CNCelite

8058/8060
8065

Canned cycles (·T· model).

Ref. 2109

FAGOR

FAGOR AUTOMATION

TRANSLATION OF THE ORIGINAL MANUAL

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

MACHINE SAFETY

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

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

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

HARDWARE EXPANSIONS

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

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

COMPUTER VIRUSES

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

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

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

DUAL-USE PRODUCTS

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

FAGOR AUTOMATION

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

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

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

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

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

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

INDEX

About this manual. 7
About the product. 9
EC Declaration of Conformity, warranty conditions and quality certificates. 15
Safety conditions. 17
Returning conditions. 21
CNC maintenance. 23
New features. 25

CHAPTER 1 MACHINING CANNED CYCLES (ISO)

1.1 General concepts 35
1.2 G81. Turning canned cycle with straight sections 37
1.3 G82. Facing canned cycle with straight sections 40
1.4 G83. Drilling / tapping canned cycle 43
1.5 G84. Turning canned cycle with arcs 46
1.6 G85. Facing canned cycle with arcs 49
1.7 G86. Longitudinal threading canned cycle 52
1.8 G87. Face threading canned cycle 58
1.9 G88. Grooving canned cycle along the X axis 64
1.10 G89. Grooving canned cycle along the Z axis 65
1.11 G66. Pattern repeat canned cycle 66
1.12 G68. Stock removal cycle along X axis 72
1.13 G69. Stock removal canned cycle along Z axis 79
1.14 G160. Drilling / tapping on the face of the part 86
1.15 G161. Drilling / tapping on the side of the part 90
1.16 G162. Slot milling canned cycle along the side of the part 94
1.17 G163. Radial slot milling cycle (on the face of the part) 96

CHAPTER 2 CYCLE EDITOR

2.1 Configuring the cycle editor. 102
2.2 Teach-in mode. 103
2.3 Selecting data, profiles and icons 104
2.4 Simulate a canned cycle. 105

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

3.1 Simple turning cycle. 108
3.1.1 Basic operation. 111
3.2 Turning cycle with vertex rounding. 113
3.2.1 Basic operation. 117
3.3 Simple facing cycle. 119
3.3.1 Basic operation. 122
3.4 Facing cycle with vertex rounding. 124
3.4.1 Basic operation. 128
3.5 Vertex chamfering cycle 130
3.5.1 Basic operation. 135
3.6 "Chamfering between points" cycle. 137
3.6.1 Basic operation. 142
3.7 Vertex chamfering cycle 2 144
3.7.1 Basic operation. 149
3.8 Vertex rounding cycle. 151
3.8.1 Basic operation. 156
3.9 "Rounding between points" cycle. 158
3.9.1 Basic operation. 163

CHAPTER 4 CANNED CYCLES OF THE EDITOR. TAPPING.

4.1 Longitudinal threading cycle. 166
4.1.1 Basic operation. 170
4.2 Taper threading cycle. 171
4.2.1 Basic operation. 176
4.3 Face threading cycle. 177
4.3.1 Basic operation. 181
4.4 Thread repair cycle. 182

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

4.4.1 Basic operation. 188
4.5 Multi-entry threading cycle. 189
4.5.1 Basic operation. 194

CHAPTER 5 CANNED CYCLES OF THE EDITOR. GROOVING.

5.1 Calibrate the grooving tool (location code, shape). 196
5.1.1 Outside longitudinal grooving. 196
5.1.2 Inside longitudinal grooving. 197
5.1.3 Face grooving. 198
5.2 Simple longitudinal grooving cycle. 199
5.2.1 Basic operation 203
5.3 Simple face grooving cycle 205
5.3.1 Basic operation 209
5.4 Inclined longitudinal grooving cycle. 211
5.4.1 Basic operation 216
5.5 Inclined face grooving cycle. 218
5.5.1 Basic operation 223
5.6 Cut-off cycle. 225
5.6.1 Basic operation 228

CHAPTER 6 CANNED CYCLES OF THE EDITOR. PROFILES.

6.1 Point-to-point turning cycle. 230
6.1.1 Basic operation. 236
6.1.2 Programming example. 237
6.2 Profile turning cycle. 238
6.2.1 Basic operation. 245
6.2.2 Programming example. 246
6.3 ZC/YZ plane profiling cycle. 252
6.3.1 Basic operation. 256
6.4 Rectangular pocket cycle in the ZC/YZ plane. 257
6.4.1 Basic operation. 262
6.5 Circular pocket cycle in the ZC/YZ plane. 263
6.5.1 Basic operation. 267
6.6 Circular pre-emptied pocket cycle in the ZC/YZ plane. 268
6.6.1 Basic operation. 272
6.7 2D profile pocket cycle in the ZC/YZ plane. 273
6.7.1 Basic operation. 278
6.8 XC/XY plane profiling cycle. 279
6.8.1 Basic operation. 283
6.9 Rectangular pocket cycle in the XC/XY plane. 284
6.9.1 Basic operation. 289
6.10 Circular pocket cycle in the XC/XY plane. 290
6.10.1 Basic operation. 294
6.11 Circular pre-emptied pocket in the XC/XY plane. 295
6.11.1 Basic operation. 299
6.12 2D profile pocket cycle in the XC/XY plane. 300
6.12.1 Basic operation. 305

CHAPTER 7 CANNED CYCLES OF THE EDITOR. LONGITUDINAL MACHINING.

7.1 Center punching. 308
7.1.1 Basic operation. 311
7.2 Drilling. 312
7.2.1 Basic operation. 315
7.3 Tapping. 316
7.3.1 Basic operation. 318
7.4 Multiple center-punching. 319
7.4.1 Basic operation. 322
7.5 Multiple drilling. 323
7.5.1 Basic operation. 327
7.6 Multiple tapping. 328
7.6.1 Basic operation. 331
7.7 Multiple reaming. 332
7.7.1 Basic operation. 335
7.8 Multiple boring. 336
7.8.1 Basic operation. 340
7.9 Multiple bore milling. 341
7.9.1 Basic operation. 344
7.10 Multiple thread milling. 345
7.10.1 Basic operation. 349
7.11 Multiple slot milling. 350
7.11.1 Basic operation. 354

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

CHAPTER 8 CANNED CYCLES OF THE EDITOR. POSITIONING.

8.1 Positioning cycle. ... 356
8.2 Positioning cycle with M functions. ... 358

CHAPTER 9 CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.

9.1 Multiple machining in straight line. ... 362
9.1.1 Programming example. ... 363
9.2 Multiple machining in arc pattern. ... 364
9.2.1 Programming example. ... 365
9.3 Multiple machining in rectangular pattern. ... 366
9.3.1 Programming example. ... 367
9.4 Multiple machining in grid pattern. ... 368
9.4.1 Programming example. ... 369
9.5 Random multiple machining. ... 370
9.5.1 Programming example. ... 370

CHAPTER 10 DYNAMIC DISTRIBUTION OF THE MACHINING OPERATIONS BETWEEN CHANNELS.

10.1 Activating and canceling dynamic machining distribution. ... 373
10.1.1 Distribution of passes between channels. ... 375
10.1.2 Synchronized identical passes ... 376

CHAPTER 11 STANDARD THREADS

11.1 Regular pitch metric thread — M (S.I.) ... 378
11.2 Fine pitch metric thread — M (S.I.F.) ... 379
11.3 Regular pitch Whitworth thread — BSW (W.) ... 380
11.4 Fin pitch Whitworth thread — BSF ... 381
11.5 Regular pitch Unified American thread — UNC (NC, USS) ... 382
11.6 Fine pitch Unified American thread — UNF (NF, SAE) ... 383
11.7 Whitworth gas thread — BSP ... 384
11.8 Threads for pipes of the oil sector — A.P.I. ... 385

CHAPTER 12 USER CYCLES.

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

ABOUT THIS MANUAL.

Title. Canned cycles (·T· model).

Models. CNCelite
8058 8060
8065

Type of documentation. End user manual. This manual describes the machining canned cycles, ISO and conversational, for the T model.

Remarks.

Always use the manual reference associated with the software version or a later manual reference. You can download the latest manual reference from the download section on our website.

Limitations.

The availability of some of the features described in this manual are dependent on the acquired software options. Moreover, the machine manufacturer (OEM) customizes the CNC performance of each machine using the machine parameters and the PLC. Because of this, the manual may describe features that are not available for the CNC or the machine. Consult the machine manufacturer for the available features.

Electronic document. man_elite_58_60_65_t_cyc.pdf. Manual available from the download section of our website.

Language. English [EN]. Refer to our website, download area, the languages available for each manual.

Date of publication. September, 2021

Manual reference Ref. 2109

Associated version. v2.00.01

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

Trademarks. This manual may contain third-party trademarks or trade names, however, they do not have them associated ® or ™ symbols. All the trade marks appearing in the manual belong to the corresponding owners. The use of these marks by third parties for their own purpose could violate the rights of the owners.

Website / Email. http://www.fagorautomation.com

Email: info@fagorautomation.es

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

FAGOR AUTOMATION

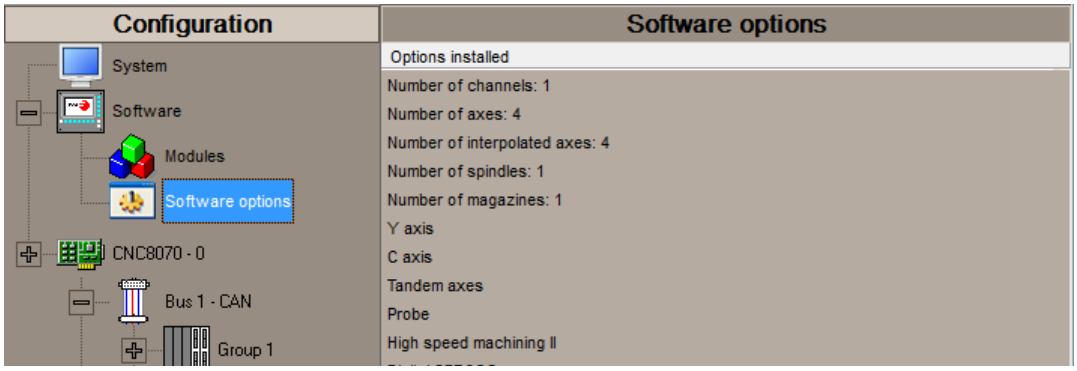
CNCelite
8058 8060
8065

REF. 2109

About the product.

SOFTWARE OPTIONS.

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



Software option	Description.
SOFT ADDIT AXES	Option to add axes to the default configuration.
SOFT ADDIT SPINDLES	Option to add spindles to the default configuration.
SOFT ADDIT TOOL MAGAZ	Option to add magazines to the default configuration.
SOFT ADDIT CHANNELS	Option to add channels to the default configuration.
SOFT 4 AXES INTERPOLATION LIMIT	Limited to 4 interpolated axes.
SOFT DIGITAL SERCOS	Option for a Sercos digital bus.
SOFT THIRD PARTY DRIVES	Option to use EtherCAT third party drives.
SOFT THIRD PARTY I/Os	Option to use third party I/O modules.
SOFT OPEN SYSTEM	Option for open systems. The CNC is a closed system that offers all the features needed to machine parts. Nevertheless, at times there are some customers who use third-party applications to take measurements, perform statistics or other tasks apart from machining a part. This feature must be active when installing this type of application, even if they are Office files. Once the application has been installed, it is recommended to close the CNC in order to prevent the operators from installing other kinds of applications that could slow the system down and affect the machining operations.

Software option	Description.
SOFT i4.0 CONNECTIVITY PACK	Options for Industry 4.0 connectivity. This option provides various data exchange standards (for example, OPC UA), which allows the CNC (and therefore the machine tool) to be integrated into a data acquisition network or into a MES or SCADA system.
SOFT EDIT/SIMUL	Option to enable edisimu mode (edition and simulation) on the CNC, which can edit, modify and simulate part programs.
SOFT DUAL-PURPOSE (M-T)	Option to enable the dual-purpose machine, which allows milling and turning cycles. On Y-axis lathes, this option allows for pockets, bosses and even irregular pockets with islands to be made during milling cycles. On a C-axis mill, this option allows turning cycles to be used.
SOFT TOOL RADIUS COMP	Option to enable radius compensation. This compensation programs the contour to be machined based on part dimensions without taking into account the dimensions of the tool that will be used later on. This avoids having to calculate and define the tool paths based on the tool radius.
SOFT PROFILE EDITOR	Option to enable the profile editor in edisimu mode and in the cycle editor. This editor can graphically, and in a guided way, define rectangular, circular profiles or any profile made up of straight and circular sections an it can also import dxf files. After defining the profile, the CNC generates the required blocks and add them to the program.
SOFT HD GRAPHICS In a multi-channel system, this feature requires the MP-PLUS (83700201) processor.	High definition solid 3D graphics for the execution and simulation of part-programs and canned cycles of the editor. During machining, the HD graphics display, in real time, the tool removing the material from the part, allowing the condition of the part to be seen at all times. These graphics are required for the collision control (FCAS).
SOFT IIP CONVERSATIONAL	The IIP (Interactive Icon-based Pages) mode, or conversational mode, works with the CNC in a graphical and guided way based on predefined cycles. There is no need to work with part programs, have any previous programming knowledge or be familiar with Fagor CNCs. Working in conversational mode is easier than in ISO mode, as it ensures proper data entry and minimizes the number of operations to be defined.
SOFT RTCP This feature requires the MP-PLUS (83700201) processor.	Option to enable dynamic RTCP (Rotating Tool Center Point) required to machine with 4, 5 and 6 axis kinematics; for example, angular and orthogonal spindles, tilting tables, etc. The RTCP orientation of the tool may be changed without modifying the position occupied by the tool tip on the part.
SOFT C AXIS	Option to enable C-axis kinematics and associated canned cycles. The machine parameters of each axis or spindle indicate whether it can operate as a C axis or not. For this reason, it is not necessary to add specific axes to the configuration.
SOFT Y AXIS	Option to enable lathe Y-axis kinematics and associated canned cycles.
SOFT TANDEM AXES	Option to enable tandem axle control. A tandem axis consists of two motors mechanically coupled to each other forming a single transmission system (axis or spindle). A tandem axis helps provide the necessary torque to move an axis when a single motor is not capable of supplying enough torque to do it. When activating this feature, it should be kept in mind that for each tandem axis of the machine, another axis must be added to the entire configuration. For example, on a large 3-axis lathe (X Z and tailstock), if the tailstock is a tandem axis, the final purchase order for the machine must indicate 4 axes.

Software option	Description.
SOFT SYNCHRONISM	Option to enable the synchronization of paired axes and spindles, in speed or position, and through a given ratio.
SOFT KINEMATIC CALIBRATION	Option to enable tool calibration. For the first time, this kinematics calibration allows for the kinematics offsets to be calculated using various approximate data and, also, from time to time to correct any possible deviations caused by day-to-day machining operations.
SOFT 60 HSSA I MACHINING SYSTEM	Option to enable the HSSA-I (High Speed Surface Accuracy) algorithm for high speed machining (HSC). This new HSSA algorithm allows for high speed machining optimization, where higher cutting speeds, smoother contours, a better surface finishing and greater precision are achieved.
SOFT HSSA II MACHINING SYSTEM	Option to enable the HSSA-II (High Speed Surface Accuracy) algorithm for high speed machining (HSC). This new HSSA algorithm allows for high speed machining optimization, where higher cutting speeds, smoother contours, a better surface finishing and greater precision are achieved. The HSSA-II algorithm has the following advantages compared to the HSSA-I algorithm. • Advanced algorithm for point preprocessing in real time. • Extended curvature algorithm with dynamic limitations. Improved acceleration and jerk control. • Greater number of pre-processed points. • Filters to smooth out the dynamic machine behavior.
SOFT TANGENTIAL CONTROL	Option to enable tangential control. "Tangential Control" maintains a rotary axis always in the same orientation with respect to the programmed tool path. The machining path is defined on the axes of the active plane and the CNC maintains the orientation of the rotary axis along the entire tool path.
SOFT PROBE	Option to enable functions G100, G103 and G104 (for probe movements) and probe canned cycles (which help to measure part surfaces and to calibrate tools). For the laser model, it only activates the non-cycle function G100. The CNC may have two probes; usually a tabletop probe to calibrate tools and a measuring probe to measure the part.
SOFT FVC STANDARD **SOFT FVC UP TO 10m3** **SOFT FVC MORE TO 10m3**	Options to enable volumetric compensation. The precision of the parts is limited by the machine manufacturing tolerances, wear, the effect of temperature, etc., especially on 5-axis machines. Volumetric compensation corrects these geometric errors to a larger extent, thus improving the precision of the positioning. The volume to be compensated is defined by a point cloud and for each point the error to be corrected is measured. When mapping the total work volume of the machine, the CNC knows the exact position of the tool at all times. There are 3 options, which depend on the size of the machine. • FVC STANDARD: Compensation for 15625 points (maximum 1000 points per axis). Quick calibration (time), but less precise than the other two, but sufficient for the desired tolerances. • FVC UP TO 10m3: Volume compensation up to 10 m³. More accurate than FVC STANDARD, but requires a more accurate calibration using a Tracer or Tracker laser). • FVC MORE TO 10m3: Volume compensation greater than 10 m³. More accurate than FVC STANDARD, but requires a more accurate calibration using a Tracer or Tracker laser.

