065

(REF: 1502)

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.

ω 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.

β Sideways penetration angle.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.

Circular pocket cycle in the XC/XY plane.

FAGOR 
CNC 8060
CNC 8065

(REF: 1502)

6.

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 , δ) (δ , δ).

Sideways penetration angle.

θ Sideways penetration angle.

Finishing stocks.


δ Side finishing stock.


δx 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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.

Circular pocket cycle in the X/Z/Y plane.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

6.10.1 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 (δ , δx), 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.

6.

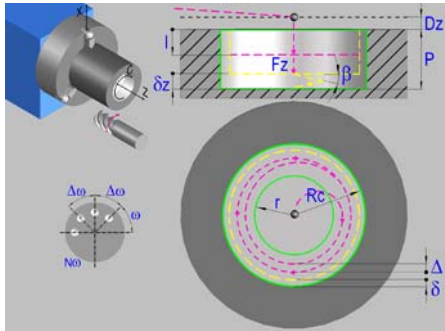
CANNED CYCLES OF THE EDITOR. PROFILES.
Circular pocket cycle in the XC/XY plane.



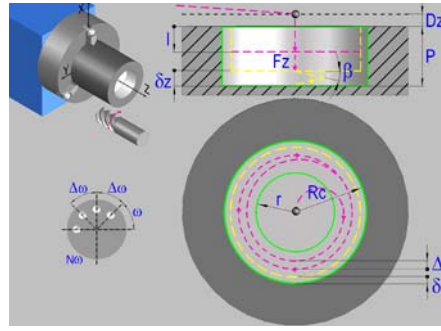
CNC 8060
CNC 8065

(REF: 1502)

6.11 Circular pre-empted pocket in the XC/XY plane.



Circular pre-empted pocket in the XC plane.



Circular pre-empted 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). The X axis coordinate must always be programmed in radius.

Xc, Yc Coordinates of the center of the pocket (if XY plane). The X axis coordinate must always be programmed in radius.

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.

r Inside radius of the pocket.

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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.
Circular pre-empted pocket in the XC/XY plane.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

6.

CANNED CYCLES OF THE EDITOR. PROFILES.
Circular pre-empted pocket 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.

ω 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 ω 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.

β 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.

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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.
Circular pre-empted pocket in the XC/XY plane.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

6.



Finishing stocks.

- δ Side finishing stock.
- δx 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.

6.11.1 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 (δ , δx), 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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.

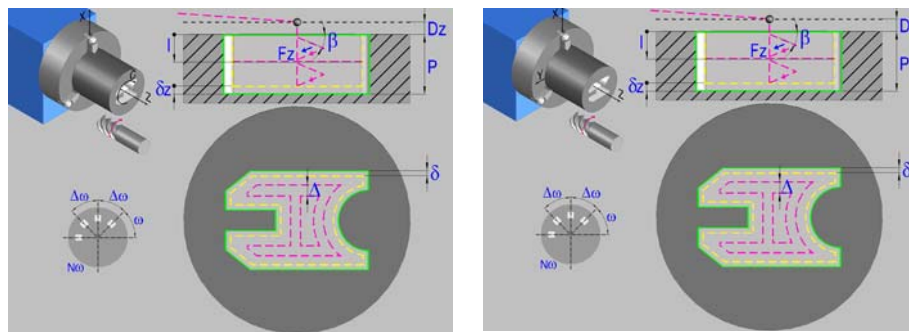
Circular pre-empted pocket in the XC/XY plane.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

6.12 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. ω 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 ω Total number of operations.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.
2D profile pocket cycle in the XC/XY plane.



CNC 8060
CNC 8065

(REF: 1502)

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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.

2D profile pocket cycle in the XC/XY plane.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

6.

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.

- β Sideways penetration angle.

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.

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 , δI) (δ , δI).

Sideways penetration angle.

θ Sideways penetration angle.

Finishing stocks.

δ Side finishing stock.

δx 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.

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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.
2D profile pocket cycle in the XC/XY plane.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

6.12.1 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 (δ , δx), 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.

6.

CANNED CYCLES OF THE EDITOR. PROFILES.
2D profile pocket cycle in the XC/XY plane.



CNC 8060
CNC 8065

(REF: 1502)

CANNED CYCLES OF THE EDITOR. LONGITUDINAL MACHINING.

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.



- 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 ["9 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.



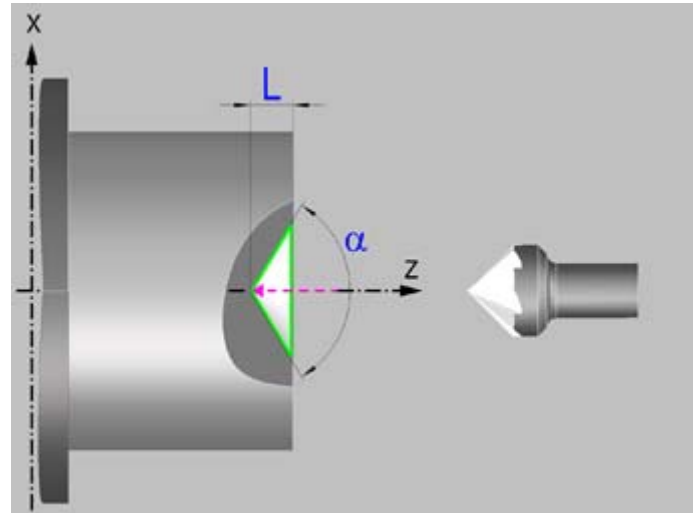
CNC 8060
CNC 8065

(REF: 1502)

7.1 Center punching.

7.

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.

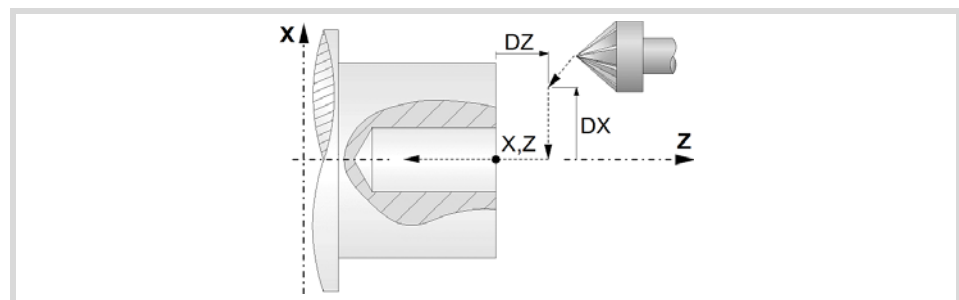
Although the center punching is normally done in the rotation center, the CNC enables X to be defined with a value other than 0 and make grooves in the front face of the part. For center punching, X must be 0.

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: 1502)

FAGOR 

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.
- α Center-punching angle.
- ϕ 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.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL
Center punching.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

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.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL
Center punching.



CNC 8060
CNC 8065

(REF: 1502)

7.1.1 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).

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL
Center punching.



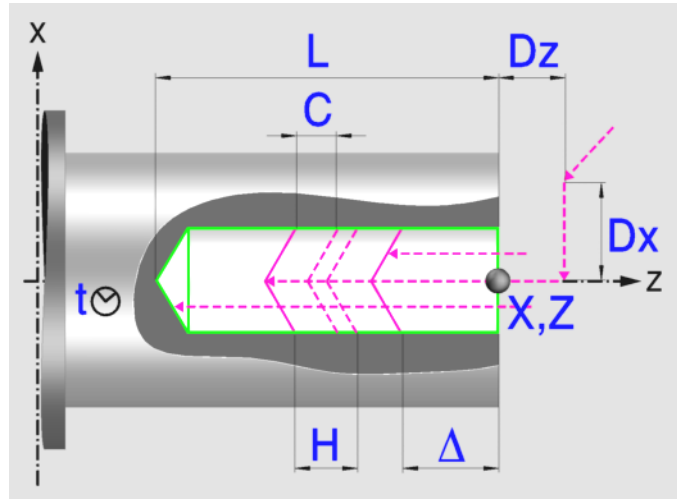
CNC 8060
CNC 8065

(REF: 1502)

7.2 Drilling.

7.

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.

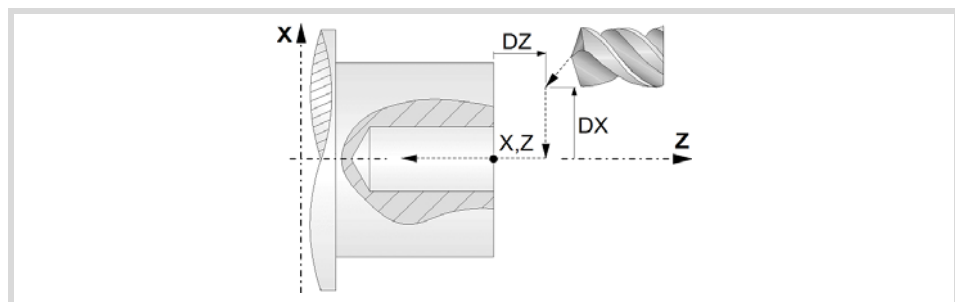
Although the drilling is normally done in the rotation center, the CNC enables X to be defined with a value other than 0 and make grooves in the front face of the part. For drilling, X must be 0.

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.

(REF: 1502)

FAGOR 

CNC 8060
CNC 8065

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.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL Drilling.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

7.

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.

7.2.1 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 " δ ". 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 Δ defines the drilling peck and $K\Delta$ the reduction factor for this peck. The first drilling peck will be " Δ ", the second one " $K\Delta$ ", 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).

7.

Drilling.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL

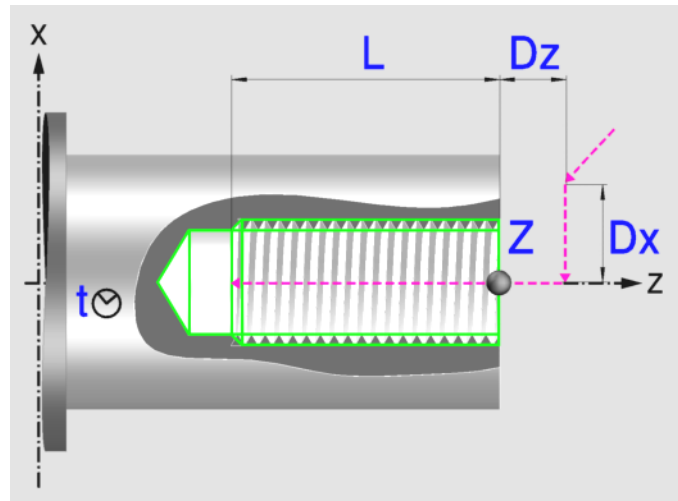
FAGOR 
CNC 8060
CNC 8065

(REF: 1502)

7.3 Tapping.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL Tapping.



Geometric parameters.