Software option	Description.
SOFT CONV USER CYCLES	Option to enable user conversational cycles. The user and the OEM can add their own canned cycles (user cycles) using the FGUIM application that comes installed on the CNC. The application offers a guided way to define a new component and its softkey menu without having to be familiar with script languages. User cycles work in a similar way as Fagor canned cycles.
SOFT PROGTL3	Option to enable the ProGTL3 programming language (ISO language extension), allowing profiles to be programmed using a geometric language and without the need to use an external CAD system. This language can program lines and circles where the end point is defined as the intersection of 2 other sections, pockets, ruled surfaces, etc.
SOFT PPTRANS	Option to enable the program translator, which can convert programs written in other languages to Fagor ISO code.
SOFT DMC	Option to enable the DMC (Dynamic Machining Control). DMC adapts the feedrate during machining to maintain the cutting power as close as possible to ideal machining conditions.
SOFT FMC	Option to enable the FMC (Fagor Machining Calculator). The FMC application consists of a database of materials to be machined and machining operations, with an interface to choose suitable cutting conditions for these operations.
SOFT FFC	Option to enable the FFC (Fagor Feed Control). During the execution of a canned cycle of the editor, the FFC function makes it possible to replace the feedrate and speed programmed in the cycle with the active values of the execution, which are acted upon by the feed override and speed override.
SOFT 60/65/70 OPERATING TERMS	Option to enable a temporary user license for the CNC, which is valid until the date set by the OEM. While the license is valid, the CNC will be fully operational (according to the purchased software options).
SOFT FCAS	Option to enable the FCAS (Fagor Collision Avoidance System). The FCAS option, within the system limitations, monitors the automatic, MDI/MDA, manual and tool inspection movements in real time, so as to avoid collisions between the tool and the machine. The FCAS option requires that the HD graphics to be active and that there is a defined a model configuration of the machine adjusted to reality (.xca file), which includes all its moving parts.
SOFT GENERATE ISO CODE	ISO generation converts canned cycles, calls to subroutines, loops, etc. into their equivalent ISO code (G, F, S, etc functions), so the user can modify it and adapt it to his needs (eliminate unwanted movements, etc.). The CNC generates the new ISO code while simulating the program, either from the DISIMU mode or from the conversational mode.
SOFT PWM CONTROL	Option to enable PWM (Pulse - Width Modulation) control on laser machines. This feature is essential for cutting very thick sheets, where the CNC must create a series of PWM pulses to control laser power when drilling the initial point. This function is only available for Sercos bus control systems and must also use one of the two fast digital outputs available from the central unit.
SOFT GAP CONTROL	Option to enable gap control, which makes it possible to set a fixed distance between the laser nozzle and the sheet surface with the use of a sensor. The CNC compensates the difference between the distance measured by the sensor and the programmed distance with additional movements on the axis programmed for the gap.

Software option	Description.
SOFT MANUAL NESTING	Option to enable nesting in the automatic option. Nesting consists of creating a pattern on the sheet material using previously defined figures (in dxf, dwg or parametric files), so as to use most of the sheet as possible. Once the pattern has been defined, the CNC creates a program. During manual nesting, the operator distributes the parts on top of the sheet material.
SOFT AUTO NESTING	Option to enable nesting in the automatic option. Nesting consists of creating a pattern on the sheet material using previously defined figures (in dxf, dwg or parametric files), so as to use most of the sheet as possible. Once the pattern has been defined, the CNC creates a program. During automatic nesting, the application distributes the figures on the sheet material and optimizes the spaces.
SOFT DRILL CYCL OL	Option to enable ISO drilling cycles (G80, G81, G82, G83).

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

EC DECLARATION OF CONFORMITY, WARRANTY CONDITIONS AND QUALITY CERTIFICATES.

EC-DECLARATION OF CONFORMITY

The declaration of conformity is available from the downloads section of the Fagor Automation corporate website.

https://www.fagorautomation.com/en/downloads/

Type of file: Declaration of conformity.

NRTL CERTIFICATE FOR USA AND CANADA

The quality certificates are available from ·the company· label on the Fagor Automation corporate website.

https://www.fagorautomation.com/en/sections/quality/

WARRANTY TERMS

The sales and warranty conditions are available from the downloads section of the Fagor Automation corporate website.

https://www.fagorautomation.com/en/downloads/

Type of file: General sales - warranty conditions.

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

SAFETY CONDITIONS.

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

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

PRECAUTIONS BEFORE CLEANING THE UNIT

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

PRECAUTIONS DURING REPAIRS

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

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

PRECAUTIONS AGAINST PERSONAL HARM

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

PRECAUTIONS AGAINST DAMAGE TO THE PRODUCT

Work environment.

This unit is ready to be used in industrial environments complying with the directives and regulations effective in the European Community.

Fagor Automation shall not be held responsible for any damage suffered or caused by the CNC when installed in other environments (residential, homes, etc.).

Install this unit in the proper place.

It is recommended, whenever possible, to install the CNC away from coolants, chemical product, blows, etc. that could damage it.

This unit meets the European directives on electromagnetic compatibility. Nevertheless, it is recommended to keep it away from sources of electromagnetic disturbance such as:

- Powerful loads connected to the same mains as the unit.
- Nearby portable transmitters (radio-telephones, Ham radio transmitters).
- Nearby radio / TC transmitters.
- Nearby arc welding machines.
- Nearby high voltage lines.

Enclosures.

It is up to the manufacturer to guarantee that the enclosure where the unit has been installed meets all the relevant directives of the European Union.

Avoid disturbances coming from the machine.

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

Use the proper power supply.

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

Connecting the power supply to ground.

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

Analog inputs and outputs connection.

Use shielded cables connecting all their meshes to the corresponding pin.

Ambient conditions.

Maintain the CNC within the recommended temperature range, both when running and not running. See the corresponding chapter in the hardware manual.

Central unit enclosure.

To maintain the right ambient conditions in the enclosure of the central unit, it must meet the requirements indicated by Fagor. See the corresponding chapter in the hardware manual.

Power switch.

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

SAFETY SYMBOLS

Symbols that may appear in the manual.

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

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

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

Information symbol.
This symbol indicates notes, warnings and advises.

Symbol for additional documentation.
This symbol indicates that there is another document with more detailed and specific information.

Symbols that the product may carry.

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

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

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

RETURNING CONDITIONS.

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

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

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

CNC MAINTENANCE.

CLEANING

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

To clean the operator panel and the monitor, a smooth cloth should be used which has been dipped into de-ionized water and /or non abrasive dish-washer soap (liquid, never powder) or 75º alcohol. Never use air compressed at high pressure to clean the unit because it could cause the accumulation of electrostatic charges that could result in electrostatic shocks.

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

PRECAUTIONS BEFORE CLEANING THE UNIT

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

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

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

NEW FEATURES.

Manual reference:	Ref. 2109
Date of publication:	September, 2021
Associated software:	v2.00.01

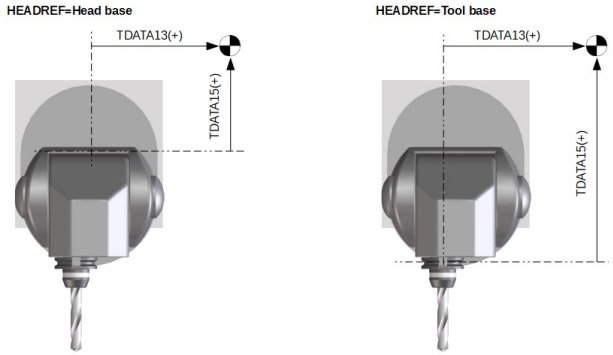
Below is a list of the features added in this software version and the manuals that describe them.

List of features.	Manual
OPC-UA Protocol	
New laser model. • CUT_VIEW mode for previewing the part to be cut as well as the piercing points. • Technological tables (piercing, cutting and generic) to define the necessary parameters for cutting (power, speed, etc.). • Material files relating the material characteristics (type, thickness, etc.) to the technology tables. • Manual and automatic nesting. • Radius compensation. • Subroutines to perform the initial and final laser operations. • Change from the part-programme of the status of PLC marks, logic inputs and outputs. • Activation of a material from the part-programme. • Activation of a generic technology table from the part-programme. • Piercing and cutting with associated subroutines. • The CNC makes it possible to run a program with microjoints that is generated without them. • Control of laser power, PWM duty and/or PWM frequency depending on the feed rate. • Gap control. • Leapfrog. • CBDS (Constant Beam Delivery System).	[LSR]
Integration of Lantek Expert Inside software for laser models. From figures drawn in dxf, dwg and parametric files, this software allows these to be distributed on a sheet to optimize their use (nesting), define the input/output strategy (lead-in/lead-out), etc., to ultimately create a part-program in Fagor language. Fagor provides two types of nesting, which are available via software options; manual nesting (the user places the pieces on the sheet as required) and automatic nesting (the software places the pieces on the sheet based on their configuration parameters).	[LSR]
New software option "SOFT 60 Y AXIS". Option to enable lathe Y-axis kinematics and associated canned cycles.	
New software option "SOFT FVC STANDARD". Option to enable standard volumetric compensation. Compensation for 15625 points (maximum 1000 points per axis). Quick calibration (time), but less precise than the other two, but sufficient for the desired tolerances.	
New software option "SOFT GAP CONTROL". Option to enable gap control, which makes it possible to set a fixed distance between the laser nozzle and the sheet surface with the use of a sensor.	
New software option "SOFT DMC". Option to enable the DMC (Dynamic Machining Control). DMC adapts the feedrate during machining to maintain the cutting power as close as possible to ideal machining conditions.	
New software option "SOFT FMC". Option to enable the FMC (Fagor Machining Calculator). The FMC application consists of a database of materials to be machined and machining operations, with an interface to choose suitable cutting conditions for these operations.	
New software option "SOFT FFC". Option to enable the FFC (Fagor Feed Control). During the execution of a canned cycle of the editor, the FFC function makes it possible to replace the feedrate and speed programmed in the cycle with the active values of the execution, which are acted upon by the feed override and speed override.	

[CYC-M]..... Machining canned cycles (·M· model).
[CYC-T]..... Machining canned cycles (·T· model).
[EXA-T]..... Examples manual (·T· model).
[FGUIM]..... Customizing guide (FGUIM & API).
[INST]..... Installation manual.
[LSR]..... Operating and programming manual (·L· model).
[OPT] ..... Operating manual.
[OPT-MC]..... Operating manual (MC option).
[OPT-TC]..... Operating manual (TC option).
[PRG]..... Programming manual.
[PROGTL3]..... ProGTL3 language.
[VAR]..... CNC variables.

List of features.	Manual
New software option "SOFT FCAS". Option to enable the FCAS (Fagor Collision Avoidance System). The FCAS option, within the system limitations, monitors the automatic, MDI/MDA, manual and tool inspection movements in real time, so as to avoid collisions between the tool and the machine.	
New software option "SOFT 60/65/70 OPERATING TERMS". Option to enable a temporary user license for the CNC, which is valid until the date set by the OEM.	
New software option "SOFT MANUAL NESTING". Option to enable nesting in the automatic option. Nesting consists of creating a pattern on the sheet material using previously defined figures (in dxf, dwg or parametric files), so as to use most of the sheet as possible. During manual nesting, the operator distributes the figures on the sheet.	
New software option "SOFT AUTO NESTING". Option to enable nesting in the automatic option. Nesting consists of creating a pattern on the sheet material using previously defined figures (in dxf, dwg or parametric files), so as to use most of the sheet as possible. During automatic nesting, the application distributes the figures on the sheet material and optimizes the spaces.	
This version must be installed in administrator mode, so that the process properly registers the new FCombo.ocx control.	
New languages for the HMI. • Polish and Turkish. - Machine parameters: LANGUAGE	[INST]
HMI. • Define the initial position of the interface on the screen. - Machine parameter: POSX, POSY • The interface can be moved by clicking with the mouse on the icon showing the program status of the active channel. Double click on this icon and the interface is positioned in the upper left corner (POSX=0 POSY=0). • Define the size of the interface. - Machine parameter: RESOLUTION, WIDTH, HEIGHT • Customize the number of softkeys and their size; between 7 and 12 horizontal softkeys and between 5 and 12 vertical softkeys. - Machine parameter: HMENUNSOFTKEY, HMENUNSOFTKEY, SOFTKEYMETRICS	[INST]
Basic volumetric compensation. • Compensation for 15625 points (maximum 1000 points per axis). Quick to calibrate (time) but less accurate than the other two (corrects fewer error components), but sufficient for the desired tolerances. - Machine parameter: VMOVAXIS1, VMOVAXIS2, VMOVAXIS3 NPOINTS1, NPOINTS2, NPOINTS3 INIPOSAX1, INIPOSAX2, INIPOSAX3 INCREAX1, INCREAX2, INCREAX3	[INST]
Volumetric compensation. • Definition of rotary axes as axes of motion and compensation. - Machine parameter: VMOVAXIS, VCOMPAXIS • Definition of 1 or 2-axis offsets. - Machine parameter: VMOVAXIS, VCOMPAXIS	[INST]
Number the digital inputs of the CANopen bus based on logic blocks. - Machine parameter: NDIMOD, DIMODADDR, NODE, BLOCK, ADDRESS, NDI	[INST]
Number the digital outputs of the CANopen bus based on logic blocks. - Machine parameter: NDOMOD, DOMODADDR, NODE, BLOCK, ADDRESS, NDO	[INST]
Number of special analog inputs (non-voltage). - Machine parameters: NSAI, ADDRESS, FACTOR	[INST]
PLC program. • Increases the limit of PDEF symbols. The PDEF directing instruction can define up to 200 symbols and can be used in the PLC program, a PLC subroutine in C language, part program (using variables) or external application. • Subroutines in C language support DEF/PDEF symbols. Every time the PLC compiles the program, it generates the file plc_pdef.h with the #define of all the symbols defined in the PLC program. The file is saved in the folder ..\MTB\PLC\Project.	[INST]
Asynchronous magazine management. • New TMOPERATION=12 operation for asynchronous magazine. Leave the spindle tool in the magazine and take a tool from the same magazine (as TMOPERATION=5). It is an optimised operation, valid when both tools are special and can collide in the arm or when it is not desirable for them to coincide in the changer arm. - PLC registration: TMPOPERATION	[INST]

[CYC-M]..... Machining canned cycles (·M· model).
[CYC-T]..... Machining canned cycles (·T· model).
[EXA-T]..... Examples manual (·T· model).
[FGUIM]..... Customizing guide (FGUIM & API).
[INST]..... Installation manual.
[LSR]..... Operating and programming manual (·L· model).
[OPT]..... Operating manual.
[OPT-MC]..... Operating manual (MC option).
[OPT-TC]..... Operating manual (TC option).
[PRG]..... Programming manual.
[PROGTL3]..... ProGTL3 language.
[VAR]..... CNC variables.

List of features.	Manual
Disable the local feedback input alarms. - PLC mark: LCOUNTALARMOFF1	[INST]
Observe the G00 feedrate during synchronized switching. - Machine parameter: SWTG0FEED	[INST]
Configure the axes of a mill to execute the rotary axis cycles. - Machine parameter: LONGAXIS, FACEAXIS, CAXIS	[INST]
Calculate the maximum scanning feed rate. If PROBEFEED is set to 0, the CNC calculates and uses the maximum possible value of this parameter. - Machine parameter: PROBEFEED	[INST]
Defining kinematics. • The kinematics table only indicates the parameters that affect a kinematics type. • Point of origin of TDATA parameters; "Tool base", "Head base" and "Auto tool base". With the new "Auto Tool Base" value, on activating the CNC kinematics, calculate the value of TDATA13/TDATA14/TDATA15 (non-vectorial kinematics) or TDATA1/TDATA2/TDATA3 (vectorial kinematics). - Machine parameter: HEADREF 	[INST]
If a movement smaller than the INPOSW parameter is programmed on a dead axis, the CNC does not enable or move the axis. - Machine parameters: INPOSW	[INST]
Changes to read-only MTBPAR parameters using variables in Setup mode. - Machine parameters: MTBPAR - Variables: (V.)MTB.P, (V.)MTB.PF	[INST]
HD graphics. • Configure the HD graphics for the rotary axis cycles (·M· model). • If the HD graphics work with the tool tip coordinates, then they respond to the parameters FACEAXIS and LONGAXIS. If the HD graphics work with the tool base coordinates, then they respond to the parameters FACEAXIS and LONGAXIS. - Machine parameters: FACEAXIS, LONGAXIS	[INST]
Non-modal G02 and G03 functions. - Machine parameter: G2G3MODAL - Function: G2, G3	[INST] [PRG]
Non-modal function G00. - Machine parameter: G0MODAL, IMOVE - Function: G0.	[INST] [PRG]
New axis names; E, E1..E9. • Values of the variable (V.)G.AXISNAMEn for the E axis; E=110, E1=111, etc. • Values of the variable (V.)G.GAXISNAMEn for the E axis; E=110, E1=111, etc.	[INST] [PRG] [VAR]
If the execution of an interruption subroutine (marks INT1/INT4 of the PLC) ends with M30, then the program execution also ends. If the execution of an interruption subroutine ends with #RET, then the program execution continues. - PLC marks: INT1 / INT4	[INST] [PRG]
In programs that include #SET AX (change configuration of axes) and #DEFGRAF (load graphic configuration), the CNC will allow the program to be simulated after executing a #SET AX in MDI different to that of the program.	
New radius compensation algorithm, optimized to solve stepped profiles.	
Subroutine associated with the reset. - Subroutine: PROGRAM_RESET	[INST] [PRG]
Being the CNC in SETUP mode, the OEM subroutines with an fst extension will be loaded into RAM memory during the preparation of the blocks.	[INST] [PRG]
OEM subroutines for kinematics calibration can execute motions in two axes at the same time.	

[CYC-M]..... Machining canned cycles (·M· model).
[CYC-T]..... Machining canned cycles (·T· model).
[EXA-T]..... Examples manual (·T· model).
[FGUIM]..... Customizing guide (FGUIM & API).
[INST]..... Installation manual.
[LSR]..... Operating and programming manual (·L· model).
[OPT] ..... Operating manual.
[OPT-MC]..... Operating manual (MC option).
[OPT-TC]..... Operating manual (TC option).
[PRG]..... Programming manual.
[PROGTL3]..... ProGTL3 language.
[VAR]..... CNC variables.

List of features.	Manual
New search criteria for help files associated with the subroutines. - File: pcall.txt - Archivo: subroutine_name.txt subroutine_name.bmp	[INST] [PRG]
The subroutines associated with functions G500-599 may also have help files which are displayed during editing. - File: G500.txt, G501.txt, etc. G500.bmp, G501.bmp, etc.	[INST] [PRG]
The subroutines associated with functions G8000-8999 may also have help files which are displayed during editing. - File: G8000.txt, G8001.txt, etc. G8000.bmp, G8001.bmp, etc.	[INST] [PRG]
User subroutines (G500-G599) do not alter the modal/non-modal status of other possible active subroutines (#MCALL). - Function: G500, G501, etc. - Statement: #MCALL	[PRG]
User subroutines (G8000-G8099) do not alter the modal/non-modal status of other possible active subroutines (#MCALL). - Function: G8000, G8001, etc. - Statement: #MCALL	[PRG]
Calls to subroutines with parameter initialization allow the programming of 32 additional parameters (P26 to P57), which can also be defined as "D0= " to "D31=", so that "D0=" is equal to P26, "D1=" to P27 and so forth. - Function: G500, G501, etc. G180, G181, etc. G380, G381, etc. - Statement: #PCALL, #MCALL	[PRG]
Work zones. The command "E" must always be placed after the function G122; otherwise the CNC will interpret it as the name of an axis. - Function: G122.	[PRG]
Scaling factor per axis. - Function: G72.	[PRG]
Defining the path and the name of a program in a macro. - Statement: #DEF.	[PRG]
CNC messages. - Statement: #MSG • Format identifiers (%D, %i, %u, etc). • Define messages using a number. • The OEM or the user can define a text group for the cncMsg.txt file.	[PRG]
Errors and warnings. - Statement: #ERROR, #WARNING, #WARNINGSTOP • Format identifiers (%D, %i, %u, etc). • Change location of cncError.txt file. • Programming a warning with a value of 0 clears all the warnings that are being displayed.	[PRG]
Change the HMI variables from the workpiece programme. - Statement: #MSGVAR. - Variable: (V.)G.CNCMSG[num]	[PRG]
Table+spindle kinematics (type 52). • The CNC does not take into account the axes of the table that are not programmed. - Statement: #SELECT ORI. • When positioning the tool, take into account the position of the table. - Statement: #SELECT ORI. ALL command. • When activating the RTCP, define the part of the kinematics to be used (table or head) and the type of RTCP (static or dynamic). - Statement: #RTCP. HEAD and TABLE commands. • Activation of the RTCP according to the values defined in the machine parameters (TDATA). - Statement: #RTCP. CLEAR command. • Rotate the part coordinate system when rotating the table. - Statement: #RTCP. COROT command. • Working mode with RTCP in spindle+table kinematics. - Variable: (V.)G.RTCPMODE	[PRG]
New mode to define an inclined plane (MODE 7), rotating first on the third axis of the part, then on the second axis of the new plane and finally on the first axis of the new plane. - Statement: #CS / #ACS. MODE 7.	[PRG]
Management of discontinuities in the orientation of rotary axes. Orient the tool in the programmed direction of the main or secondary rotary axis. - Statement: #DEFROT. DIRF, DIRS commands.	[PRG]

[CYC-M]..... Machining canned cycles (·M· model).
[CYC-T]..... Machining canned cycles (·T· model).
[EXA-T]..... Examples manual (·T· model).
[FGUIM]..... Customizing guide (FGUIM & API).
[INST]..... Installation manual.
[LSR]..... Operating and programming manual (·L· model).
[OPT]..... Operating manual.
[OPT-MC]..... Operating manual (MC option).
[OPT-TC]..... Operating manual (TC option).
[PRG]..... Programming manual.
[PROGTL3]..... ProGTL3 language.
[VAR]..... CNC variables.

List of features.	Manual
Opening and writing files. • Does not prompt an error for operations using #OPEN, #WRITE y #CLOSE. - Statement: #OPEN. MUTED command. • Verify if the file exists. - Statement: #OPEN. R(READ) command.	[PRG]
If the CNC reaches the end of a program without an M02/M30, then it ends the execution without prompting a warning.	[PRG]
The #CYL instruction admits programming of the CZ plane (#CYL[C,Z]).	
Activate a kinematic displaying the tool tip dimensions. - Statement: #KIN ID. TIP command. - Variable: (V.)G.KINIDMODE	[PRG] [VAR]
The user variables have reading permission via the interface. - Variable: (V.)P.var_name (V.)P.var_name[nb] (V.)P.var_name (V.)P.var_name[nb]	[PRG] [VAR]
Array user variables. - Variable: (V.)P.var_name[nb] (V.)P.var_name[nb]	[PRG] [VAR]
Correcting the values indicated by the hardware. - Variable: (V.)G.HARDTYPE	[VAR]
Status of the power-up and power-down process. - Variable: (V.)E.CNCLOGST	[VAR]
Identifying the installation folder. - Variable: (V.)G.FOLDERID	[VAR]
Identification of the language selected in the CNC. - Variable: (V.)G.LANGUAGEID	[VAR]
Monitoring the precision of rigid tapping. - Variable: (V.)G.RIGIER	[VAR]
Error code for #OPEN, #WRITE or #CLOSE statements. - Variable: (V.)G.FILEERRNO	[VAR]
From the part-program (#MSGVAR) change the HMI variables. - Variable: (V.)G.CNCMSG[num]	[VAR]
Axis logic number, based on its order in the channel. - Variable: (V.)G.NLOGAXIS1 (V.)G.NLOGAXIS2 .. (V.)G.NLOGAXISn	[VAR]
Spindle logic number, based on its order in the channel. - Variable: (V.)G.NLOGSPDL1 (V.)G.NLOGSPDL2 .. (V.)G.NLOGSPDLn	[VAR]
Modify the proportional gain (Kp) of the tandem from the PLC. - Variable: (V.)MPG.TPROGAIN[nb]	[VAR]
Number of teeth on the tool. - Variable: (V.)G.NCUTTERS (V.)TM.NCUTTERS[offset] (V.)TM.NCUTTERST[tool][offset]	[VAR]
Percentage of feedrate in G00. - Variable: (V.)PLC.G00FRO	[VAR]
Operating Terms. Temporary license for use at the CNC. - Variable: G.CNCDISSTAT	[OPT] [VAR]
FMC Function (Fagor Machining Calculator) • A database of materials to be machined and of machining operations (milling and turning) and an interface to choose suitable cutting conditions.	[OPT]
FCAS (Fagor Collision Avoidance System). • The FCAS (Fagor Collision Avoidance System) option monitors automatically, in MDI/MDA, manually and tool inspection movements in real time, so as to avoid collisions between the tool and the machine. - Variable: (V.)G.COLLISIONPERF	[OPT] [VAR]
FCC Function (Fagor Feed Control). • The FFC function makes it possible to replace the feedrate and speed programmed in the cycle with the active values of the execution, which are acted upon by the feed override and speed override.	[OPT]

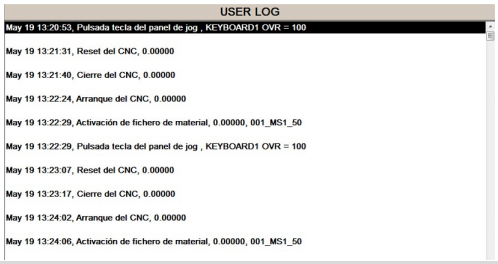
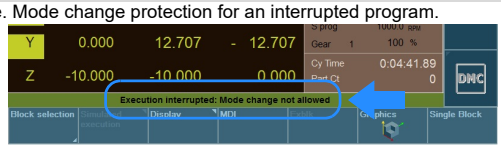
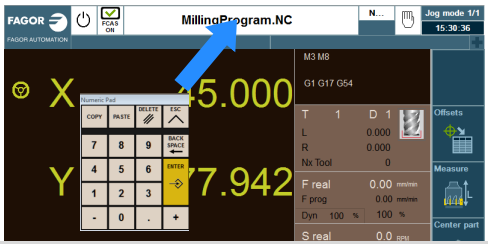
[CYC-M]..... Machining canned cycles (·M· model).
[CYC-T]..... Machining canned cycles (·T· model).
[EXA-T]..... Examples manual (·T· model).
[FGUIM]..... Customizing guide (FGUIM & API).
[INST]..... Installation manual.
[LSR]..... Operating and programming manual (·L· model).
[OPT] ..... Operating manual.
[OPT-MC]..... Operating manual (MC option).
[OPT-TC]..... Operating manual (TC option).
[PRG]..... Programming manual.
[PROGTL3]..... ProGTL3 language.
[VAR]..... CNC variables.

List of features.	Manual
DMC Function (Dynamic Machining Control). • DMC adapts the feedrate during machining to maintain the cutting power as close as possible to ideal machining conditions. - Machine parameter: MINDMCOVR MAXDMCOVR DMCPEAKSIZE DMCPWRFACTOR DMCSPDL - PLC mark: DMCPWRPEAK DMCPWRHOLD - Statement: #DMC - Variable: (V.)MPG.MINDMCOVR (V.)MPG.MAXDMCOVR (V.)PLC.DMCON (V.)G.FRO (V.)G.PRGFRO (V.)G.PLC.FRO (V.)G.CNCFRO (V.)G.DMCPWRSP (V.)G.DMCOVRMIN (V.)G.DMCOVRMAX (V.)G.DMCFZMIN (V.)G.DMCFZMAX (V.)G.DMCON (V.)G.LEARNEDPWRSP (V.)G.DMCACTPWR (V.)G.DMCOVR (V.)G.DMCFZ (V.)G.DMCNOLOADPWR (V.)G.DMCSAVEDTIME	[OPT] [PRG] [VAR]
Identification and registration of the devices connected to the CNC, in setup and user modes.	[OPT]
Encrypting/decrypting a PLC project.	[OPT]
Kinematics calibration. • The kinematics calibration cycle has two associated subroutines (KinCal_Begin.nc and KinCal_End.nc), which the CNC executes before and after the cycle. • The cycle allows the calibration of all types of kinematics (parameter TYPE). • New icon to verify the probe adjustment. • GMOVE macro in the subrouting KinCal_Begin.nc. • The axes may be Hirth axes. • The cycle makes it possible to start the calibration of the main rotary axis at any position of the secondary rotary axis. • If the rotary axis is of the "module" type, the cycle will only allow part of the travel; the part of the itinerary to be calibrated; the not calibrated part is not compensated. • In user mode, the data calculated by the recalibration of the kinematics are saved in the "Offsets" field of the machine parameter table; the "Value" field defined by the OEM cannot be modified. • When calibrating kinematics in Setup mode, if the calculated value is equal to the sum of TDATA+Offset saved, the CNC will not offer to update TDATA and set the offset to 0. • The cycle creates the KinCalReport.mp file with the results of the calibration.	[OPT]
Tool table. • New Zn data. Number of teeth on the tool.	[OPT]
User tables. • Active offsets table. The user tables offer a new table for showing the active values in different functions; G92, G159, G201, etc.	[OPT]
Edisimu mode. The program translator written in the 8055 language can define the destination folder for the translated program (by default ..\Users\Prg\PRG_8055_TO_8070).	[OPT]
Translator 8055. • Translation of the G72+G16+G15 sequence. • Translation of conversational cycles.	[OPT]

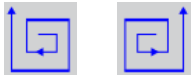
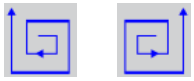
Channel 1 : Active Offsets

	X (mm)	Y (mm)	Z (mm)	B (deg.)
G92	00000.0000	00000.0000	00000.0000	00000.000
G54..G159	00000.0000	00000.0000	00000.0000	00000.000
Δ	00000.0000	00000.0000	00000.0000	00000.000
G158	00000.0000	00000.0000	00000.0000	00000.000
FIX	00000.0000	00000.0000	00000.0000	00000.000
G101	00000.0000	00000.0000	00000.0000	00000.000
G201	00000.0000	00000.0000	00000.0000	00000.000
ACS/CS	00000.0000	00000.0000	00000.0000	
RTCP	00000.0000	00000.0000	00000.0000	
HEAD	00000.0000	00000.0000	00000.0000	00000.000
PLCOF	00000.0000	00000.0000	00000.0000	00000.000

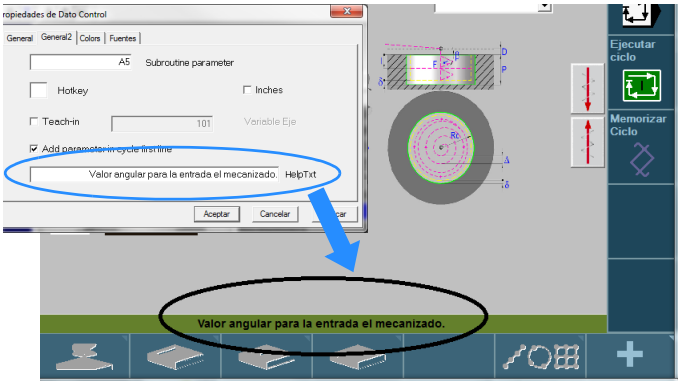
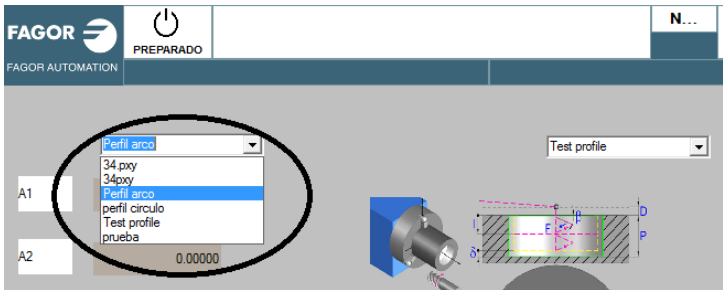
[CYC-M]..... Machining canned cycles (·M· model).
[CYC-T]..... Machining canned cycles (·T· model).
[EXA-T]..... Examples manual (·T· model).
[FGUIM]..... Customizing guide (FGUIM & API).
[INST]..... Installation manual.
[LSR]..... Operating and programming manual (·L· model).
[OPT]..... Operating manual.
[OPT-MC]..... Operating manual (MC option).
[OPT-TC]..... Operating manual (TC option).
[PRG]..... Programming manual.
[PROGTL3]..... ProGTL3 language.
[VAR]..... CNC variables.

List of features.	Manual
"Intel Graphics Control Panel" application. Configuring the brightness and contrast of the monitors.	[OPT]
User log, which chronologically stores events that have occurred on the CNC, which may be important to the user, e.g. errors, keystrokes on the operator panel, etc. - Variable: (V.)E.USERLOG 	[OPT]
Utilities mode. The CNC will ask for confirmation before backing up the data and before restoring data from a back up.	[OPT]
Utilities mode. Encryption of scripts (scp files) and subroutines associated with user cycles.	[OPT]
Automatic mode. From this and later versions, after interrupting the execution of the program and canceling it using the "Cancel-Continue" softkey, the CNC allows the machine to be moved using the jog keys.	[OPT]
Automatic mode. Mode change protection for an interrupted program. 	[OPT]
Improved resolution of the help drawings displayed on various CNC screens; for example, machining cycles, geometric aids, etc.	[OPT]
Edisimu mode. The selected simulation options are maintained after the CNC has been restarted.	[OPT]
STD graphics. • In conversational mode, the CNC remembers the type of window, "Simple" (only shows the graphic) or "Full" (shows dimensions, feeds, etc.). • The selected window configuration is maintained after the CNC has been restarted. • Selecting the simulation speed to a minimum will pause the simulation. • The selected simulation speed is maintained after the CNC has been restarted.	[OPT]
HD graphics. • The default configuration shows the axes and tool paths. • Lathe HD graphics take into account offsets in the X axis of part zero.	[OPT]
Error 3753. Absolute encoder. Monitor the coordinate difference during startup (parameter MAXDIFREF).	[OPT]
The number of softkeys in the menus can be configured. Depending on the configuration (resolution and number of softkeys), if the softkeys are too small to display their image then only their text will be displayed.	[OPT]
Mini-numeric keypad. By clicking on the top center of the status bar (name of the selected program), the CNC displays a mini numeric keypad to make it easier to enter data by using the touch screen. 	[OPT]
The profile editor shows the CZ plane oriented correctly.	
Cycle editor (model M). • New configurator for the cycles associated with the rotary axes. • Softkey to hide Fagor cycles (machining and probe cycles). The softkey is also available in conversational mode. 	[CYC-M] [OPT-MC]

[CYC-M]..... Machining canned cycles (·M· model).
[CYC-T]..... Machining canned cycles (·T· model).
[EXA-T]..... Examples manual (·T· model).
[FGUIM]..... Customizing guide (FGUIM & API).
[INST]..... Installation manual.
[LSR]..... Operating and programming manual (·L· model).
[OPT] ..... Operating manual.
[OPT-MC]..... Operating manual (MC option).
[OPT-TC]..... Operating manual (TC option).
[PRG]..... Programming manual.
[PROGTL3]..... ProGTL3 language.
[VAR]..... CNC variables.

List of features.	Manual
Editor cycles (M model). New canned cycles associated with the rotary axes. • Multiple center-punching. • Multiple reaming. • Multiple boring. • Multiple bore milling. • Multiple thread milling. • Multiple drilling. • Multiple tapping. • Multiple slot milling. • Profile on the cylindrical surface of the part. • Rectangular pocket on the cylindrical surface of the part. • Circular pocket on the cylindrical surface of the part. • Pre-emptied circular pocket on the cylindrical surface of the part. • 2D pocket profile on the cylindrical surface of the part. • Profile on the front surface of the part. • Rectangular pocket on the front surface of the part. • Circular pocket on the front surface of the part. • Pre-emptied circular pocket on the front surface of the part. • 2D pocket profile on the front surface of the part.	[CYC-M] [OPT-MC]
Editor cycles (M model). 2D profile pocket cycle. • Define the machining strategy; from the inside out or vice versa. • Defining the direction of the machining; clockwise or counterclockwise. 	[CYC-M] [OPT-MC]
Cycle editor (T model). • Softkey to hide the Fagor cycles from the editor (machining and probe cycles). The softkey is also available in conversational mode. 	[CYC-T] [OPT-TC]
Editor cycles (T model). 2D profile pocket in the ZC/YZ plane. • Define the machining strategy; from the inside out or vice versa. • Defining the direction of the machining; clockwise or counterclockwise. 	[CYC-T] [OPT-TC]
Editor cycles (T model). Profile turning cycle. • New operation to remove the residual material during roughing operations.	[CYC-T] [OPT-TC]
Editor cycles (T model). Grooving cycles (except cutting cycles). • Penetration with chip removal in all passes. 	[CYC-T] [OPT-TC]
Cycle editor (T model). Profiles • The canned cycles may be defined by the CZ plane. 	[CYC-T] [OPT-TC]

[CYC-M]..... Machining canned cycles (·M· model).
[CYC-T]..... Machining canned cycles (·T· model).
[EXA-T]..... Examples manual (·T· model).
[FGUIM]..... Customizing guide (FGUIM & API).
[INST]..... Installation manual.
[LSR]..... Operating and programming manual (·L· model).
[OPT]..... Operating manual.
[OPT-MC]..... Operating manual (MC option).
[OPT-TC]..... Operating manual (TC option).
[PRG]..... Programming manual.
[PROGTL3]..... ProGTL3 language.
[VAR]..... CNC variables.