Coordinates of the starting point

Z Z coordinate of the starting point.

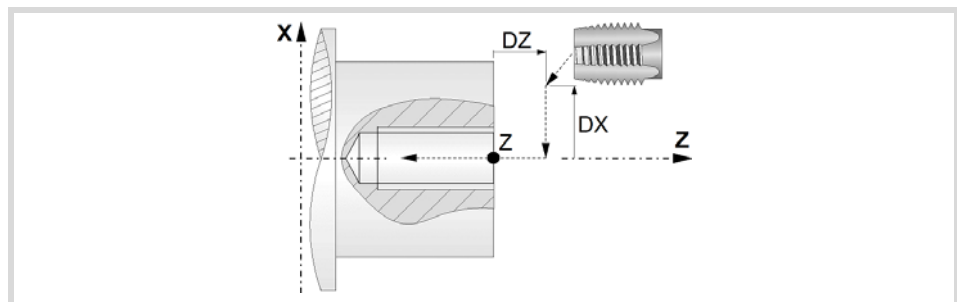
To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

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.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL Tapping.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

7.3.1 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.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL Tapping.

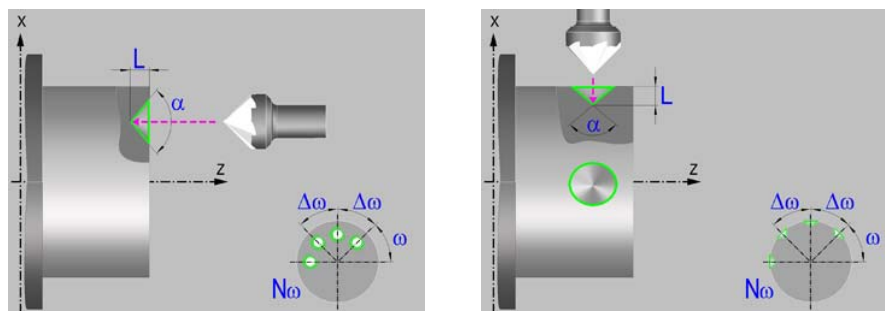


CNC 8060
CNC 8065

(REF: 1502)

7.4 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. In the ZX plane, the coordinate must be given in the active units, radius or diameter. With C axis or Y axis, the coordinate must always be programmed in radius.

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.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL
Multiple center-punching.

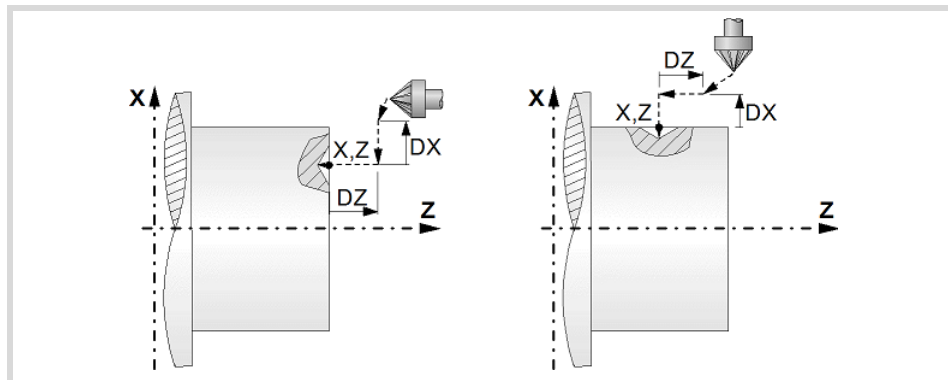
FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

7.

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.

α Center-punching angle.

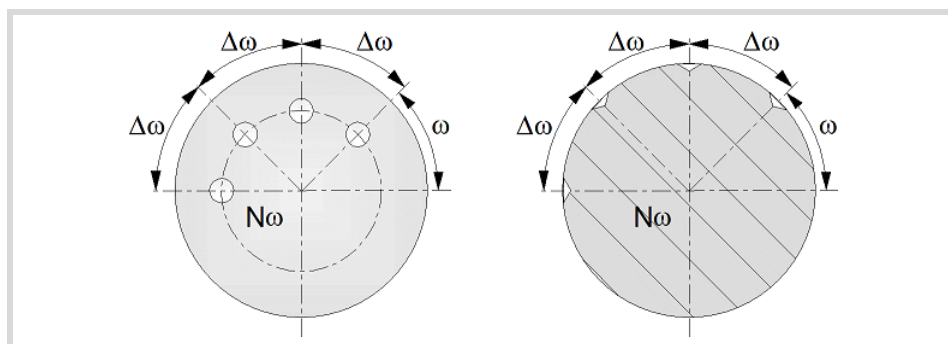
ϕ Center-punching diameter.

Angular position of the machining operations.

ω 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.



CNC 8060
CNC 8065

(REF: 1502)

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.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL
Multiple center-punching.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

7.4.1 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.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL
Multiple center-punching.

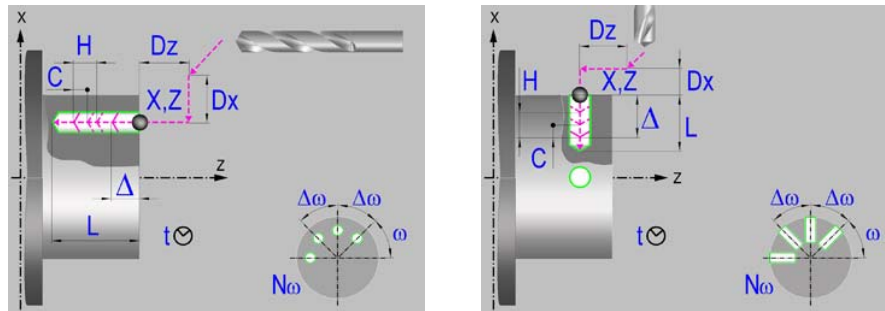


CNC 8060
CNC 8065

(REF: 1502)

7.5 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. In the ZX plane, the coordinate must be given in the active units, radius or diameter. With C axis or Y axis, the coordinate must always be programmed in radius.

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.

7.

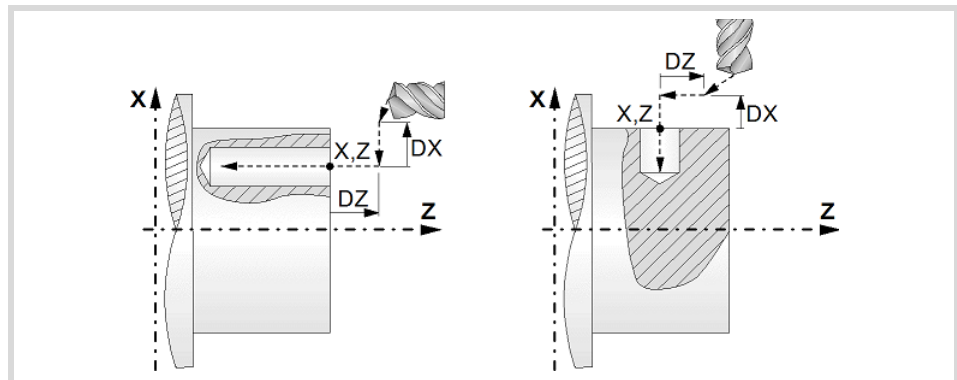
CANNED CYCLES OF THE EDITOR. LONGITUDINAL Multiple drilling.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

- 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.

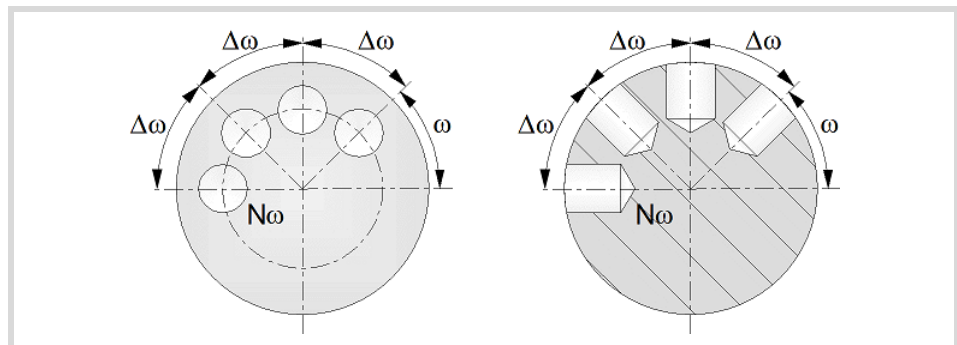


Programming of the drilling operation.

- 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.

- ω 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.**Drilling step (peck); constant-peck or variable-peck drilling.**

- Δ Maximum drilling peck (step).
- $K\Delta$ 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 " $K\Delta$ 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 " 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.

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.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL Multiple drilling.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

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.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL
Multiple drilling.



CNC 8060
CNC 8065

(REF: 1502)

7.5.1 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 " δ ". 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 Δ defines the drilling peck and $K\Delta$ the reduction factor for this peck. The first drilling peck will be " Δ ", the second one " $K\Delta$ ", 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.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL
Multiple drilling.