List of features.	Manual
Simulation options in conversational mode. • Radius compensation. • Conditional stop during simulation. • Software and work zone limits. • Block skip. • Cancel channel synchronization. • Synchronizing spindles. • Assume the active origins for execution. • Assume the active zones in execution.	[OPT-MC] [OPT-TC]
ProGTL3 language. Generic user subroutines (G8000-G8999).	[PROGTL3]
ProGTL3 language. Programming cycles (macros). • Milling of a surface defined by a plane profile and one or more section profiles. Machine only the bottom of the pocket. - G8736. Parameter D0 (D0=3). • Invert the machining direction of the profile - G8754. Start of profile. - G8753. End of the profile. • Circular polygon pocket defined by a profile. - G8777. Define the cycle and its parameters. - G8778. Activate the cycle. - G8701. Profile pocket. Start of the profile and internal island profiles. - G8702. Profile pocket. Profile section of the pocket and islands (optional).	[PROGTL3]
Customisation of the user cycle header. The “Data” control has a new property called “Add parameter in cycle first line”. If this property is checked, the cycle adds the data to the cycle header as “Variable=Value”. The variable will take on the name of the property “Subroutine parameter”. +#CYCLE BEGIN (1-Simple screw) (V.C.TOOL=1 V.C.SPEED=100) (CW_V_1_0)	[FGUIM]
Association of a help text to user cycle data. The “Data” control has the new “HelpTxt” property that defines the help text appearing on the CNC when selecting the data. 	[FGUIM]
New FCombo control to manage profiles in user cycles. 	[FGUIM]
Fguim allows changes to be made to the interface while the CNC is running. Any changes made by Fguim are updated in the CNC during the next startup. For the changes to be assumed at the CNC, they must be previously saved from the Fguim.	[FGUIM]

[CYC-M]..... Machining canned cycles (·M· model).
[CYC-T]..... Machining canned cycles (·T· model).
[EXA-T]..... Examples manual (·T· model).
[FGUIM]..... Customizing guide (FGUIM & API).
[INST]..... Installation manual.
[LSR]..... Operating and programming manual (·L· model).
[OPT] ..... Operating manual.
[OPT-MC]..... Operating manual (MC option).
[OPT-TC]..... Operating manual (TC option).
[PRG]..... Programming manual.
[PROGTL3]..... ProGTL3 language.
[VAR]..... CNC variables.

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

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.

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.

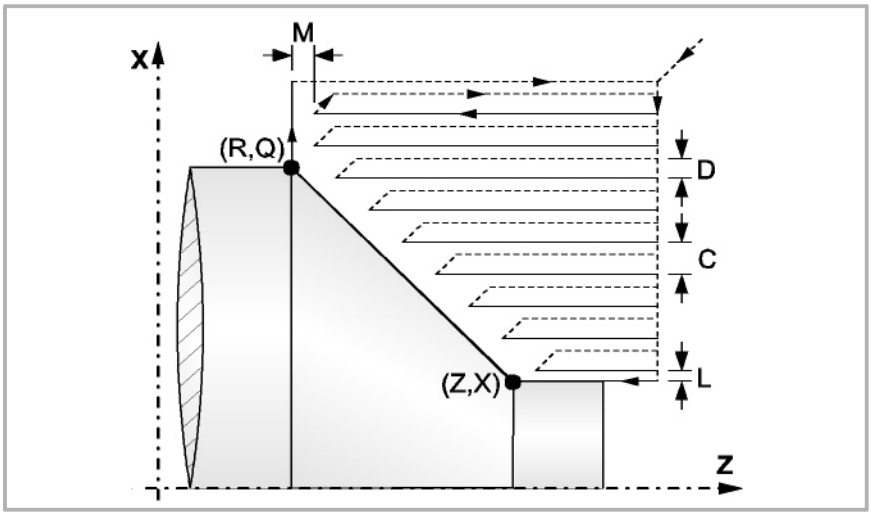
G81 ⋯	Turning canned cycle with straight sections.
#MILLCY ON	Activates the milling canned cycles.
G81 ⋯	
G86 ⋯	
#MILLCY OFF	Deactivates the milling canned cycles.

1.2 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.

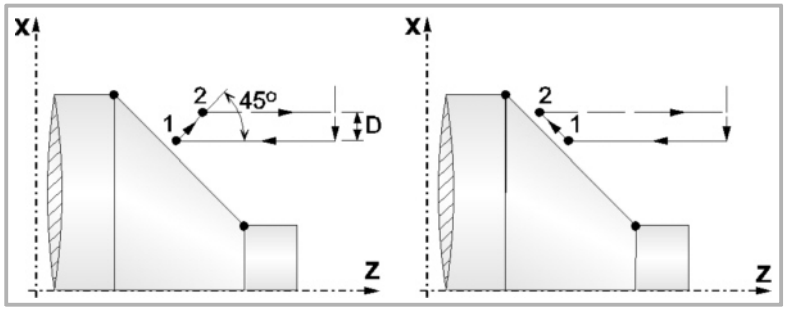
C5.5 Defines the turning pass. It is given by a positive radius value. The whole turning operation is carried out with the same pass which is the same or smaller than the programmed one "C".

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 45º 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.



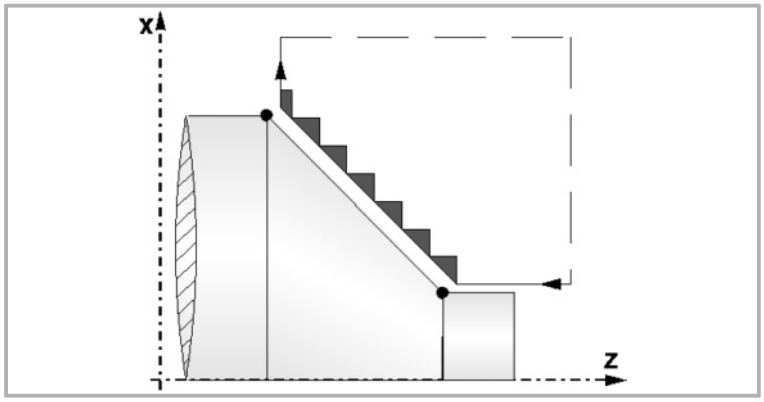
L5.5 Defines the finishing stock along X and it is given in radius.

If not programmed, a value of 0 is assumed.

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.

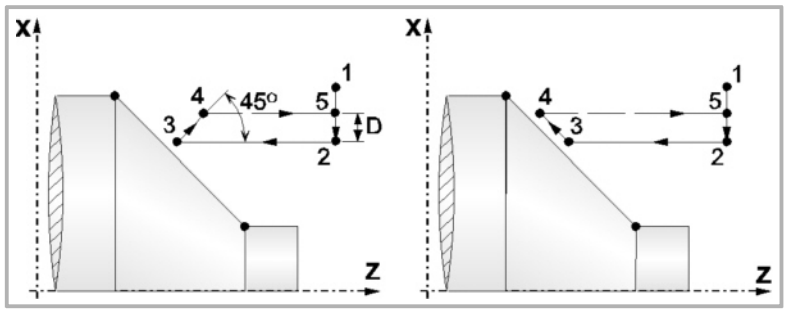


H5.5 Defines the feedrate for the finishing pass.

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:

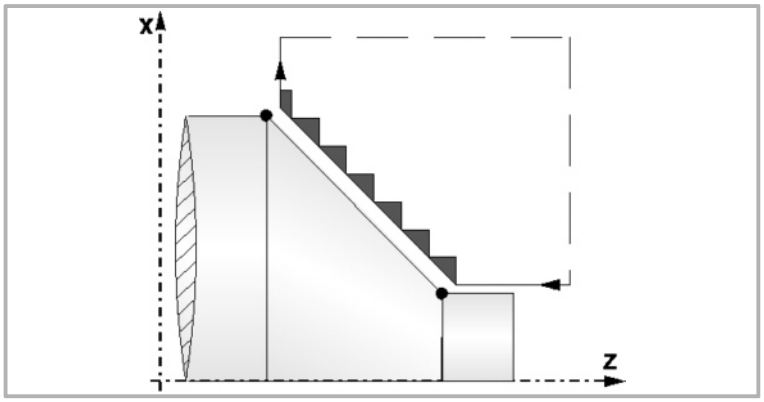


"1-2" move in rapid (G00). Move "2-3" in G01 at programmed feedrate (F).

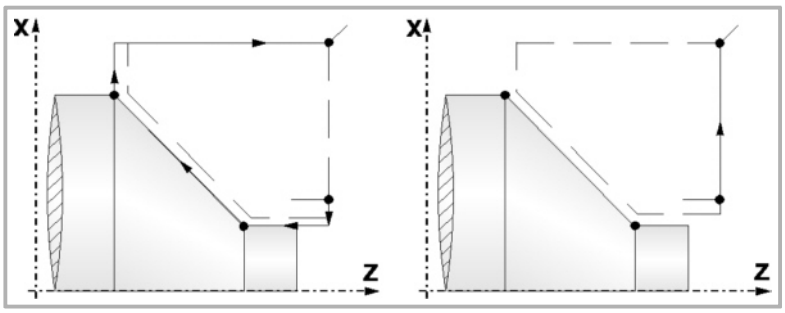
If "D" has been programmed, the 3-4 move is done in rapid (G00); but if "D" has not been programmed, the 3-4 move is done following the profile in G01 at the programmed feedrate (F).

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

- 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.

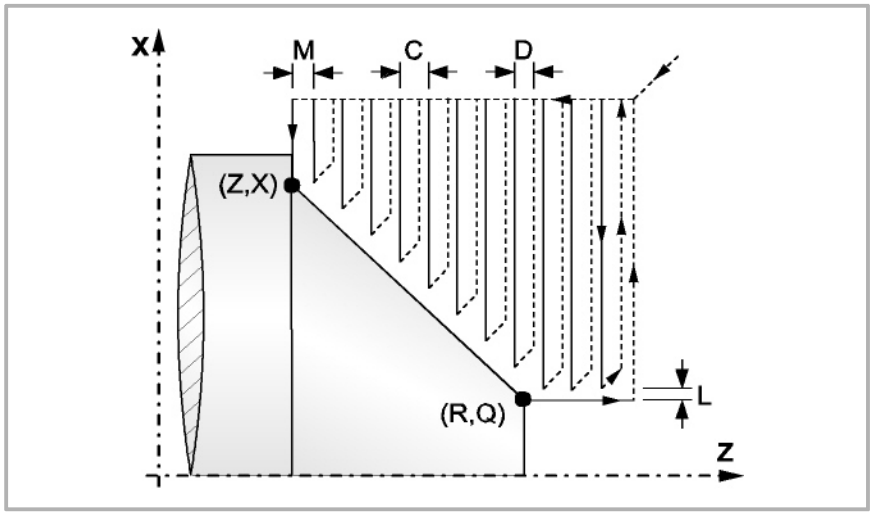


1.3 G82. Facing canned cycle with straight sections

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±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.

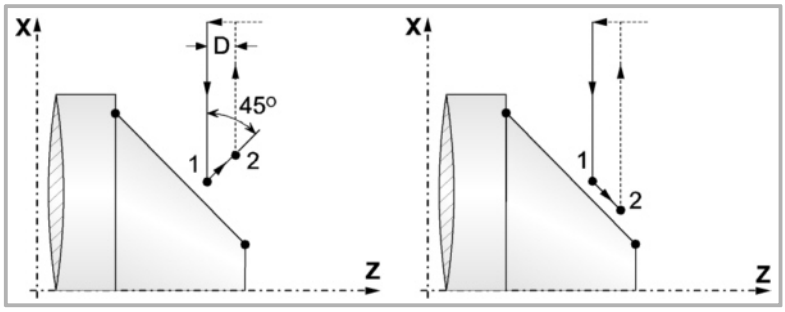
C5.5 Defines the turning pass. It is given by a positive radius value. The whole turning operation is carried out with the same pass which is the same or smaller than the programmed one "C".

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 45º 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.



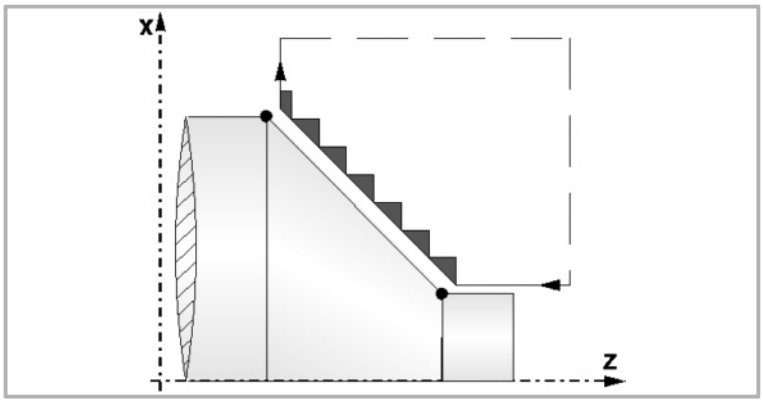
L5.5 Defines the finishing stock along X and it is given in radius.

If not programmed, a value of 0 is assumed.

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.

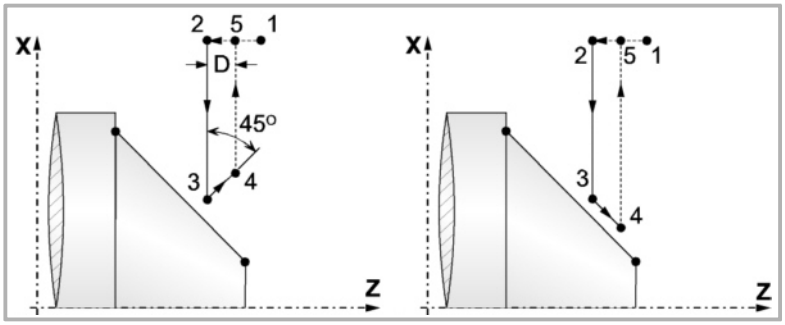


H5.5 Defines the feedrate for the finishing pass.

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:

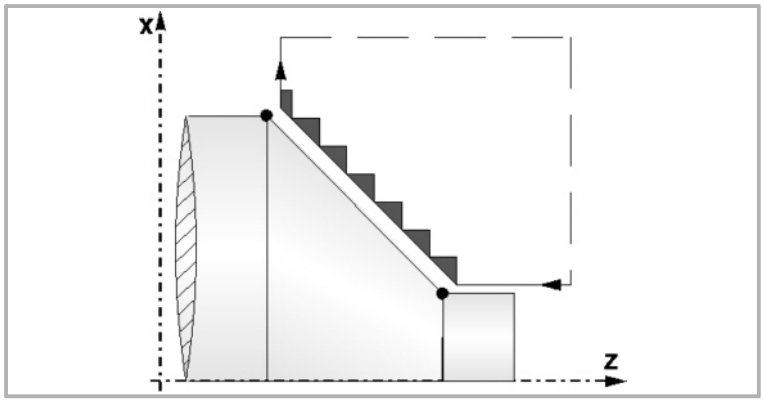


"1-2" move in rapid (G00). Move "2-3" in G01 at programmed feedrate (F).

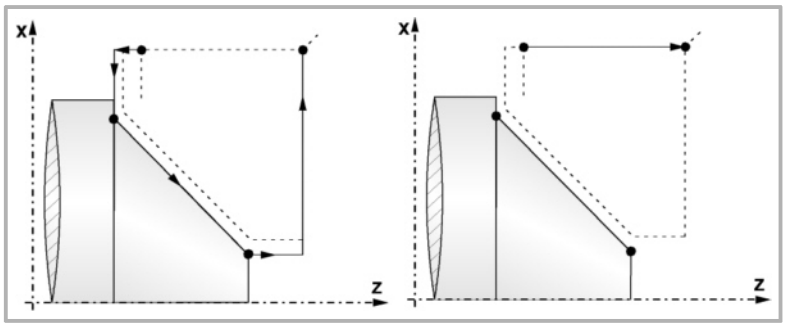
If "D" has been programmed, the 3-4 move is done in rapid (G00); but if "D" has not been programmed, the 3-4 move is done following the profile in G01 at the programmed feedrate (F).

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.



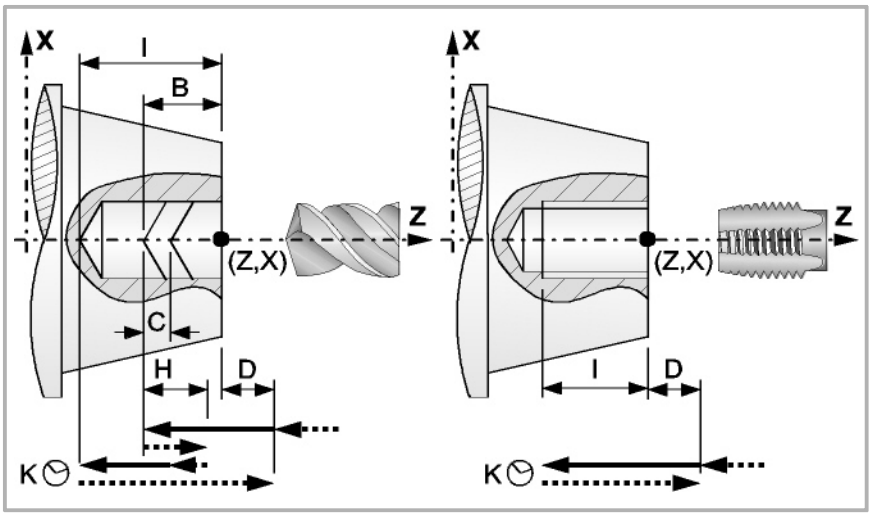
- After the facing operation (with or without finishing pass) the canned cycle will always end at the cycle calling point.



1.4 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.

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).

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.

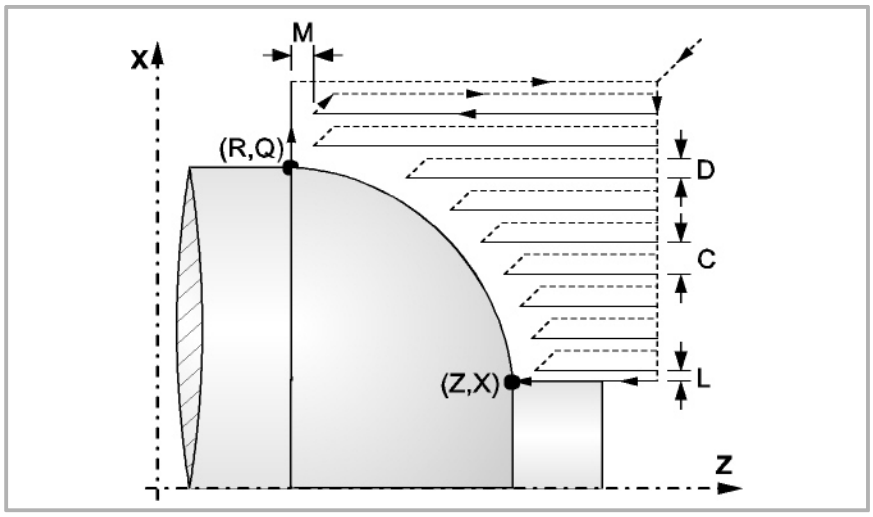
When tapping (regular or rigid), the general logic output "TAPPING" (M5517) stays active during the execution of the cycle.

1.5 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±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.

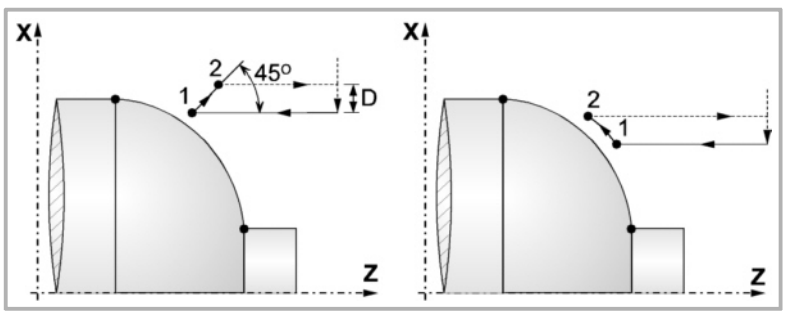
C5.5 Defines the turning pass. It is given by a positive radius value. The whole turning operation is carried out with the same pass which is the same or smaller than the programmed one "C".

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 45º 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.



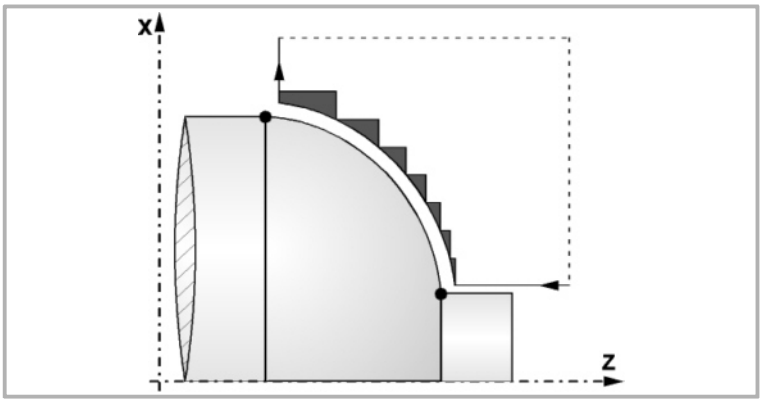
L5.5 Defines the finishing stock along X and it is given in radius.

If not programmed, a value of 0 is assumed.

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.

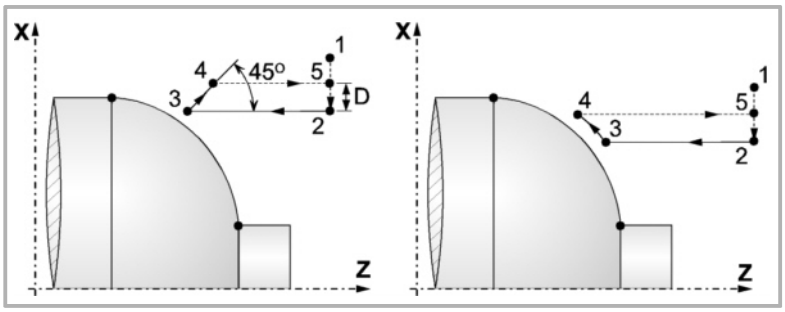
If not programmed or programmed with a value of 0, it is considered that no finishing pass is desired.

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:

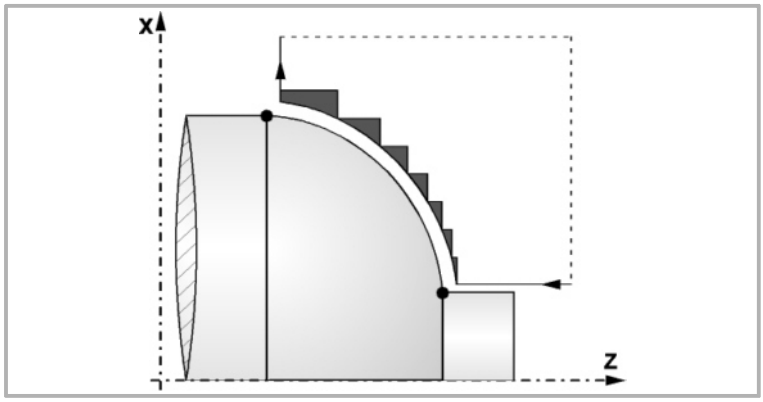


"1-2" move in rapid (G00). Move "2-3" in G01 at programmed feedrate (F).

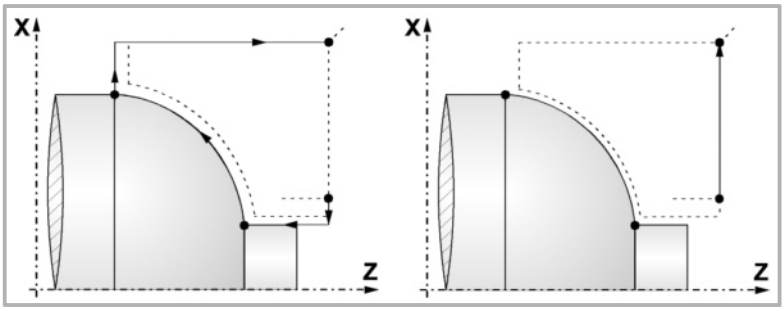
If "D" has been programmed, the 3-4 move is done in rapid (G00); but if "D" has not been programmed, the 3-4 move is done following the profile in G01 at the programmed feedrate (F).

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

- 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.

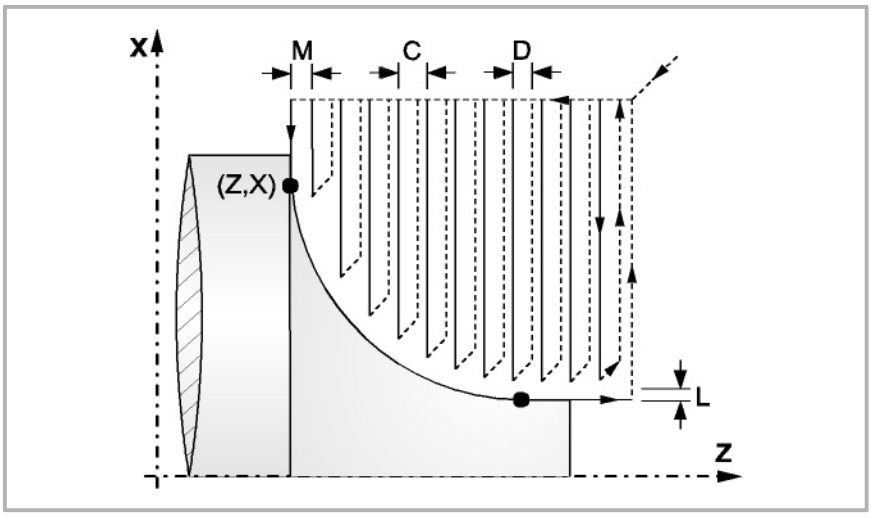


1.6 G85. Facing canned cycle with arcs

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.

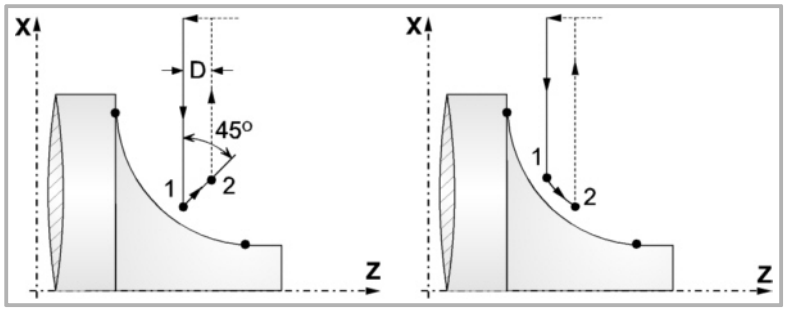
C5.5 Defines the facing pass. The whole facing operation is carried out with the same pass, this being equal to or smaller than the programmed one (C).

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 45º 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.



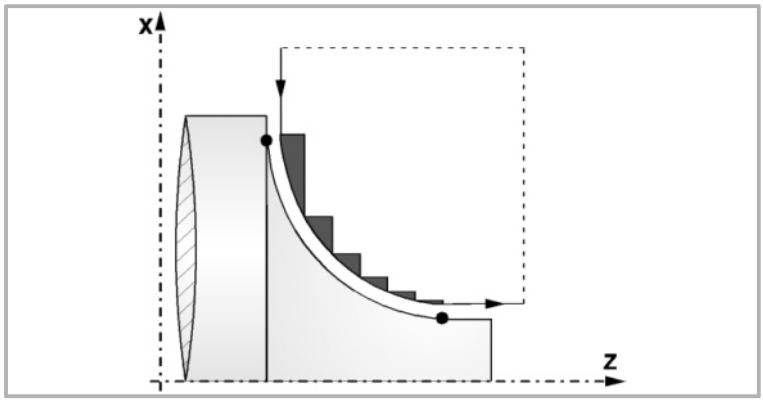
L5.5 Defines the finishing stock along X and it is given in radius.

If not programmed, a value of 0 is assumed.

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.

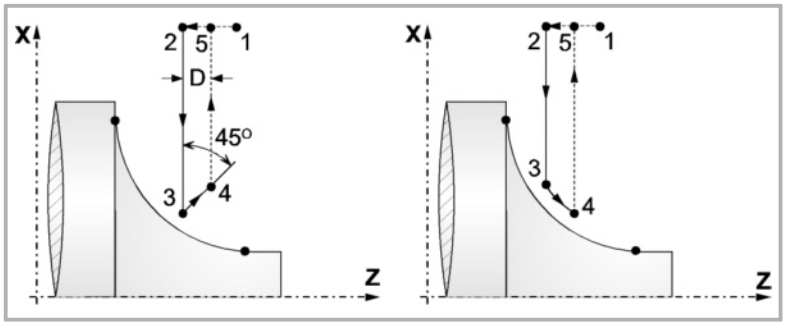
If not programmed or programmed with a value of 0, it is considered that no finishing pass is desired.

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:

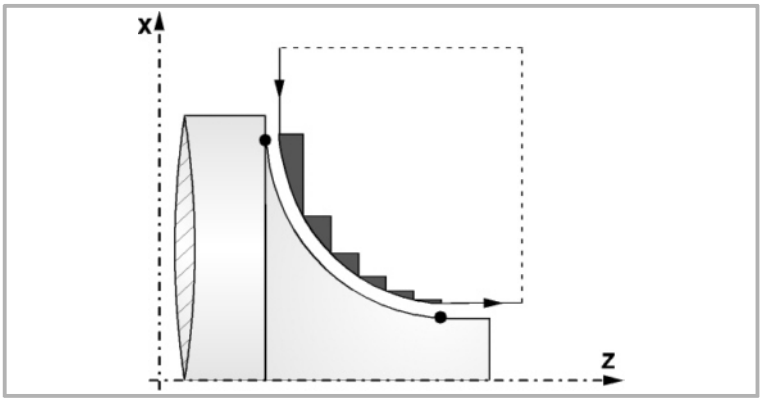


"1-2" move in rapid (G00). Move "2-3" in G01 at programmed feedrate (F).

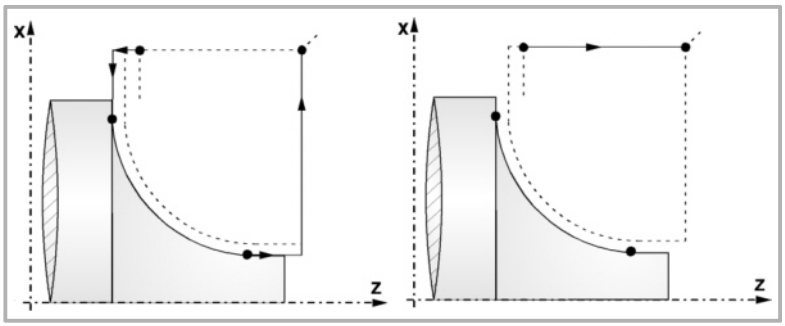
If "D" has been programmed, the 3-4 move is done in rapid (G00); but if "D" has not been programmed, the 3-4 move is done following the profile in G01 at the programmed feedrate (F).

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

- 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.

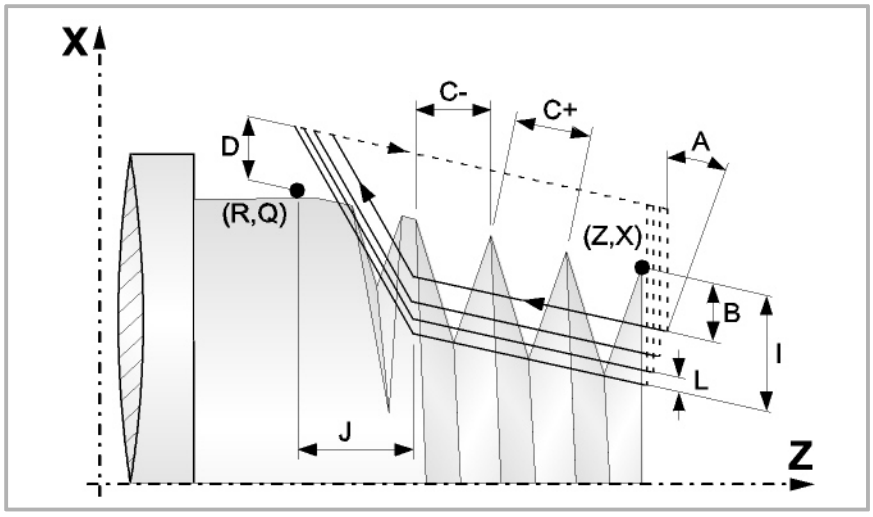


1.7 G86. Longitudinal threading canned cycle

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:

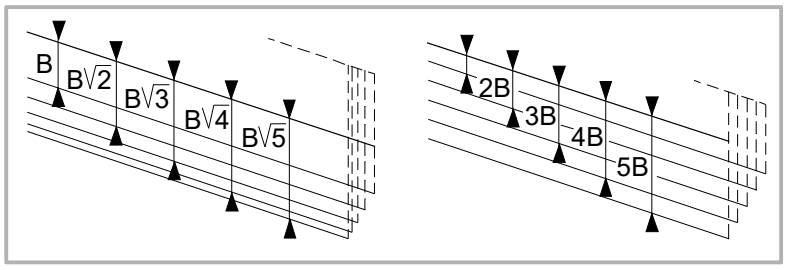
$$B,\ B\sqrt{2},B\sqrt{3},B\sqrt{4},\ldots B\sqrt{n}$$

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, ··· 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.



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).

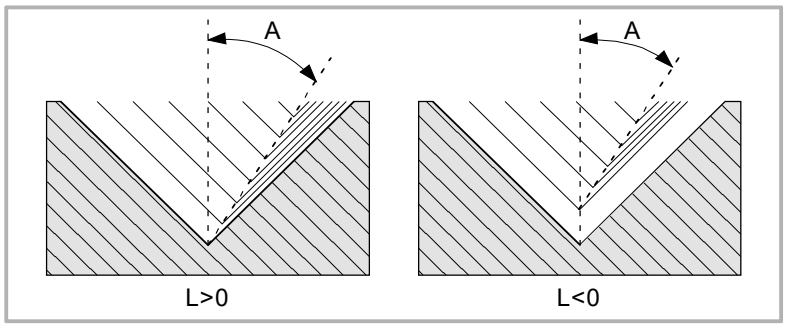
If not programmed, a value of 0 is assumed.

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.



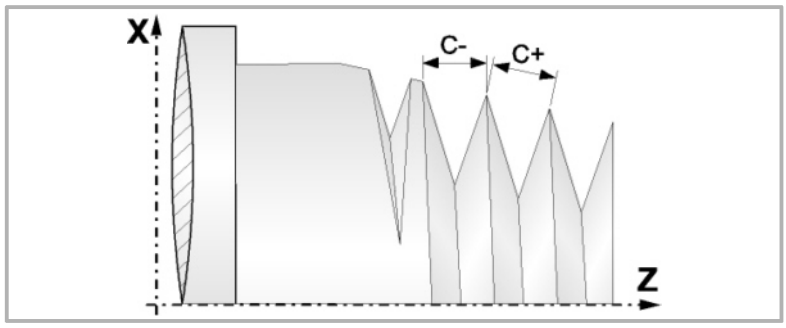
C5.5 Defines the thread’s pitch.

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.

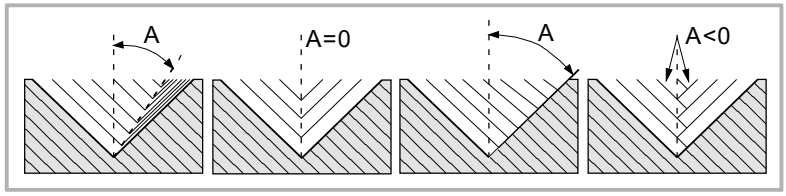


J5.5 Exit from the thread. It defines at what distance from the final point (R, Q), along the Z axis, starts the withdrawal from the thread.