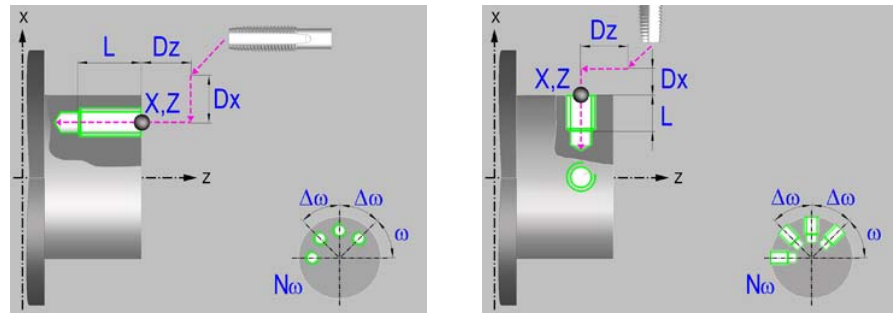


CNC 8060
CNC 8065

(REF: 1502)

7.6 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. In the ZX plane, the coordinate must be given in the active units, radius or diameter. With C axis or Y axis, the coordinate must always be programmed in radius.

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.

7.

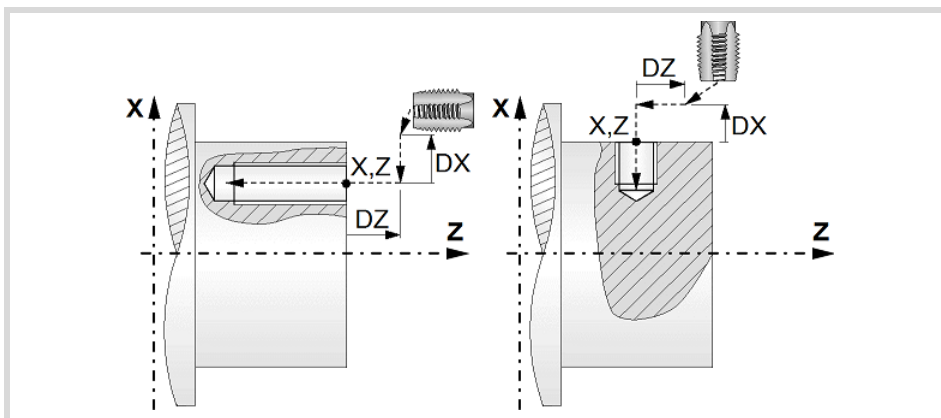
CANNED CYCLES OF THE EDITOR. LONGITUDINAL Multiple tapping.



CNC 8060
CNC 8065

(REF: 1502)

- 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.



7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL Multiple tapping.

Tapping type.



Tapping with a clutch.



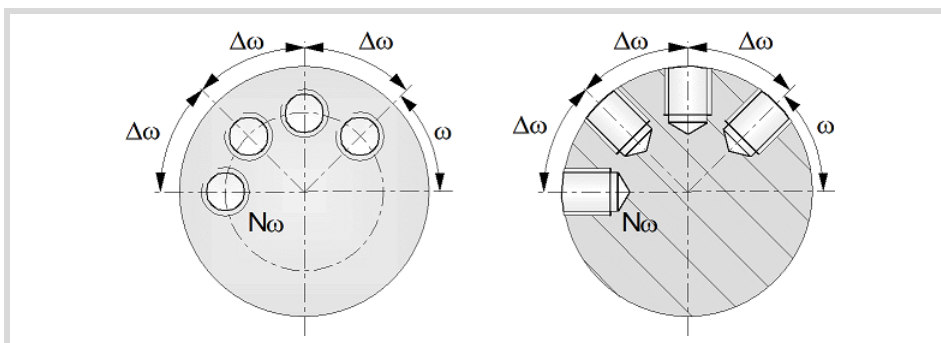
Rigid tapping. The spindle must have a servo-motor-drive system and an encoder.

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.

- ω 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.



CNC 8060
CNC 8065

(REF: 1502)

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.

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.

7.6.1 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 (I_o).
- 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.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL
Multiple tapping.

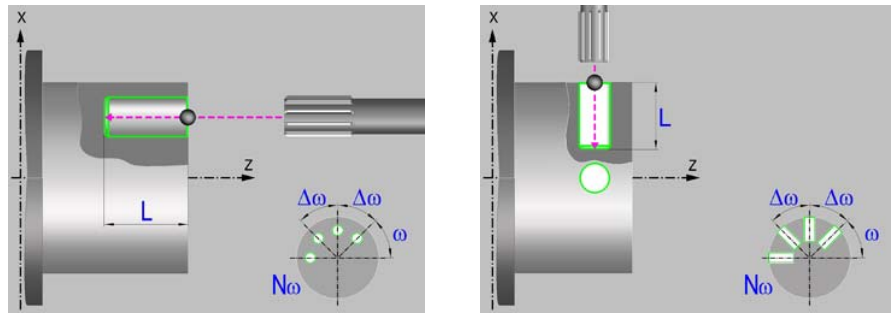
FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

7.7 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. In the ZX plane, the coordinate must be given in the active units, radius or diameter. With C axis or Y axis, the coordinate must always be programmed in radius.

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.

7.

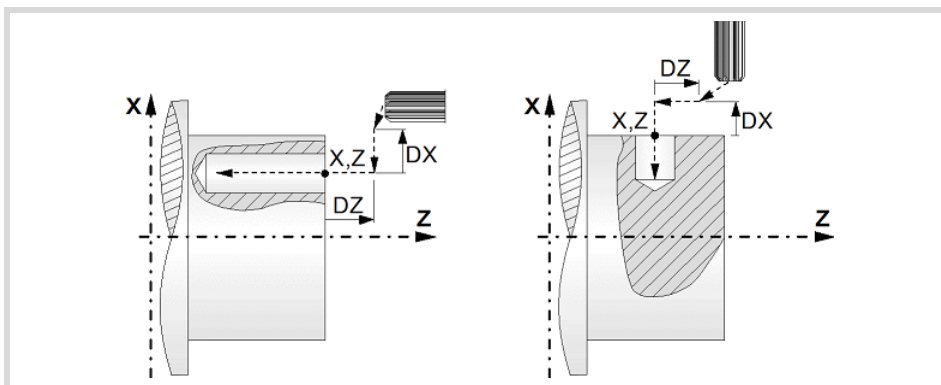
CANNED CYCLES OF THE EDITOR. LONGITUDINAL Multiple reaming.



CNC 8060
CNC 8065

(REF: 1502)

- 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.

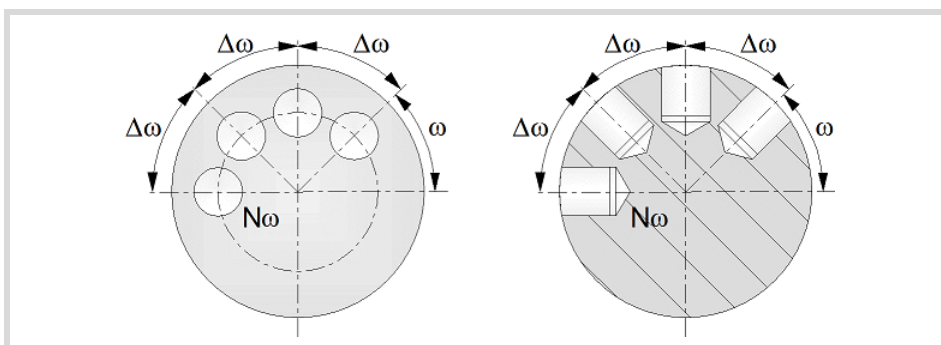


Programming the reaming operation.

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.

- ω 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.

7.**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.

7.7.1 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.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL.
Multiple reaming.

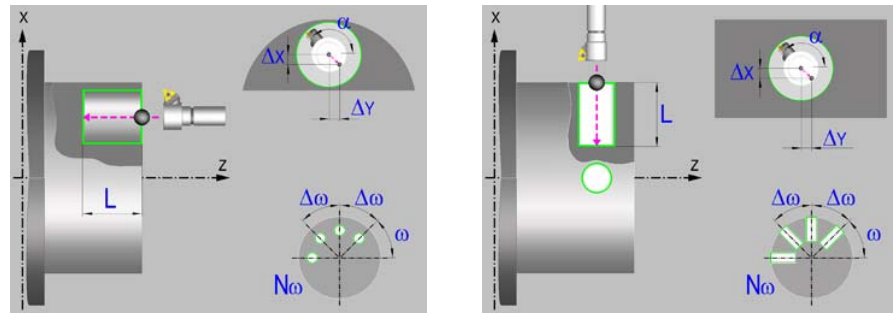


CNC 8060
CNC 8065

(REF: 1502)

7.8 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. In the ZX plane, the coordinate must be given in the active units, radius or diameter. With C axis or Y axis, the coordinate must always be programmed in radius.

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.

7.

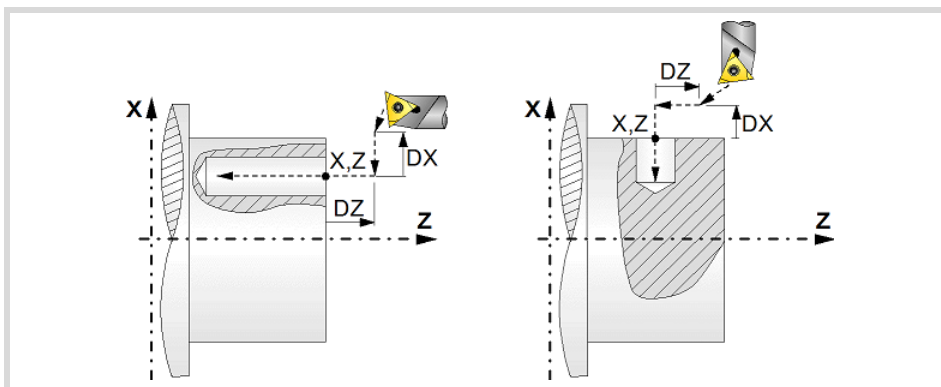
CANNED CYCLES OF THE EDITOR. LONGITUDINAL
Multiple boring.



CNC 8060
CNC 8065

(REF: 1502)

- 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.