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.

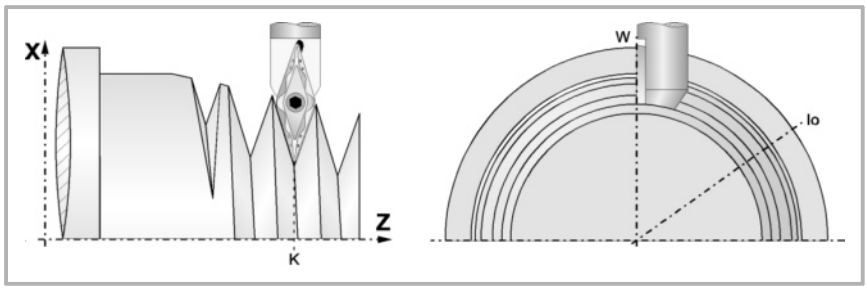
We recommend to use a penetration angle smaller than half the tool angle, never larger. The thread cannot be machined if the penetration angle is larger than half the tool angle. If the penetration angle is equal to half the tool angle, it will rub the flank of the thread at each cutting pass.



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.

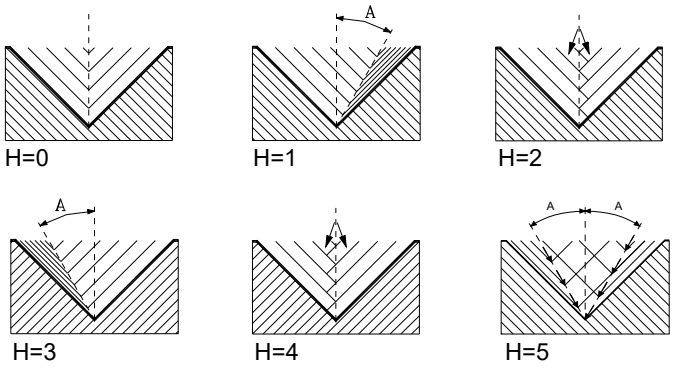
G86 X Z Q R K I B E D L C J A W0 V3

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.

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.

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.

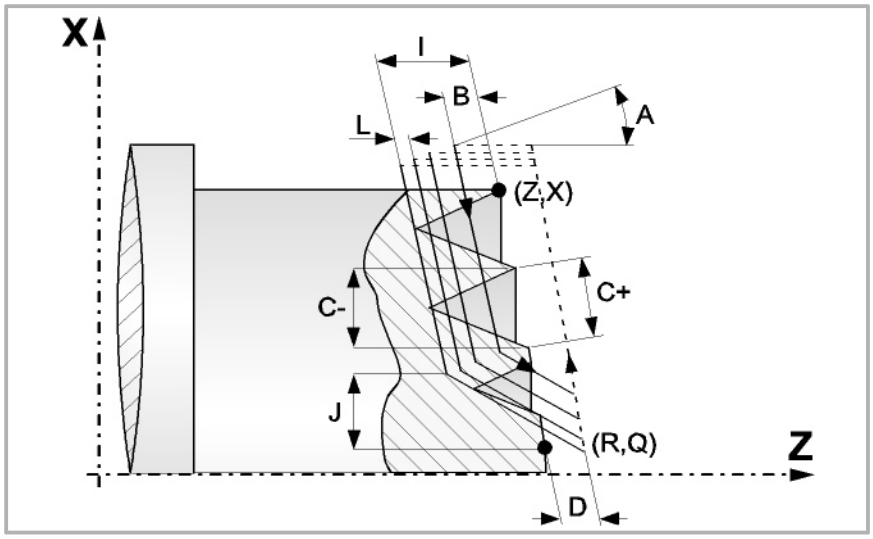
4 Execute the canned cycle.

1.8 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 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. 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:

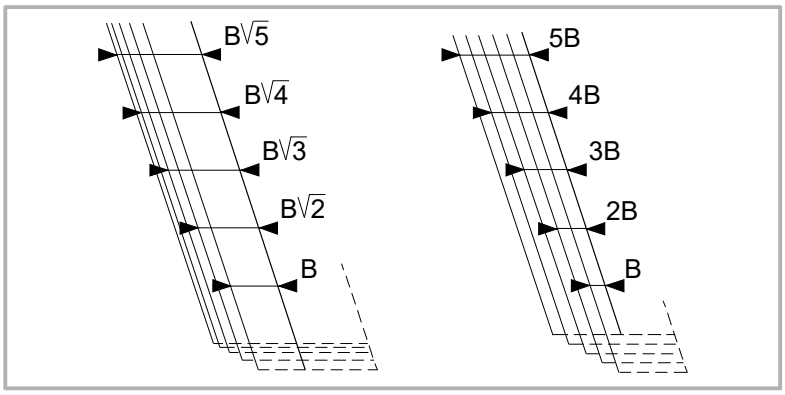
$$B,\ B\sqrt{2},B\sqrt{3},B\sqrt{4},\ldots B\sqrt{n}$$

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, ··· 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.



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).

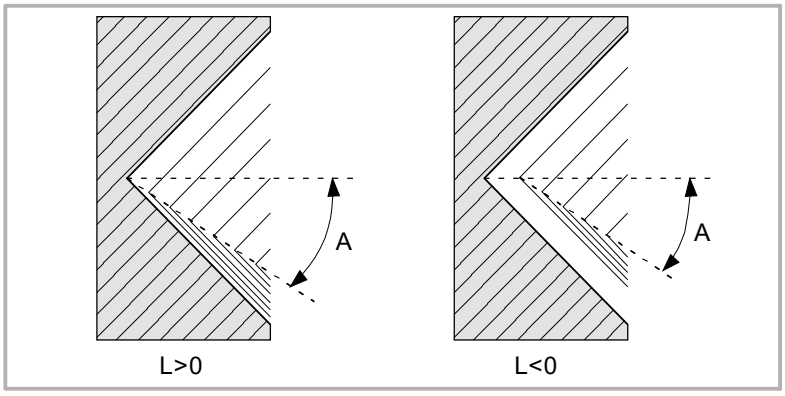
If not programmed, a value of 0 is assumed.

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.



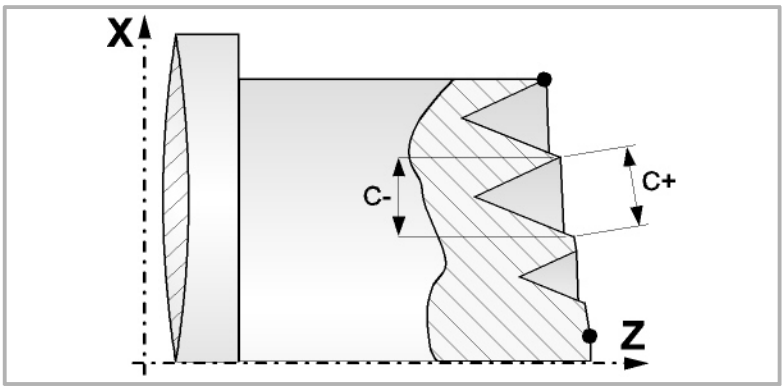
C5.5 Defines the thread’s pitch.

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.

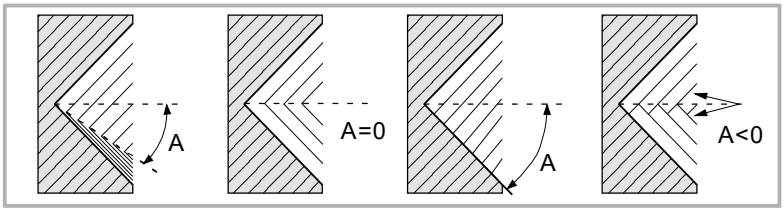


J5.5 Exit from the thread. It defines at what distance from the final point (R, Q), along the X axis, starts the withdrawal from the thread.

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.

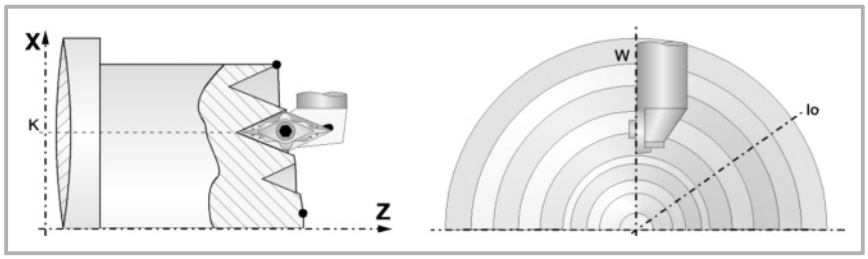
We recommend to use a penetration angle smaller than half the tool angle, never larger. The thread cannot be machined if the penetration angle is larger than half the tool angle. If the penetration angle is equal to half the tool angle, it will rub the flank of the thread at each cutting pass.



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.

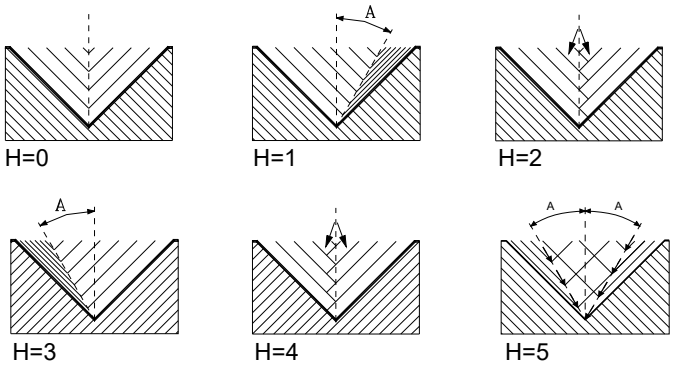
G87 X Z Q R K I B E D L C J A W0 V3

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.

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.

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.

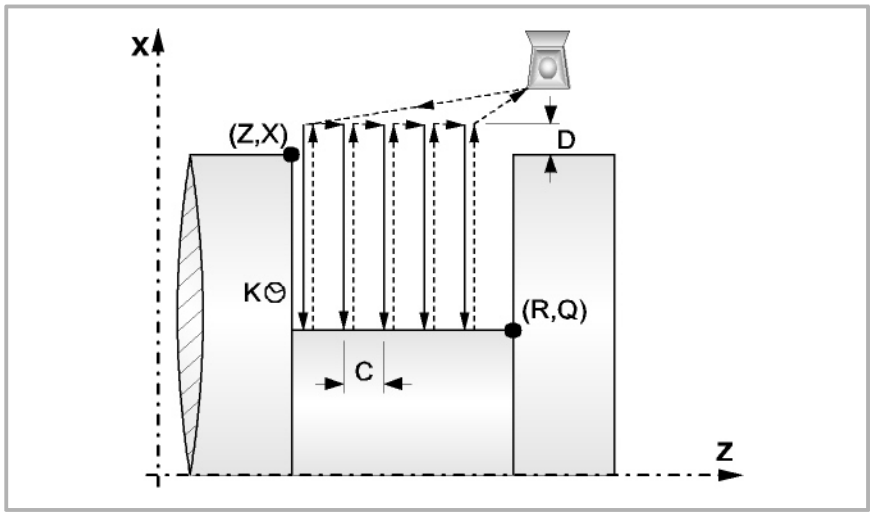
4 Execute the canned cycle.

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

R±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.

C5.5 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.

D5.5 Defines the safety distance and it must have a positive value in radius.

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

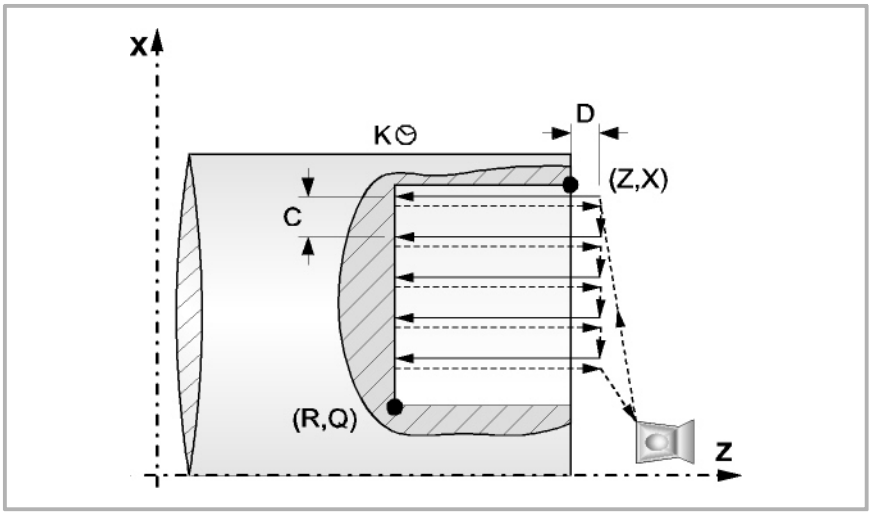
- The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the cycle. Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle. Only the tool radius compensation will be canceled (G40) if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

1.10 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 programamed.
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:

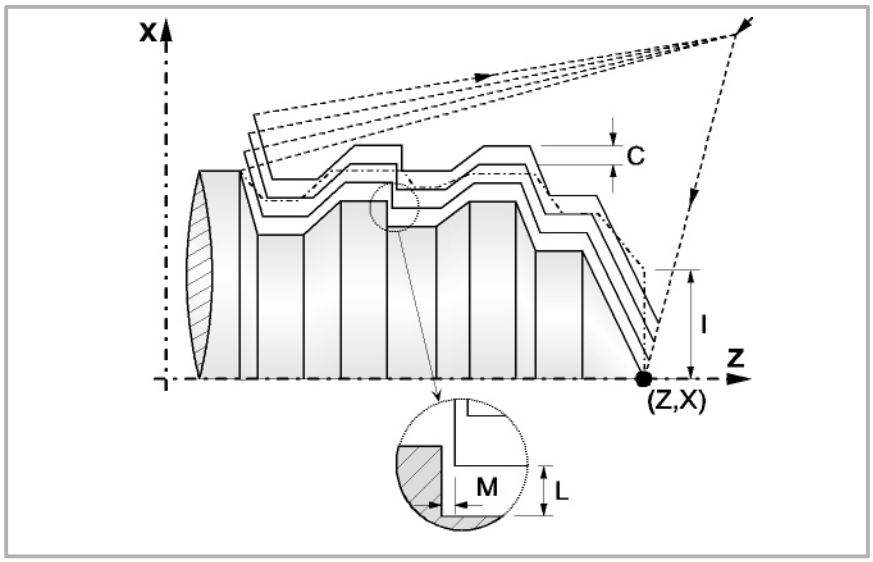
  The penetration is done at programmed feedrate (F). The withdrawal and the move to the next penetration point are done in rapid (G00).
- After completing the grooving operation, the canned cycle will always end at the cycle calling point.

1.11 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".

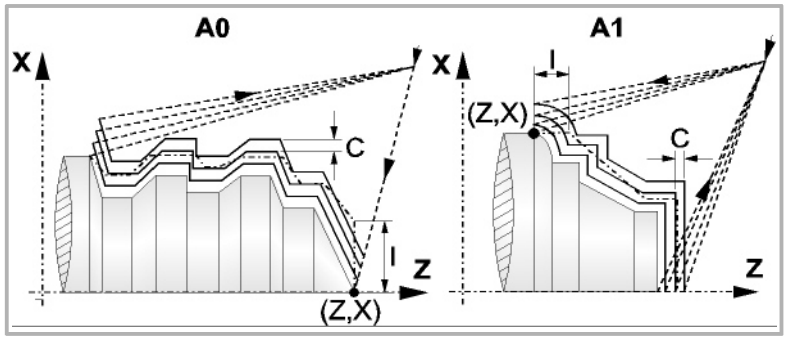
All the machining passes are performed with this value, except the last one which will eliminate the remaining stock.

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.

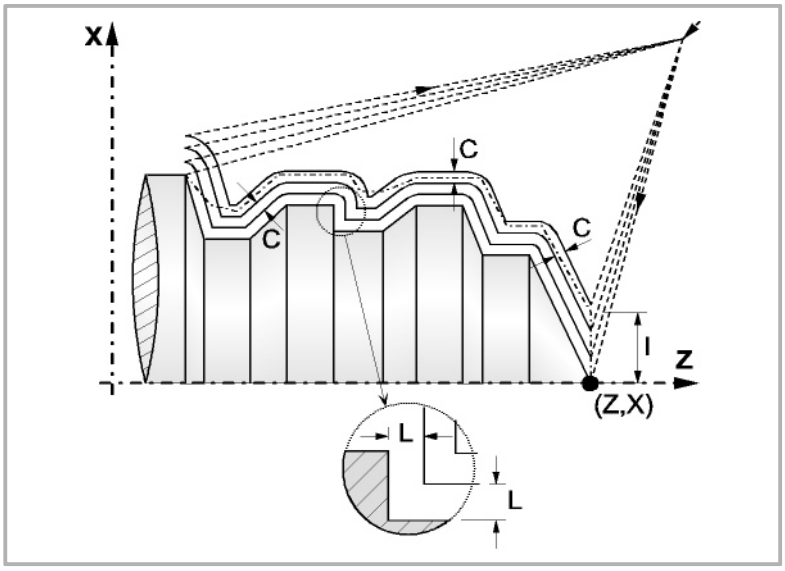
When programming "A1", the X axis will be the main axis, "I" will be taken as Z residual value and "C" as the pass along Z.



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

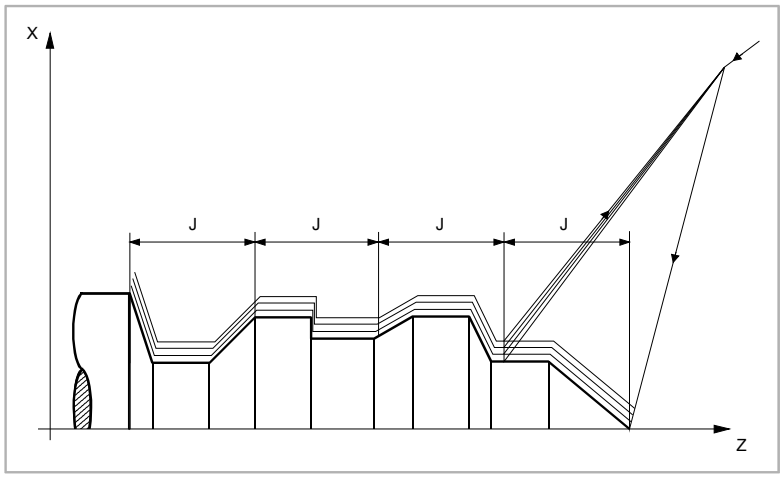
M±5.5 Defines the finishing stock along Z.

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").

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".

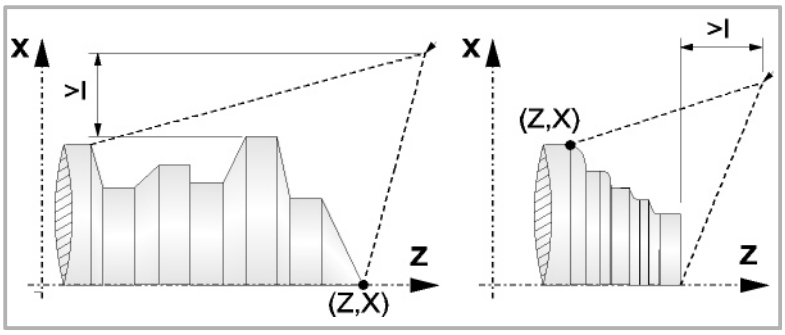
Q Name of the global subroutine, of the program that contains the definition of the profile (parameters "E" and "S") or of the program that contains the local subroutine that contains the profile (parameter "P").

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

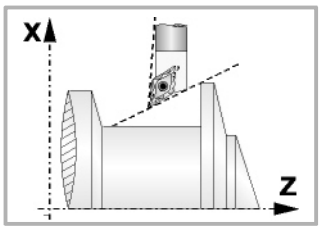
Parameters.	Profile location.
E + S	The profile is defined between the "E" and "S" blocks of the current program.
P	The profile is defined in the local subroutine "P" of the current program.
Q	The profile is defined in global subroutine "Q".
Q + E +S	The profile is defined between the "E" and "S" blocks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



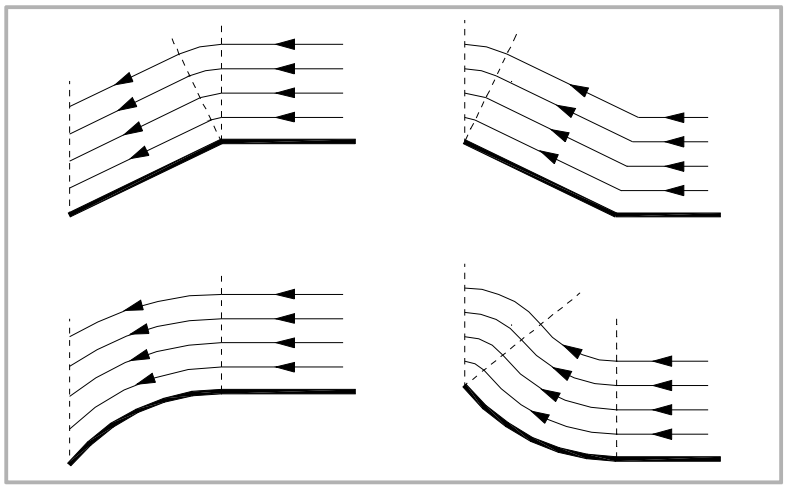
The operator will be able to stop the execution and select the proper tool.

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.

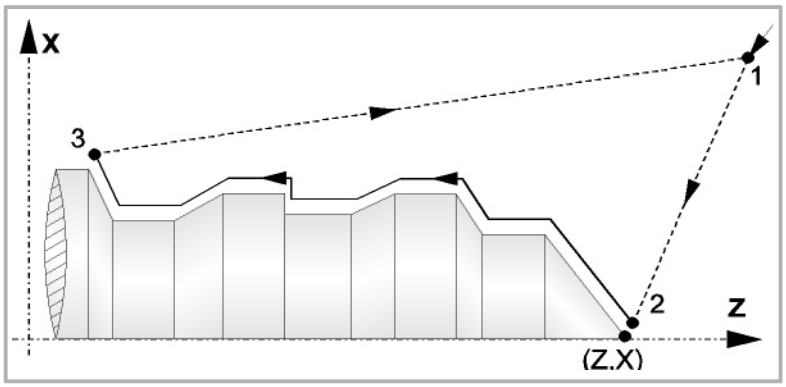
- Once the profile to be executed has been calculated, the CNC will calculate all the passes required to eliminate the programmed residual stock (I).

  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.

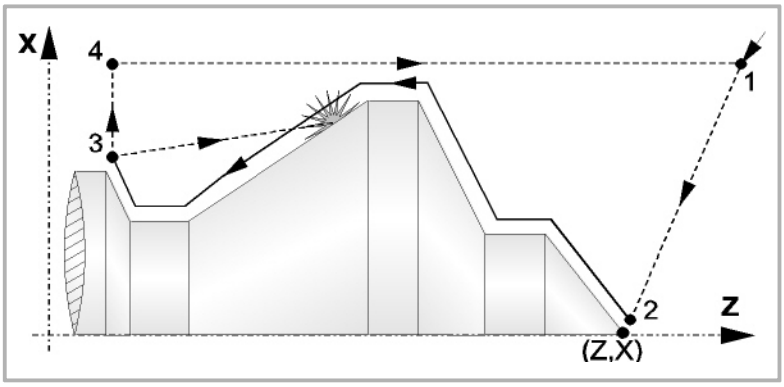


- Each pass is performed in the following way:



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).

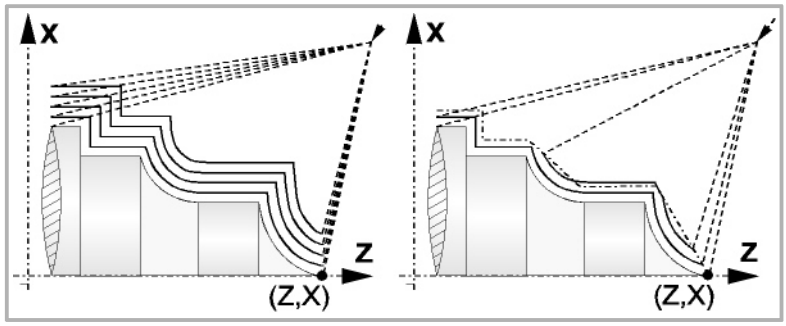
If the possibility of collision exists, the move between points "3-1", it will be carried out in two stages and in G00 (“3-4” and “4-1”) as shown in the next figure.



- 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:

- 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

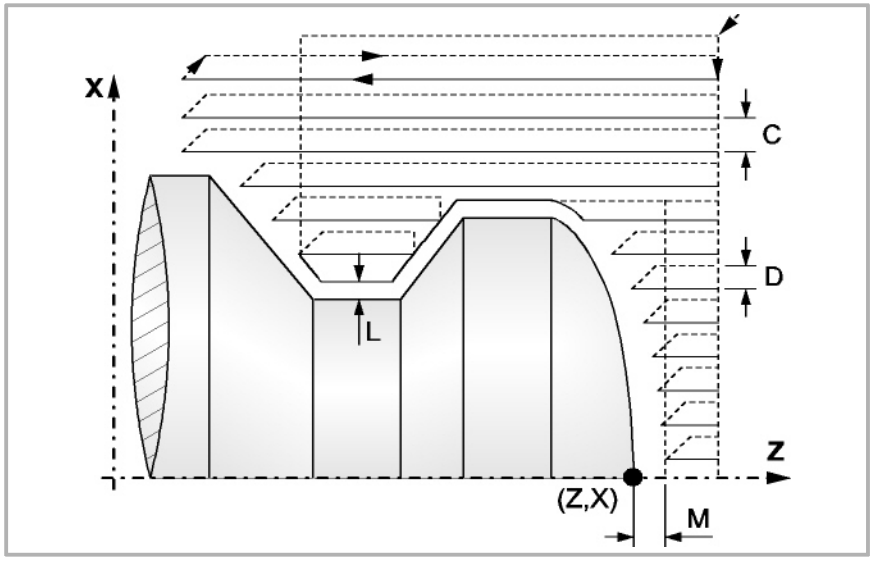
The F, S, T, D and M functions

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.

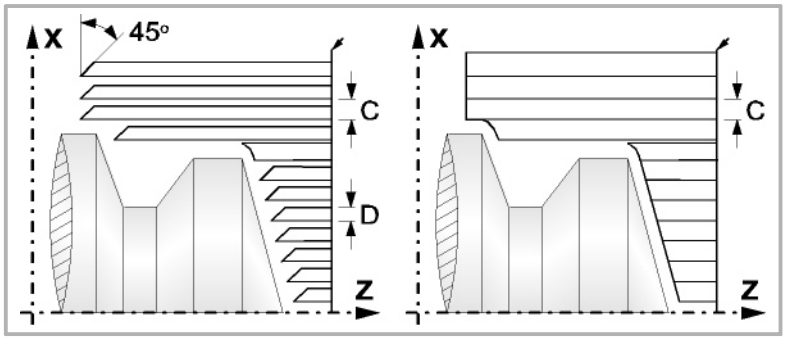
C5.5 Defines the machining pass. It is given by a positive radius value. If programmed with a 0 value, the CNC will display the corresponding error message.

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 45º 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.

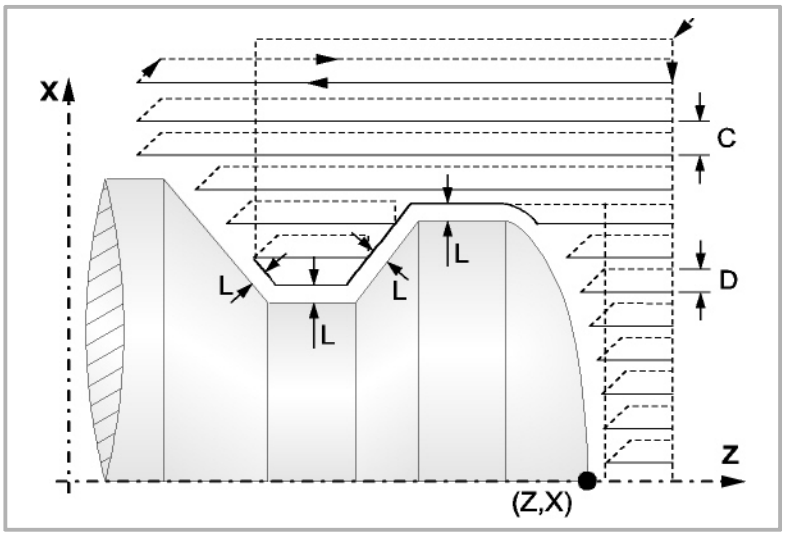
When not programming "D", the tool withdrawal is carried out following the profile up to the last pass, "C" distance (right drawing). It must be borne in mind that when "D" is not programmed, the cycle execution time is greater; but the amount of stock to be removed on the finishing pass is less.



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

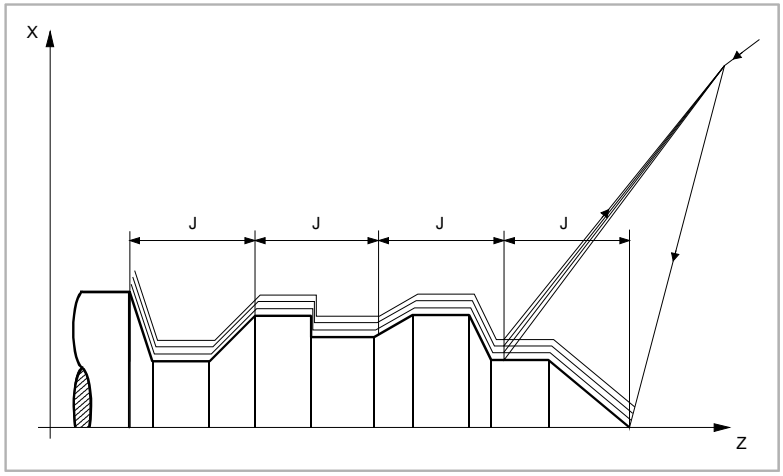
M±5.5 Defines the finishing stock along Z.

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.

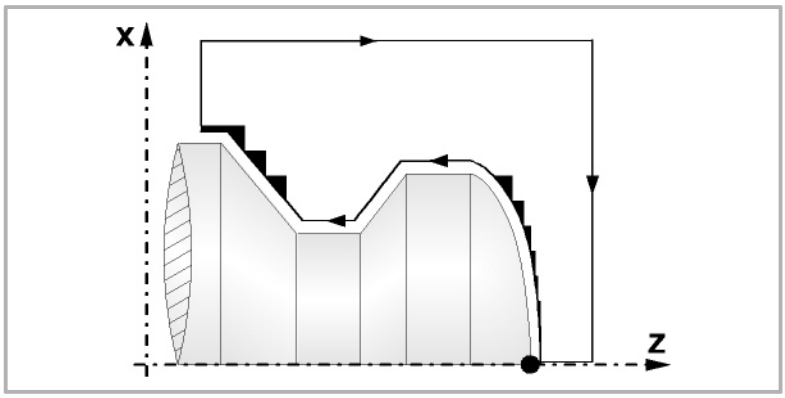
This parameter may be used to machine the cycle by segments indicating the maximum length of each segment along the machining axis. This type of roughing is useful on deep profiles; it lets using more robust tools (avoiding runout) and helping chip removal.



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.



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".

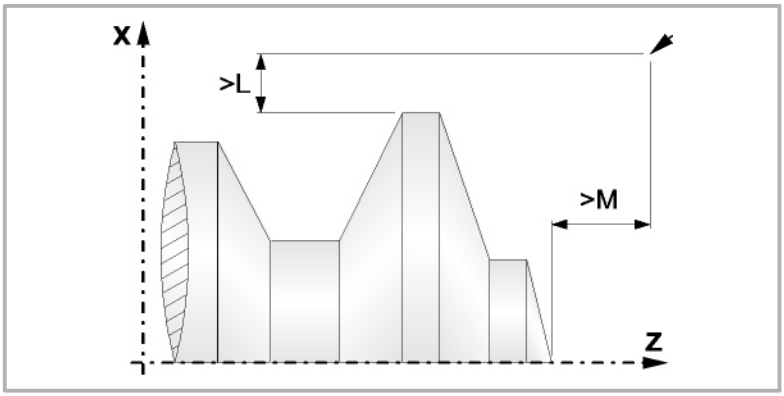
Q Name of the global subroutine, of the program that contains the definition of the profile (parameters "E" and "S") or of the program that contains the local subroutine that contains the profile (parameter "P").

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

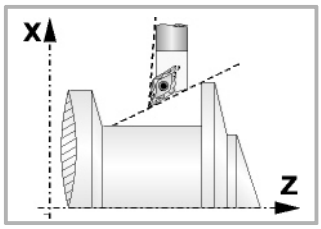
Parameters.	Profile location.
E + S	The profile is defined between the "E" and "S" blocks of the current program.
P	The profile is defined in the local subroutine "P" of the current program.
Q	The profile is defined in global subroutine "Q".
Q + E +S	The profile is defined between the "E" and "S" blocks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



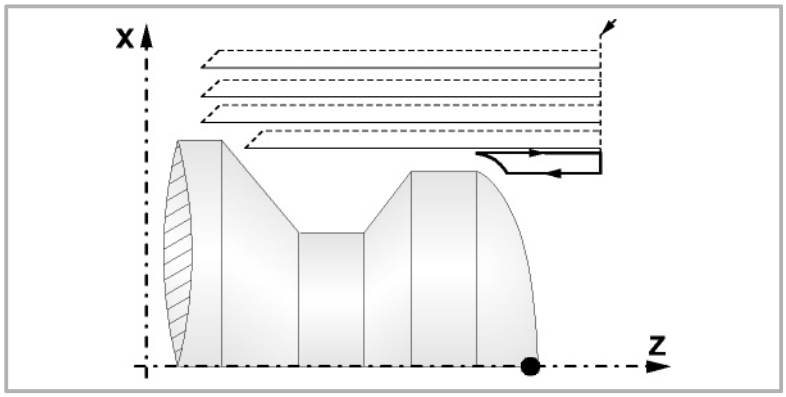
- In those cases when the programmed profile cannot be machined (valleys) with the selected tool, a warning message will be displayed at the beginning of the cycle execution.



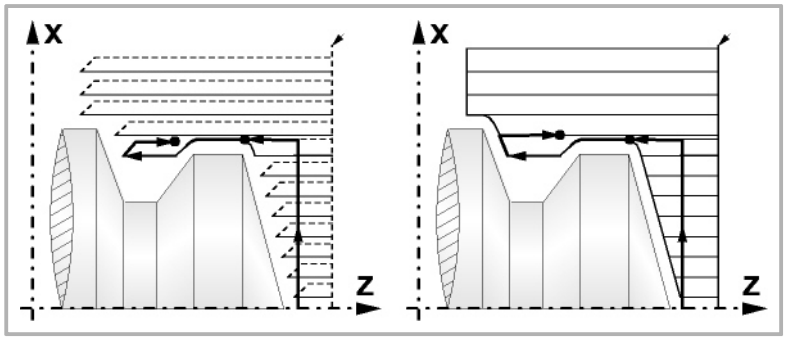
The operator will be able to stop the execution and select the proper tool.

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.

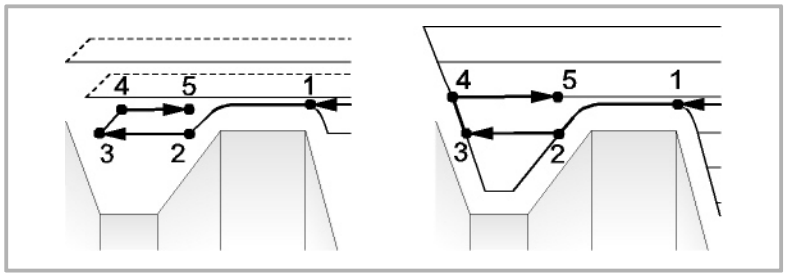
- If when making a roughing pass a channel is detected, the CNC will continue the operation ignoring the channel. The profile may have an unlimited number of channels.



- 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.