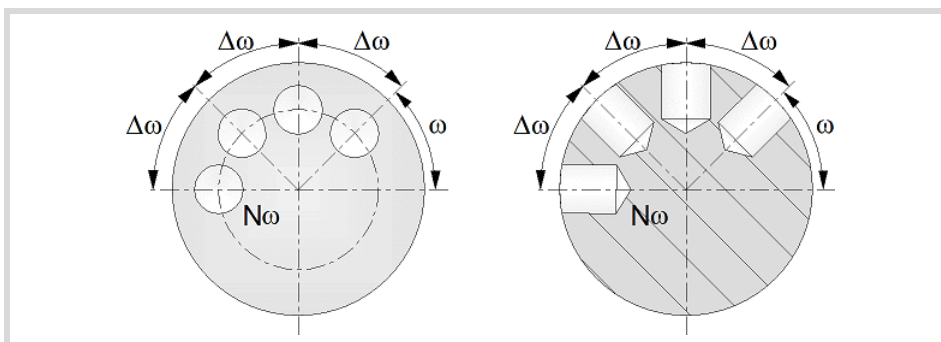


Programming the boring operation.

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.

- ω 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.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL
Multiple boring.



CNC 8060
CNC 8065

(REF: 1502)

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.**Machining parameters.****Type of withdrawal.**

Withdrawal at work feedrate while the spindle is turning.

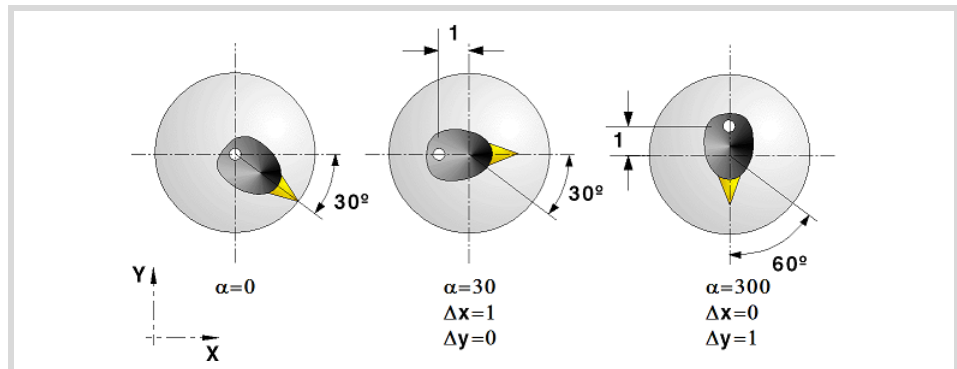


Rapid withdrawal while the spindle is stopped and the tool is touching the surface.



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.

- α Angular position, in degrees, of the spindle of the live tool for the withdrawal.
- ΔX X distance away from (off) the wall of the hole; if not programmed, the cycle will assume a 0 value.
- ΔY 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.

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.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL
Multiple boring.



CNC 8060
CNC 8065

(REF: 1502)

7.8.1 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.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL
Multiple boring.

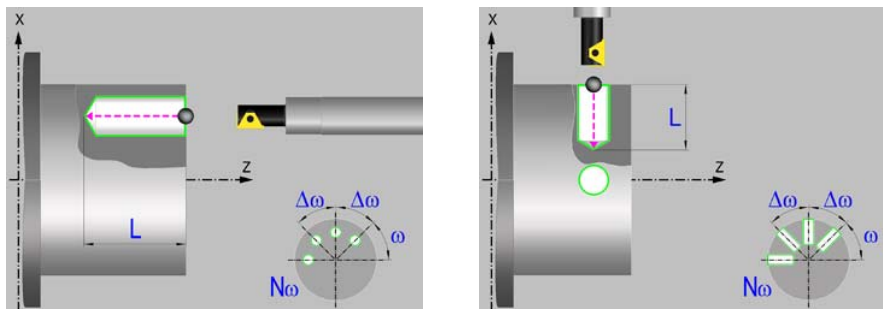


CNC 8060
CNC 8065

(REF: 1502)

7.9 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.

7.

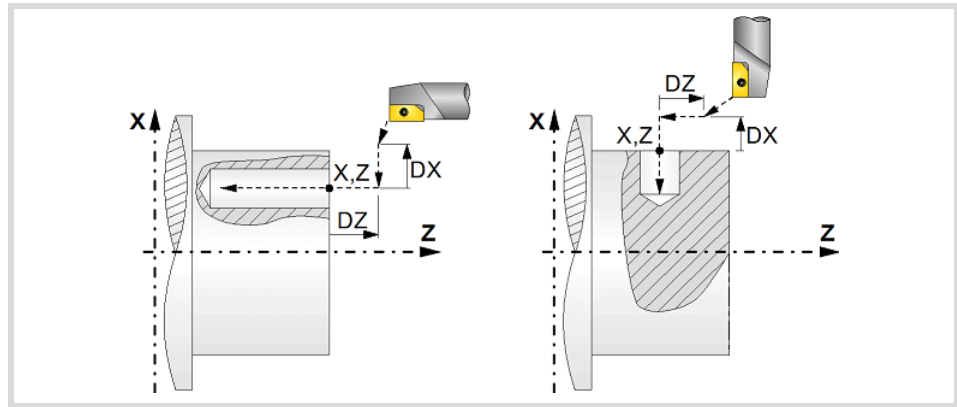
CANNED CYCLES OF THE EDITOR. LONGITUDINAL
Multiple bore milling.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

7.

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.

\emptyset Hole diameter.

$\emptyset k$ Pre-hole diameter.

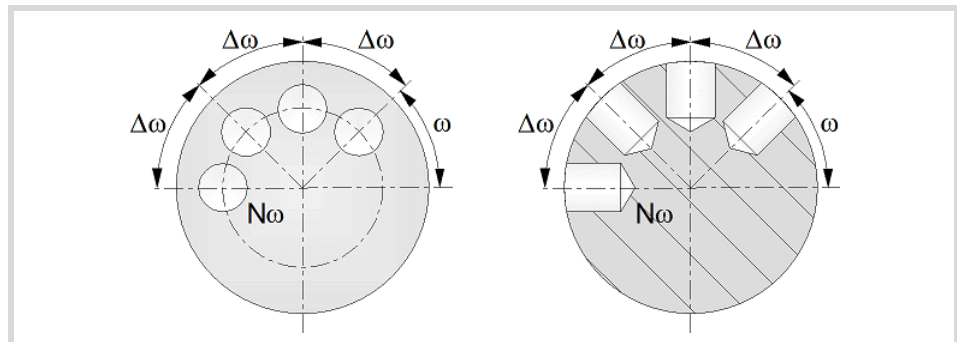
When starting from a hole that was machined earlier, parameter $\emptyset k$ defines the diameter of that hole; if not programmed or programmed with a 0 value, the cycle assumes that there is no previous hole.

Angular position of the machining operations.

ω Angular position of the spindle (in degrees) for the first machining operation.

$\Delta\omega$ 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.

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.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL Multiple bore milling.



CNC 8060
CNC 8065

(REF: 1502)

7.9.1 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.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL
Multiple bore milling.

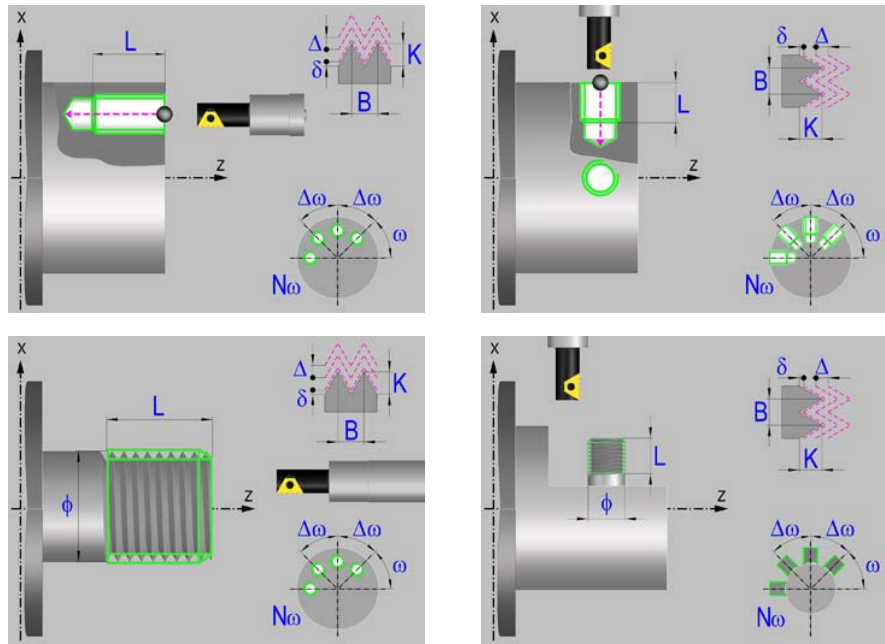


CNC 8060
CNC 8065

(REF: 1502)

7.10 Multiple thread milling.

The cycle allows multiple thread milling on the side of the part or on its face.



7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL
Multiple thread milling.

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.

FAGOR 

CNC 8060
CNC 8065

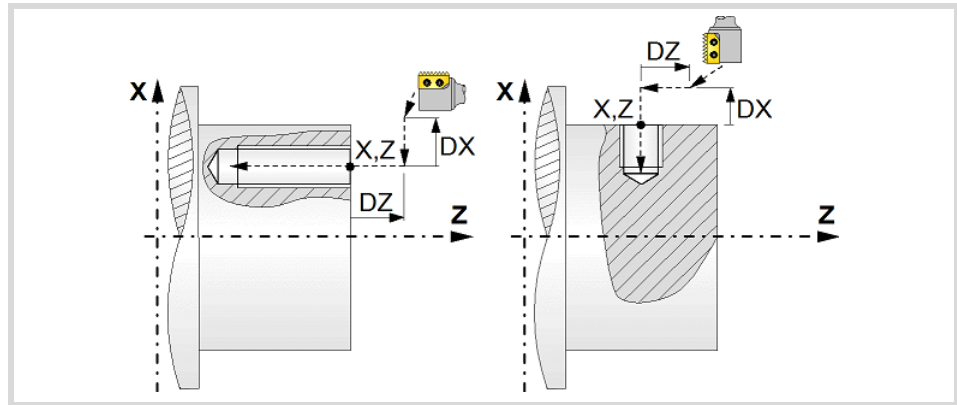
(REF: 1502)

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.

D_x, D_z 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.

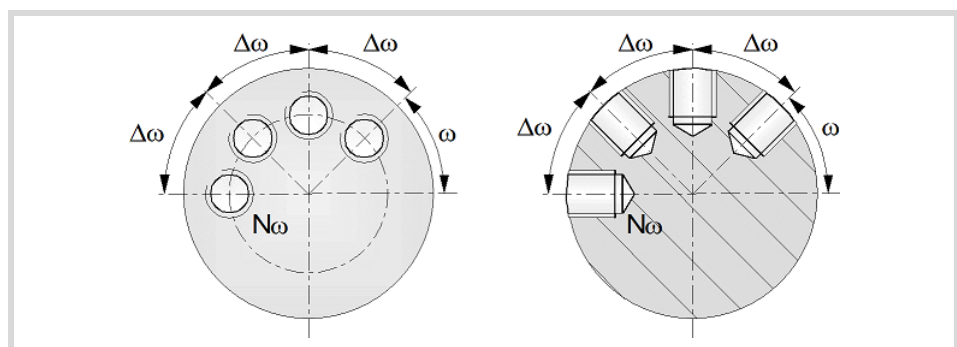
\emptyset Nominal thread diameter.

Angular position of the machining operations.

ω 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.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL
Multiple thread milling.



CNC 8060
CNC 8065

(REF: 1502)

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).

Δ 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.

δ Finishing stock; if not programmed, the cycle assumes a value of 0.

α 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.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL
Multiple thread milling.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

7.

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.

7.10.1 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.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL
Multiple thread milling.

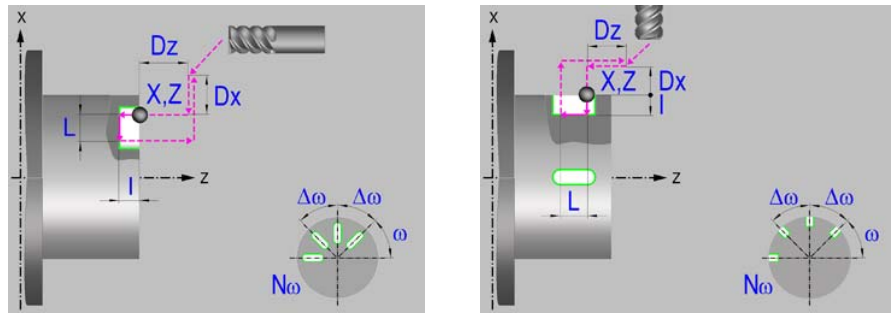
FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

7.11 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. In the ZX plane, the coordinate must be given in the active units, radius or diameter. With C axis or Y axis, the coordinate must always be programmed in radius.

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.

7.

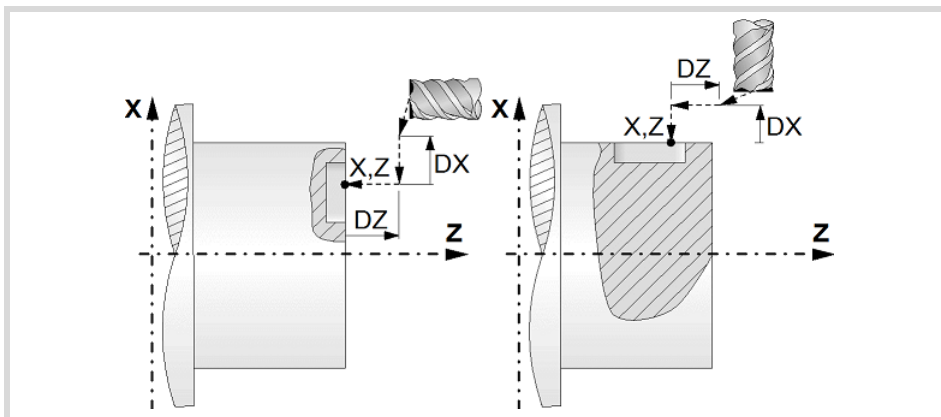
CANNED CYCLES OF THE EDITOR. LONGITUDINAL Multiple slot milling.



CNC 8060
CNC 8065

(REF: 1502)

- 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.

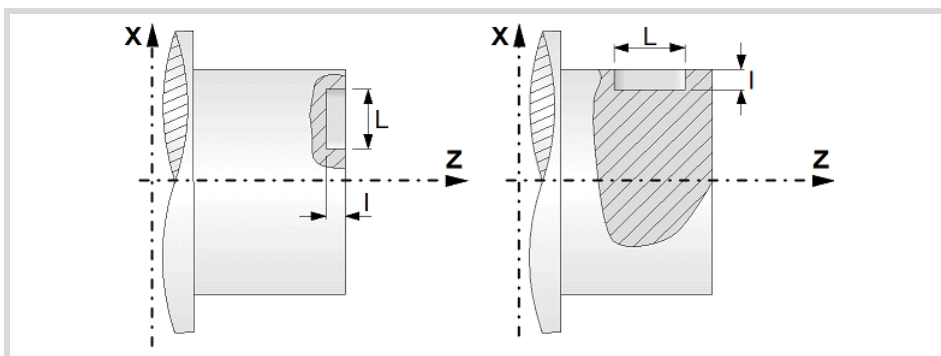


7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL Multiple slot milling.

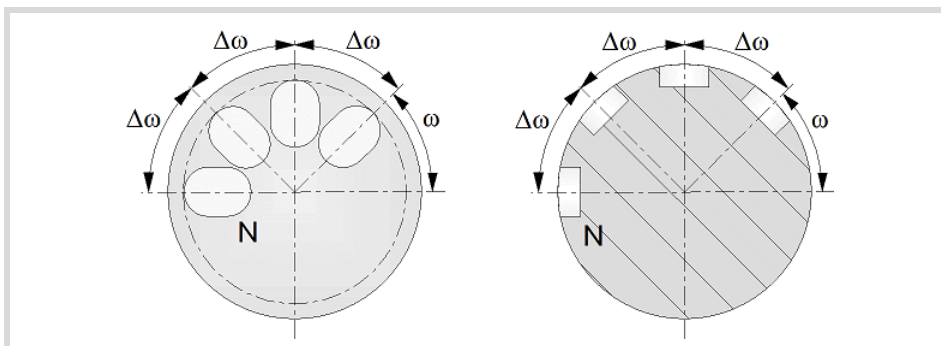
Programming of the drilling operation.

- I Depth of the slot.
- L Length of the slot.



Angular position of the machining operations.

- ω 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.



CNC 8060
CNC 8065

(REF: 1502)

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.

Machining parameters. Roughing.

Penetration step.

Δ 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 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.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL
Multiple slot milling.



CNC 8060
CNC 8065

(REF: 1502)

Machining parameters. Finishing.

Finishing stock at the bottom.

δ Finishing stock at the bottom.

Feedrate.

F Finishing feedrate.

Live-tool data.

S Live tool turning speed.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL
Multiple slot milling.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

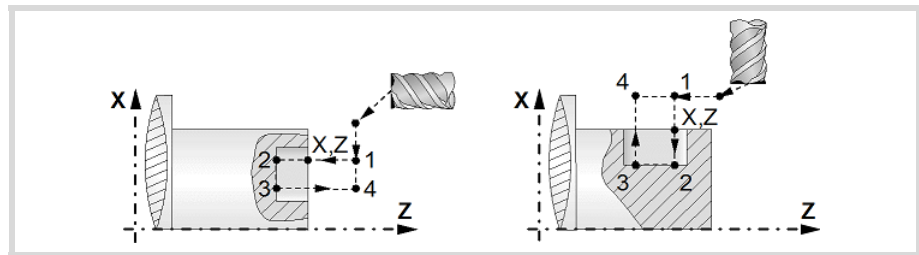
7.11.1 Basic operation.

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.

7.

CANNED CYCLES OF THE EDITOR. LONGITUDINAL Multiple slot milling.



CNC 8060
CNC 8065

(REF: 1502)

CANNED CYCLES OF THE EDITOR. POSITIONING.

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.

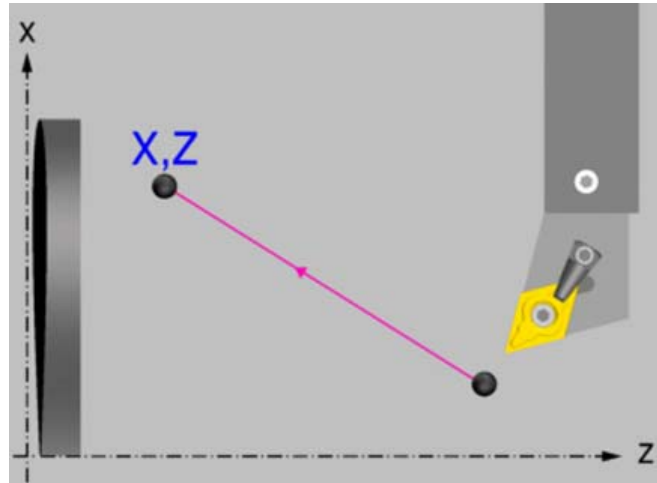


- Positioning cycle.
- Positioning cycle with M functions.

8.1 Positioning cycle.

8.

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 end point. The X axis coordinate must be programmed in the active units, radius or diameter.

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.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

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.

8.

CANNED CYCLES OF THE EDITOR. POSITIONING.

Positioning cycle.



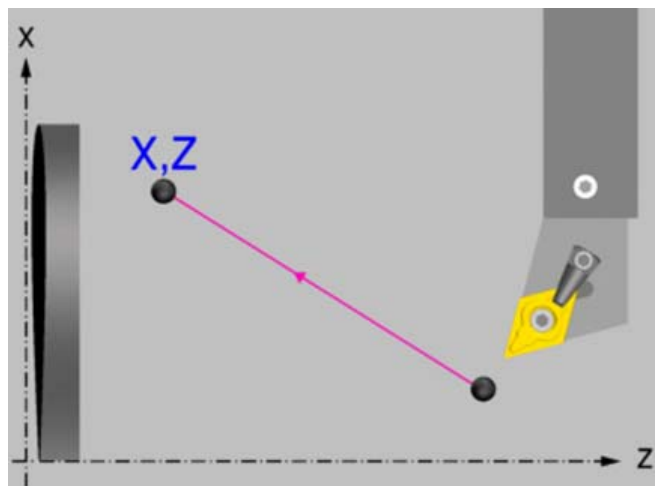
CNC 8060
CNC 8065

(REF: 1502)

8.2 Positioning cycle with M functions.

8.

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 end point. The X axis coordinate must be programmed in the active units, radius or diameter.

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.



CNC 8060
CNC 8065

(REF: 1502)

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.

8.

CANNED CYCLES OF THE EDITOR. POSITIONING.

Positioning cycle with M functions.



CNC 8060
CNC 8065

(REF: 1502)

8.

CANNED CYCLES OF THE EDITOR. POSITIONING.

Positioning cycle with M functions.



CNC 8060
CNC 8065

(REF: 1502)

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.

9

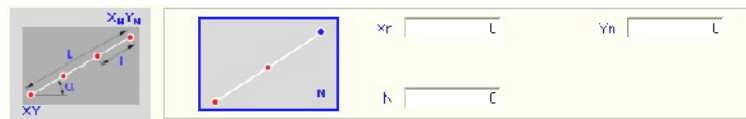
When pressing the softkey, the editor shows the cycle used last. When pressing the same softkey again, the menu shows all the cycles of the group.



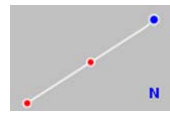
- Multiple machining in straight line
- Multiple machining in arc pattern
- Multiple machining in rectangular pattern
- Multiple machining in grid pattern
- Random multiple machining (several points defined by the user).
- Without multiple machining.

9.1 Multiple machining in straight line.

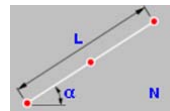
There are 5 different ways to define the machining operation. Use the icon to select the desired option.



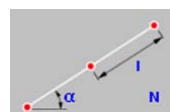
Positioning options.



End point and number of machining operations.



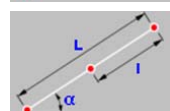
Length, angle and number of machining operations.



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.

α Angle of the line with the abscissa axis.

L Total length of the line.

I Distance between two consecutive machining operations.

9.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.
Multiple machining in straight line.

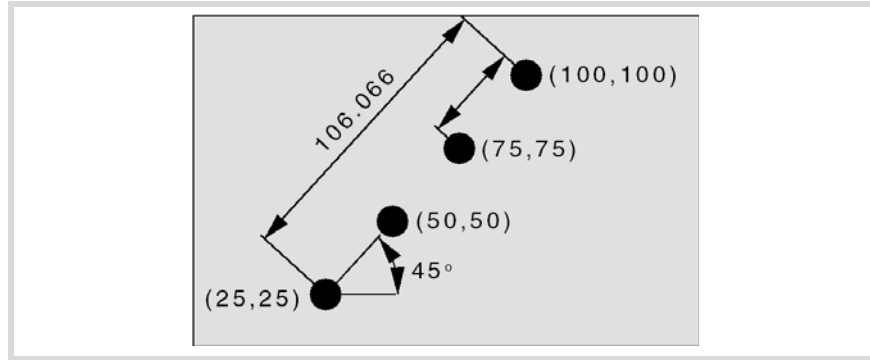


CNC 8060
CNC 8065

(REF: 1502)

9.1.1 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.	α 45
	Length of the line.	L 106.066
	Total number of machining operations.	N 4
3)	Angle of the path.	α 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.	α 45
	Length of the line.	L 106.066
	Distance between machining operations.	I 35.3553

9.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.

Multiple machining in straight line.

FAGOR 

CNC 8060
CNC 8065

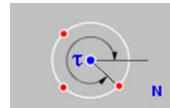
(REF: 1502)

9.2 Multiple machining in arc pattern.

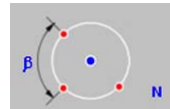
There are 9 ways to define the machining operation. Use the icon to select the desired option.



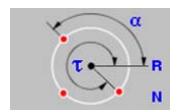
Positioning options.



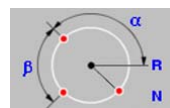
Arc center coordinates, number of machining operations and final angle of the arc.



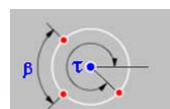
Arc center coordinates, number of machining operations and angular distance between two machining operations.



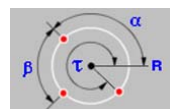
Arc radius, number of machining operations and starting and final angles of the arc.



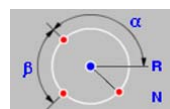
Arc radius, number of machining operations, starting angle of the arc and angular distance between two machining operations.



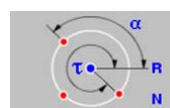
Arc center coordinates, final angle of the arc and angular distance between two machining operations.



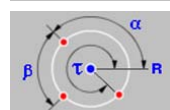
Arc radius, angular distance between two machining operations and starting and final angles of the arc.



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.

α Starting angle of the arc.

τ Final angle of the arc.

β 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.

9.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.
Multiple machining in arc pattern.

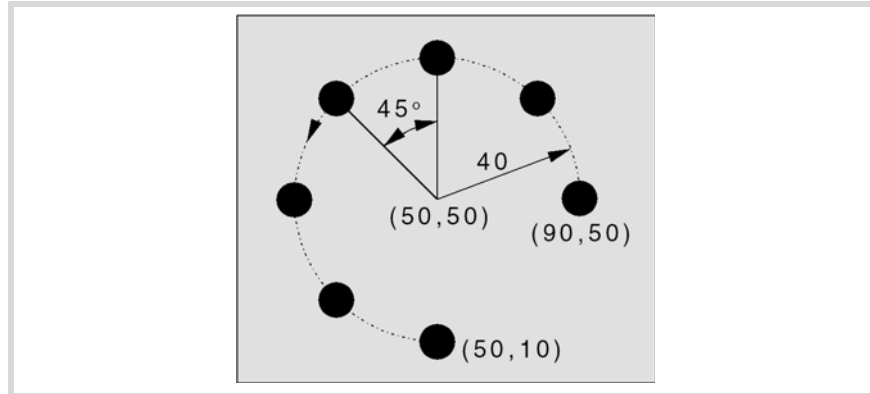


CNC 8060
CNC 8065

(REF: 1502)

9.2.1 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. | τ 270 |
| 2) | Arc center coordinates. | Xa 50, Ya 50 |
| | Total number of machining operations. | N 7 |
| | Angular distance between machining operations. | β 45 |
| 3) | Radius of the arc. | R 40 |
| | Total number of machining operations. | N 7 |
| | Angle of the starting point. | α 0 |
| | Angle of the end point. | τ 270 |
| 4) | Radius of the arc. | R 40 |
| | Total number of machining operations. | N 7 |
| | Angle of the starting point. | α 0 |
| | Angular distance between machining operations. | β 45 |
| 5) | Arc center coordinates. | Xa 50, Ya 50 |
| | Angle of the end point. | τ 270 |
| | Angular distance between machining operations. | β 45 |
| 6) | Radius of the arc. | R 40 |
| | Angle of the starting point. | α 0 |
| | Angle of the end point. | τ 270 |
| | Angular distance between machining operations. | β 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. | α 0 |
| | Angular distance between machining operations. | β 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. | α 0 |
| | Angle of the end point. | τ 270 |
| 9) | Arc center coordinates. | Xa 50, Ya 50 |
| | Radius of the arc. | R 40 |
| | Angle of the starting point. | α 0 |
| | Angle of the end point. | τ 270 |
| | Angular distance between machining operations. | β 45 |

9.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.
Multiple machining in arc pattern.

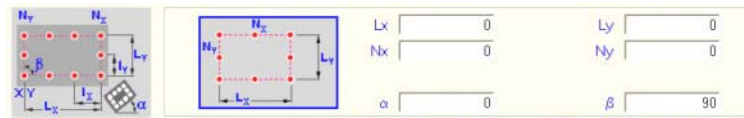
FAGOR 

CNC 8060
CNC 8065

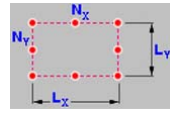
(REF: 1502)

9.3 Multiple machining in rectangular pattern.

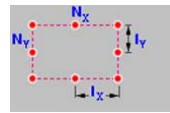
There are 3 ways to define the machining operation. Use the icon to select the desired option.



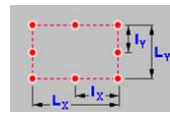
Positioning options.



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.

I_x, I_y Distance between two machining operations, on each axis.

α Angle of the rectangle with the abscissa axis.

β 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).

9.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.
Multiple machining in rectangular pattern.

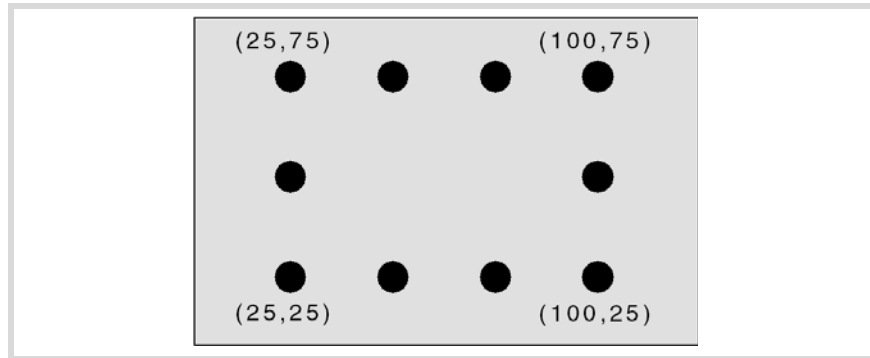


CNC 8060
CNC 8065

(REF: 1502)

9.3.1 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. | α 0 |
| | Angle between paths. | β 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. | α 0 |
| | Angle between paths. | β 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. | α 0 |
| | Angle between paths. | β 90 |

9.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.

Multiple machining in rectangular pattern.

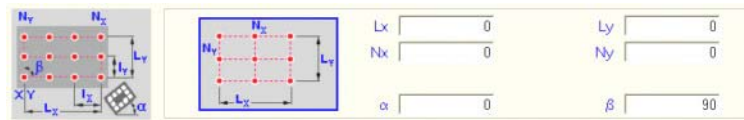
FAGOR 

CNC 8060
CNC 8065

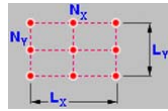
(REF: 1502)

9.4 Multiple machining in grid pattern.

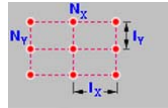
There are 3 ways to define the machining operation. Use the icon to select the desired option.



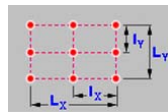
Positioning options.



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.

α Angle of the grid with the abscissa axis.

β 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).

9.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.
Multiple machining in grid pattern.

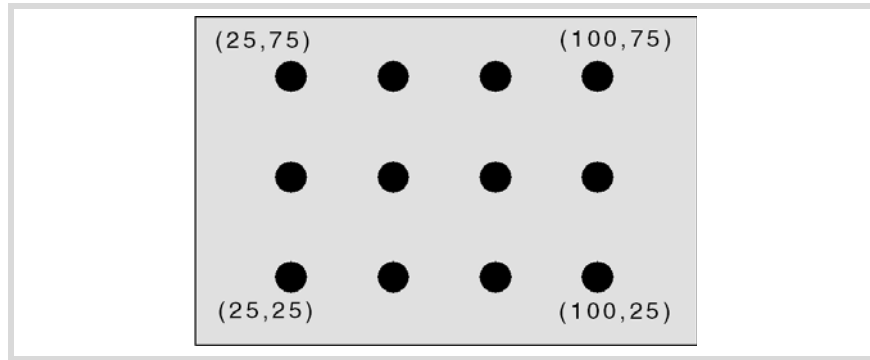


CNC 8060
CNC 8065

(REF: 1502)

9.4.1 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. | α 0 |
| | Angle between paths. | β 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. | α 0 |
| | Angle between paths. | β 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. | α 0 |
| | Angle between paths. | β 90 |

9.

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.

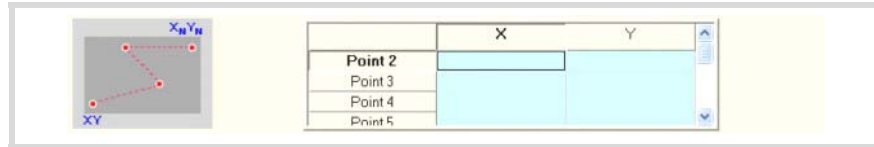
Multiple machining in grid pattern.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

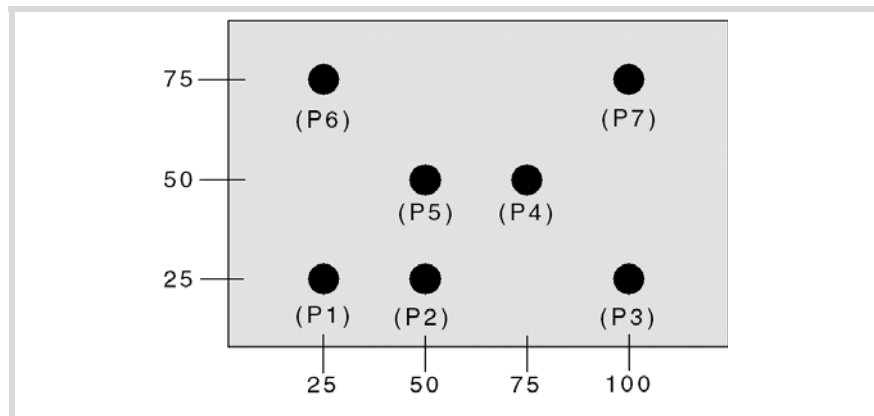
9.5 Random multiple machining.



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.

9.5.1 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
```

DYNAMIC DISTRIBUTION OF THE MACHINING OPERATIONS BETWEEN CHANNELS.

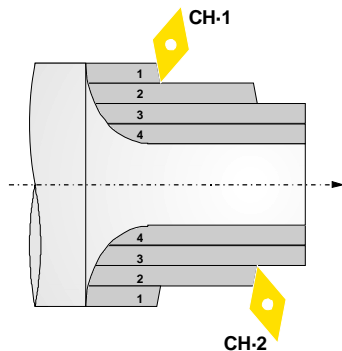
10



This feature can only be used on machines whose configuration allows it, e.g. TT lathes. The machine must have at least two channels and two turrets. If this feature is used with more channels, each one must have access to a different turret.

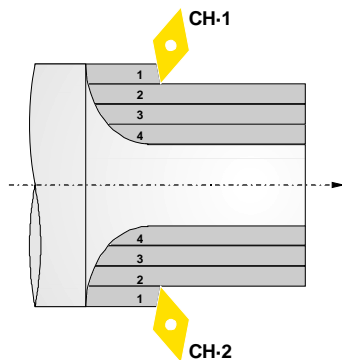
This feature may be used to optimize the machining of the canned cycles, minimizing execution time by distributing the machining of a canned cycle between several channels. The canned cycle is programmed in a single channel (master channel) and the CNC is in charge of distributing or synchronizing the passes between the other channels (slave channels), in one of the following ways.

Distribution of passes between channels.



The canned cycle is executed in a channel and the CNC distributes the machining passes between the channels involved. The CNC synchronizes the channels in such way that when each one starts a pass, another channel starts the next pass, hence machining in half the time.

Synchronized identical passes



The channel that executes the cycle control all the movements. The CNC synchronizes the necessary axes of the other channels with the channel that executes the cycle so all the channels run the same pass, but with an angular shift at the spindle. This makes it possible to increase the feed rate, because the CNC distributes the section to be machined between the tools.

For example, with two tools shifted 180° at the spindle, the feedrate could be double, because each tool machines half the depth of the section.

The CNC only applies this option to the roughing operation of the following cycles, both ISO and of the editor; it does not apply it to the rest of the canned cycles or to individual tool paths.

- Turning and facing canned cycles for arcs and straight sections.
- Grooving and cut-off canned cycles.
- Profile machining canned cycles (except XC and ZC profiles)
- Paraxial (one axis at a time) roughing canned cycles.

Considerations.

When distributing the machining of a canned cycle between different channels, bear in mind the following considerations.

- Only one channel executes the canned cycle (master channel) and the CNC is in charge of distributing the passes between the rest of the channels involved (slave channels). On



CNC 8060
CNC 8065

(REF: 1502)

10.

a TT lathe, the channel that executes the cycle must be the channel that controls the spindle.

- The CNC can apply machining distribution to several slave channels at the same time, as long as each channel has access to a different turret. The slave channels to be used are defined in the calling instruction.
- The master and slave channels that share the machining must belong to the same group of channels.
- All the channels involved must have the same units (radius / diameter) active and the same origin preset or zero offset.

Behavior of the channels that make up a group.

General behavior of a group of channels.

- All the channels are in the same work mode, JOG or automatic.
- An error in any of the channels of the group interrupts the execution in all of them.
- A reset in any of the channels of the group affects all of them.

Particular behavior with machining distribution being active.

- Interrupting and resuming the execution in any of the channels of the group affects all of them.
- The execution in single-block mode affects all the channels of the group.



The manufacturer must include in the PLC routine that interrupting and resuming the execution as well as executing in single-block mode affect all the channels involved.

Tool selection in both channels.

In the master channel, the tool is selected in the block defining the canned cycle or in the previous block. To select the tool in the slave channel, the CNC uses the following criterion.

- There is an active tool in the slave channel.

The CNC uses the active tool if it belongs to the same family as the one programmed in the cycle; if it does not belong to the same family, the CNC selects in the same magazine the first tool of the same family. If there are no tools of the same family in the magazine, the CNC uses the active tool.

- There is an active tool in the slave channel.

The CNC looks in the magazine the first tool of the same family as the one programmed in the cycle. The CNC selects the magazine depending on the slave axis; for channel ·1·, the CNC looks for the tool in magazine ·1·; for channel 2 in magazine 2 and so on.

The tool data of the slave axes must be compatible with those of the tool of the master channel because the CNC calculates all the tool paths of the cycle for the latter tool. The CNC considers that the tool is compatible if it belongs to the same family.

It is up to the user to properly define the tools in the table and make sure that the tools of the same family have the same location code (shape). The rest of the data depends on the machining operation; therefore, the user must verify that the cutter angles, the cutting angle, the edge with and the cutting length are compatible with the programmed tool.

Variables related to dynamic distribution of machining operations.

- | | |
|-------------------|--|
| (V.)A.ACCUDIST.xn | The option for distribution of passes between channels uses this variable. This means that this variable must not be used on the master spindle if the delay has been programmed with "N" or on the axes of the plane if the delay has been programmed with "D". |
|-------------------|--|

10.1 Activating and canceling dynamic machining distribution.

Activating and canceling dynamic machining distribution.

The #DINDIST instruction activates machining dynamic distribution in one or several channels, up to three.

Programming format.

The programming format is: Optional parameters are indicated between angle brackets. Both parameters D and N are optional, but only one of them may be programmed.

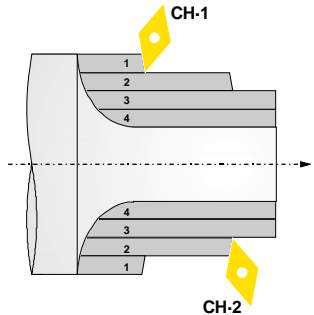
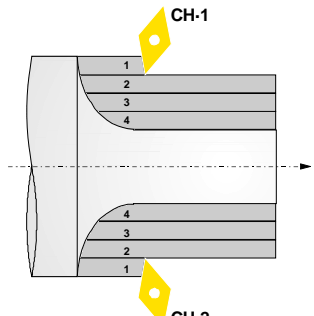
#DINDIST ON [{type} ,{channels} < ,N{turns} > < ,D{distance} >]

Parameter.	Meaning.
{type}	Type of machining. Possible values: 0/1. If -0-, distribution of passes between channels. If -1-, synchronized identical passes.
{channels}	List of channels, separated by commas, between which the machining operations are being distributed (between 1 and 3 channels).
{rotations}	Optional; by default, N1. Minimum number of spindle revolutions as delay between two consecutive passes (only for type -0-).
{distance}	Optional. Delay distance between two consecutive passes (only for type -0-).

```
#DINDIST ON [0,2,3,4,N25]
#DINDIST ON [0,2,3,4,D5]
#DINDIST ON [0,2,N30]
#DINDIST ON [1,2]
```

Type of machining.

This parameter indicates the type of machining distribution.

Type.	Meaning.
0	Distribution of passes between channels. 
1	Synchronized identical passes 

10.

DYNAMIC DISTRIBUTION OF THE MACHINING OPERATIONS
Activating and canceling dynamic machining distribution.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

Canceling dynamic machining distribution.

The #DINDIST instruction cancels machining dynamic distribution in all the channels.

Programming format.

The programming format is:

```
#DINDIST OFF
```

```
#DINDIST OFF
```

10.

DYNAMIC DISTRIBUTION OF THE MACHINING OPERATIONS
Activating and canceling dynamic machining distribution.



CNC 8060
CNC 8065

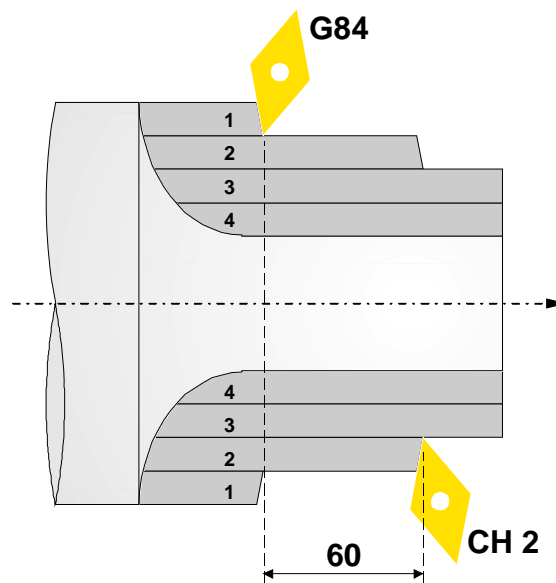
(REF: 1502)

10.1.1 Distribution of passes between channels.

In this type of machining, the master executes the canned cycle and the CNC is in charge of distributing the passes between the rest of the channels. The CNC only distributes the passes of the roughing operation; the finishing pass is executed in the master channel. After the roughing operation is done, the slave channels recover the situation prior to the execution of the cycle.

The execution of the cycle begins in the master channel. Then, the CNC starts up sequentially the slave channels involved, those programmed in the #DINDIST instruction in the order they have been programmed. If a channel is busy, the CNC waits till it becomes available. Between two consecutive passes, the CNC waits for the spindle to rotate the number of turns (parameter N) or the master axis has traveled the distance (parameter D) indicated in the instruction #DINDIST.

```
#DINDIST ON [0,2,N60]
G84 X20 Z-20 Q60 R-40 C5 F0.5 I20 K0
#DINDIST OFF
```



10.

DYNAMIC DISTRIBUTION OF THE MACHINING OPERATIONS
Activating and canceling dynamic machining distribution.

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

10.1.2 Synchronized identical passes

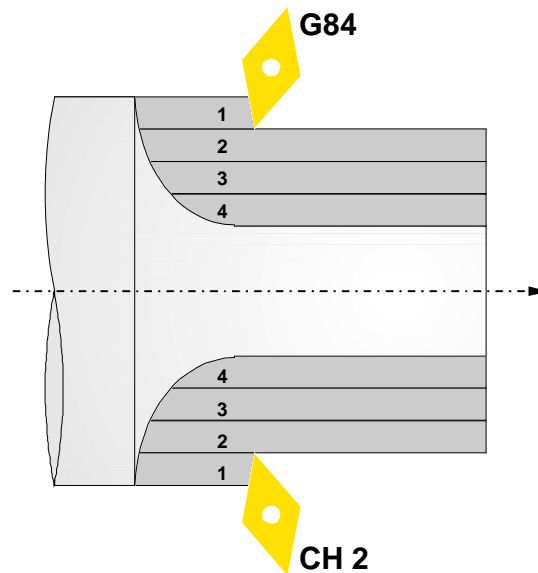
In this type of machining, the master channel executes the canned cycle and the CNC synchronizes the axes of the plane of the slave axes with the axes of the plane of the master channel so all the channels are executed in the same pass. The CNC only synchronizes the passes of the roughing operation; the finishing pass is executed in the master channel. After the roughing operation is done, the CNC releases the axes of the slave channels (#TFOLLOW instruction).

When the execution of the cycle begins, the CNC uses the #TFOLLOW instruction to couple the axes of the plane of the slave channels, the ones programmed in the #TFOLLOW instruction, to the master channel. The CNC couples the axes using a speed synchronization and a gear ratio of 1. If the axes are not available, the CNC will wait until they are before starting the machining operation.

Before executing the machining operation, the CNC moves the axes of the plane of the slave channels to the same position as the axes of the plane of the master channel. From then on, the movements of the master channel will be repeated in synchronism by the slave channels.

The instructions that do not involve motion and must be executed in the slave channels, the CNC will execute them through the #EXBLK instruction. If the slave channels cannot execute them at that moment, the CNC will wait until they can be executed.

```
#DINDIST ON [1,2]
G84 X20 Z-20 Q60 R-40 C5 F0.5 I20 K0
#DINDIST OFF
```



10.

DYNAMIC DISTRIBUTION OF THE MACHINING OPERATIONS
Activating and canceling dynamic machining distribution.



CNC 8060
CNC 8065

(REF: 1502)

STANDARD THREADS

11

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.



CNC 8060
CNC 8065

(REF: 1502)

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.**STANDARD THREADS**

CNC 8060
CNC 8065

(REF: 1502)

11.1 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.

STANDARD THREADS
Regular pitch metric thread — M (S.I.)



CNC 8060
CNC 8065

(REF: 1502)

11.2 Fine pitch metric thread — M (S.I.F.)

The CNC calculates the depth with the formulae:

- Depth in internal threads = $0.5413 \times \text{Pitch}$
- Depth in external threads = $0.6134 \times \text{Pitch}$

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.)



CNC 8060
CNC 8065

(REF: 1502)

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

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.

STANDARD THREADS
Regular pitch Whitworth thread — BSW (W.)

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

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		Edges	Pitch		Depth (mm)	
	(mm)	(inches)		(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



CNC 8060
CNC 8065

(REF: 1502)

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

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

11.

STANDARD THREADS
Regular pitch Unified American thread — UNC (NC, USS)

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

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	Pitch		Depth (mm)	
	(mm)	(inches)		(mm)	(inches)	Internal	External
0,0600	1,5240	0,0600	80	0,3175	0,0125	0,1719	0,1948
0,0730	1,8542	0,0730	72	0,3528	0,0139	0,1910	0,2164
0,0860	2,1844	0,0860	64	0,3969	0,0156	0,2148	0,2435
0,0990	2,5146	0,0990	56	0,4536	0,0179	0,2455	0,2782
0,1120	2,8448	0,1120	48	0,5292	0,0208	0,2865	0,3246
0,1250	3,1750	0,1250	44	0,5773	0,0227	0,3125	0,3541
0,1380	3,5052	0,1380	40	0,6350	0,0250	0,3437	0,3895
0,1640	4,1656	0,1640	36	0,7056	0,0278	0,3819	0,4328
0,1900	4,8260	0,1900	32	0,7937	0,0312	0,4296	0,4869
19/88	5,4864	0,2160	28	0,9071	0,0357	0,4910	0,5564
1/4	6,3500	0,2500	28	0,9071	0,0357	0,4910	0,5564
5/16	7,9375	0,3125	24	1,0583	0,0417	0,5729	0,6492
3/8	9,5250	0,3750	24	1,0583	0,0417	0,5729	0,6492
7/16	11,1125	0,4375	20	1,2700	0,0500	0,6875	0,7790
1/2	12,7000	0,5000	20	1,2700	0,0500	0,6875	0,7790
9/16	14,2875	0,5625	18	1,4111	0,0556	0,7638	0,8656
5/8	15,8750	0,6250	18	1,4111	0,0556	0,7638	0,8656
3/4	19,0500	0,7500	16	1,5875	0,0625	0,8593	0,9738
7/8	22,2250	0,8750	14	1,8143	0,0714	0,9821	1,1129
1	25,4000	1,0000	12	2,1167	0,0833	1,1458	1,2984
1 1/8	28,5750	1,1250	12	2,1167	0,0833	1,1458	1,2984
1 1/4	31,7500	1,2500	12	2,1167	0,0833	1,1458	1,2984
1 1/2	38,1000	1,5000	12	2,1167	0,0833	1,1458	1,2984

11.

STANDARD THREADS
Fine pitch Unified American thread — UNF (NF, SAE)



CNC 8060
CNC 8065

(REF: 1502)

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

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

11.

STANDARD THREADS
Whitworth gas thread — BSP

FAGOR 

CNC 8060
CNC 8065

(REF: 1502)

11.8 Threads for pipes of the oil sector — A.P.I.

Thread		TPI ¹ (Thread per inch)	TPF ² (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.



CNC 8060
CNC 8065

(REF: 1502)

The user and the OEM can add their own canned cycles (user cycles) using the FGUIM application that comes installed on the CNC. The text of the softkey to access these cycles is defined by the creator of the canned cycle. User cycles work in a similar way as Fagor canned cycles.

- It is possible to browse through the cycles
- Pressing the [ESC] key returns to the editor without inserting the cycle.
- Pressing the [INS] inserts the cycle into the program.

User cycles in ISO mode.

The softkey to access the user cycles is on the menu of the EDISIMU mode, and they can even hide the softkeys for the Fagor cycles. In this mode, the cycles help programming, they can only be inserted into the program, the editor does not allow simulating or executing a cycle from its definition screen.

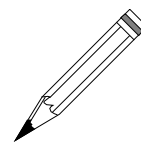
User cycles in conversational mode.

In conversational mode, the user cycle accessing softkey is on the cycles menu. In this mode, the user cycles may be inserted into the program and may be simulated and executed from their definition screen.

Defining user cycles.

The application offers a guided way to define a new component and its softkey menu without having to be familiar with script languages. Follow these steps to create a new canned cycle:

- 1 Access the FGUIM application, installed on the control.
- 2 Create a new user component which will contain the user cycles. Each page of the component will be a canned cycle and will have its own horizontal menu with the softkeys for each cycle.
- 3 Create a softkey for calling the new component with the user cycles, this new component will be integrated into the menu of the editor of the EDISIMU mode.



CNC 8060
CNC 8065

(REF: 1502)