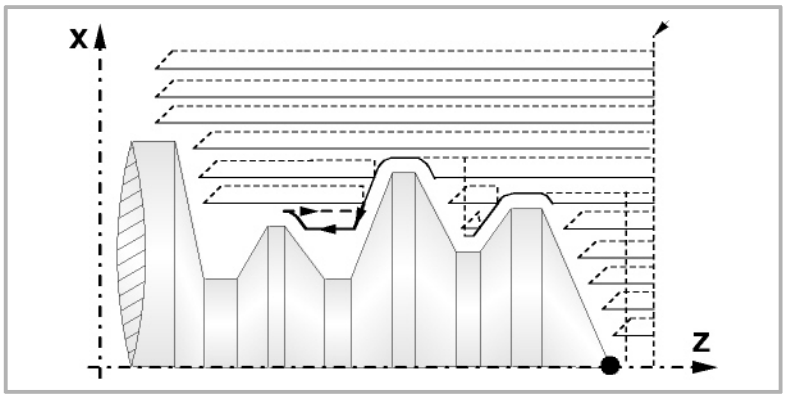


Move “2-3” in G01 at programmed feedrate (F).

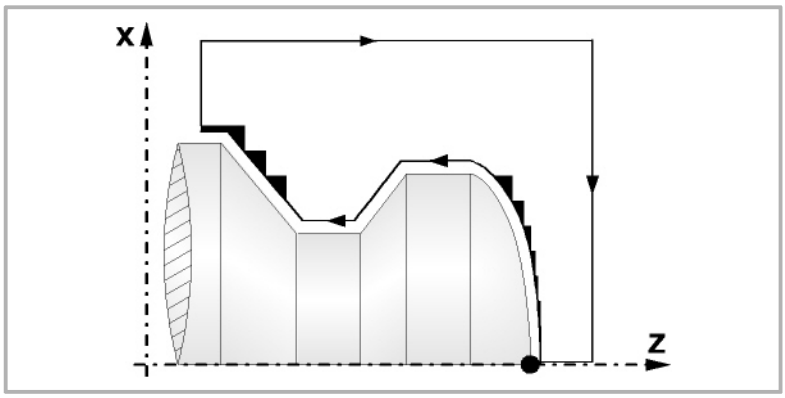
If "D" has been programmed, the 3-4 move is done in rapid (G00); but if "D" has not been programmed, the 3-4 move is done following the profile in G01 at the programmed feedrate (F).

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.

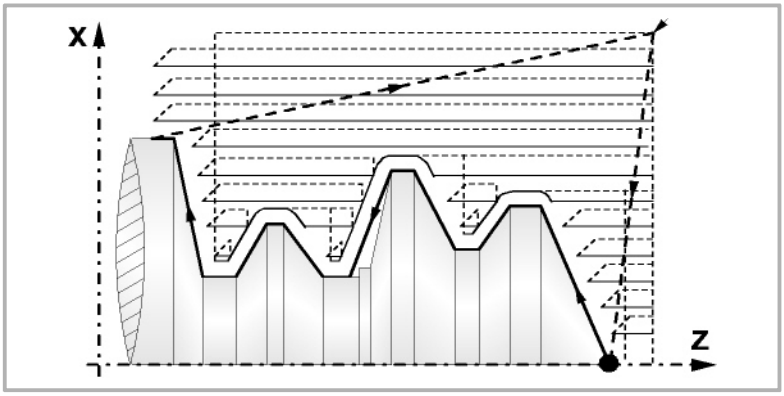


- If a final roughing pass has been programmed, the CNC will run a pass parallel to the profile maintaining an "L" stock at the set feedrate "F". This final roughing pass will eliminate the ridges that were left in the roughing stage.



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

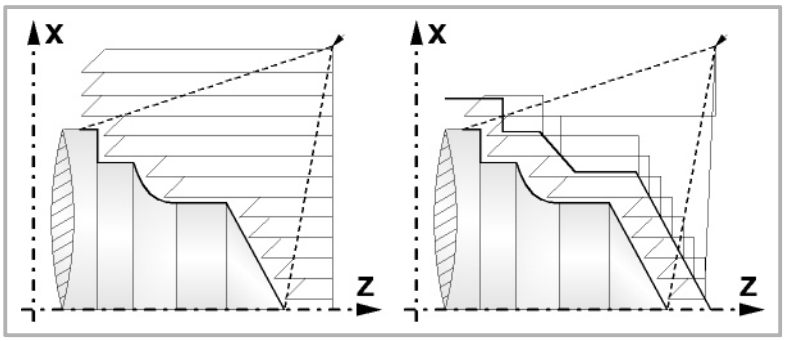
  This profile will coincide or not with the programmed profile depending on whether there are areas not accessible to the selected tool.



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

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

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

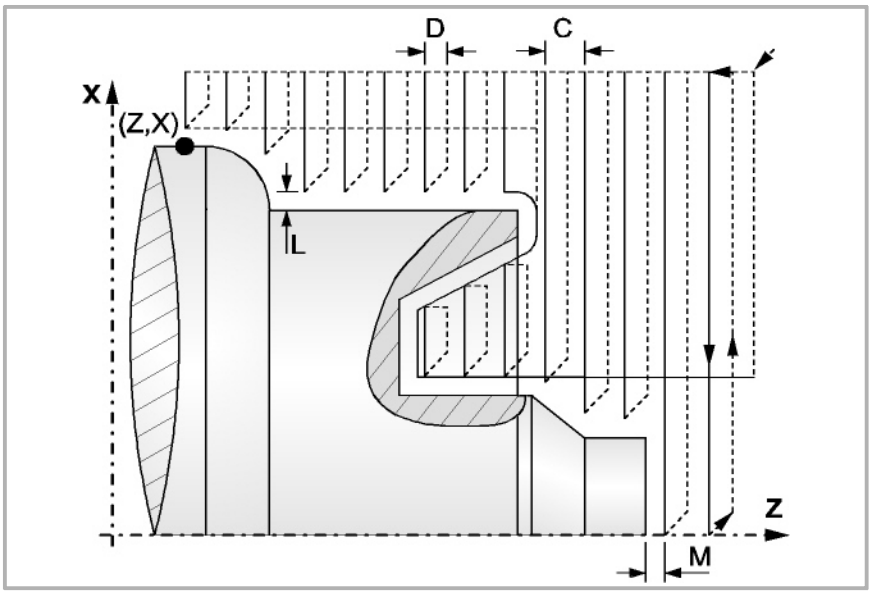
The F, S, T, D and M functions

1.13 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.

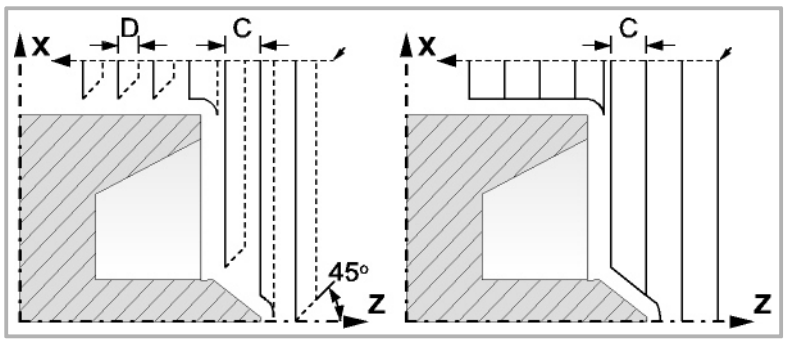
All the machining passes are performed with this value, except the last one which will eliminate the remaining stock.

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 45º 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).

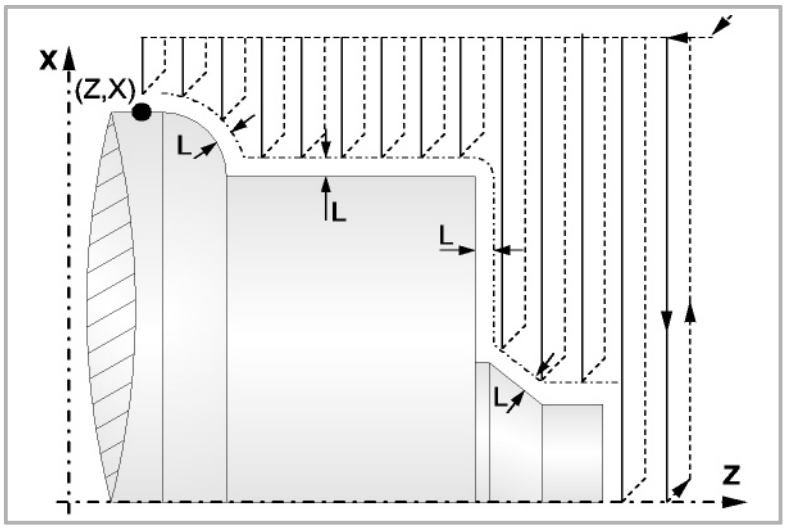
It must be borne in mind that when "D" is not programmed, the cycle execution time is greater; but the amount of stock to be removed on the finishing pass is less.



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

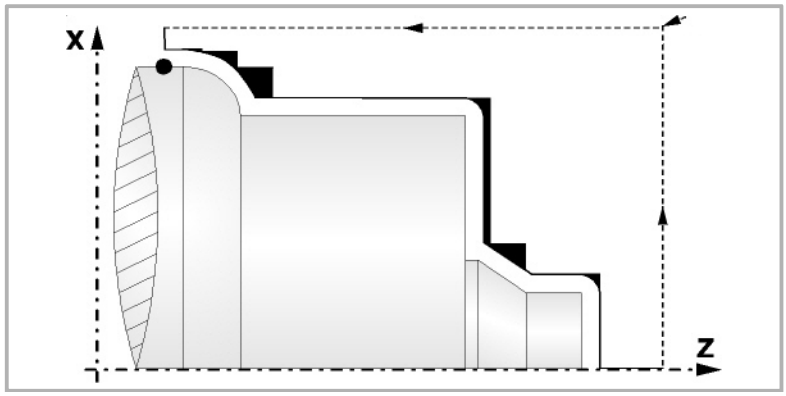
M±5.5 Defines the finishing stock along Z.

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".

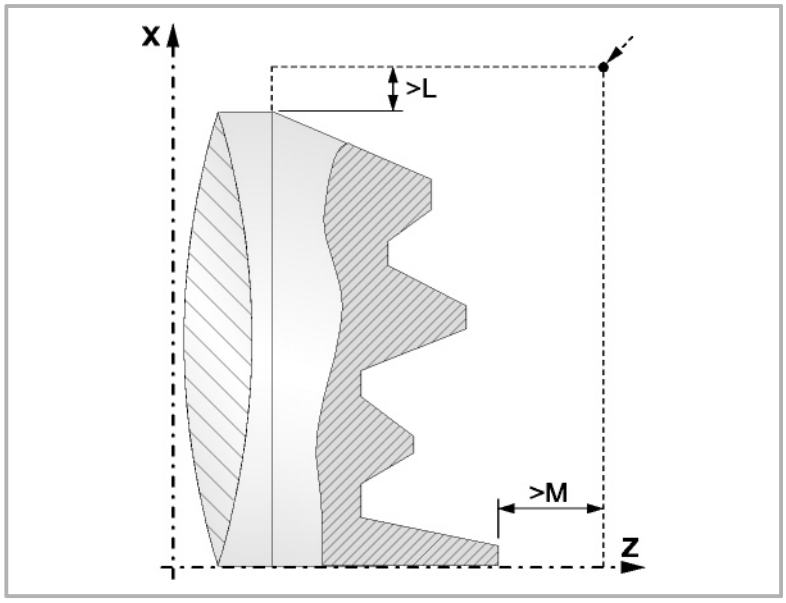
Q Name of the global subroutine, of the program that contains the definition of the profile (parameters "E" and "S") or of the program that contains the local subroutine that contains the profile (parameter "P").

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

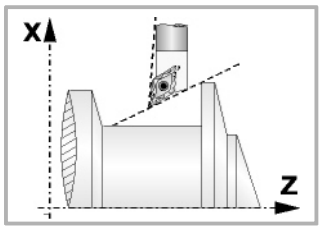
Parameters.	Profile location.
E + S	The profile is defined between the "E" and "S" blocks of the current program.
P	The profile is defined in the local subroutine "P" of the current program.
Q	The profile is defined in global subroutine "Q".
Q + E +S	The profile is defined between the "E" and "S" blocks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



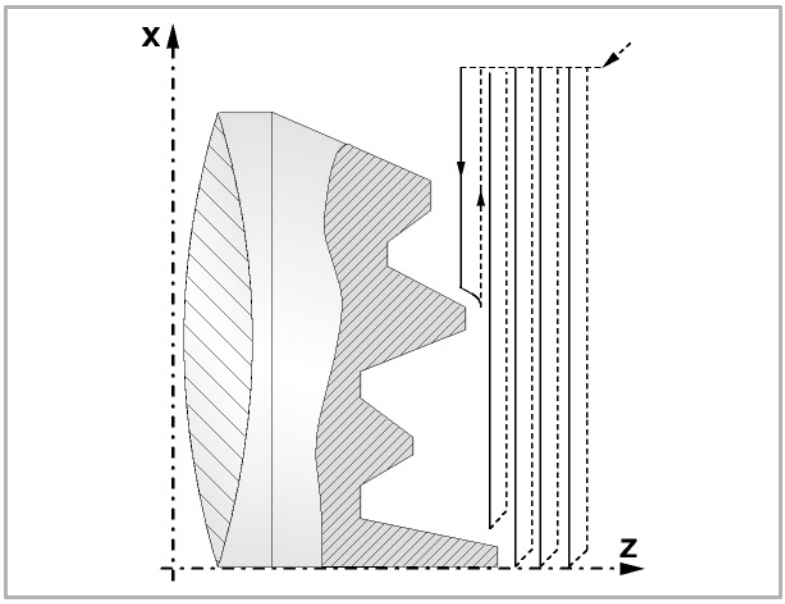
- In those cases when the programmed profile cannot be machined (valleys) with the selected tool, a warning message will be displayed at the beginning of the cycle execution.



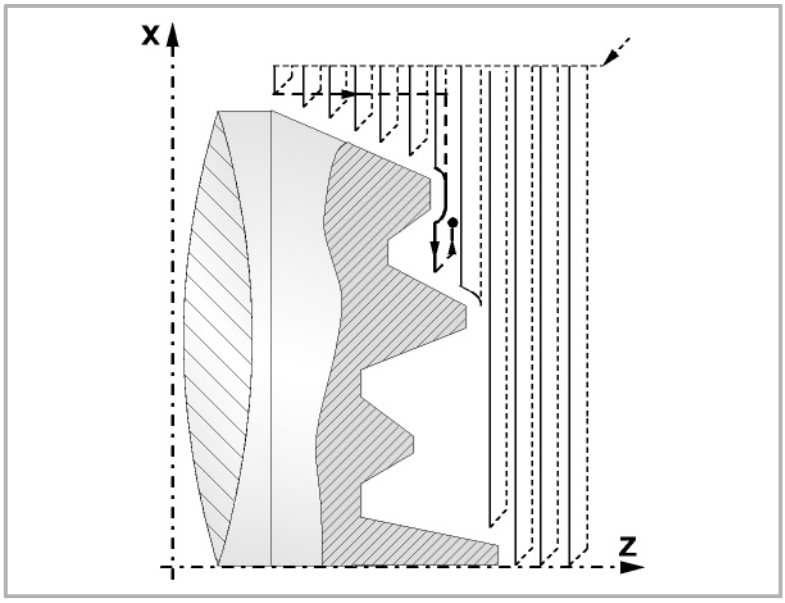
The operator will be able to stop the execution and select the proper tool.

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.

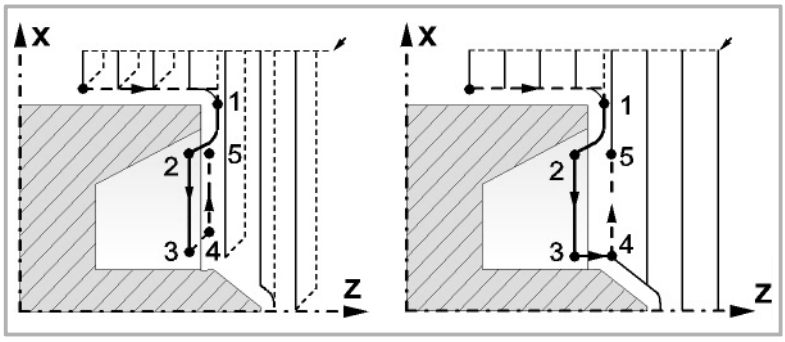
- If when making a roughing pass a channel is detected, the CNC will continue the operation ignoring the channel. The profile may have an unlimited number of channels.



- 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.

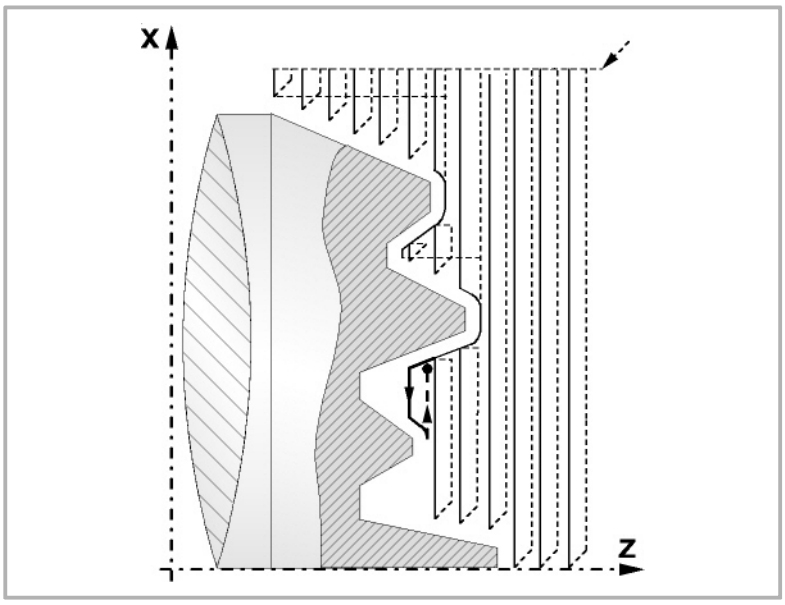


Move “2-3” in G01 at programmed feedrate (F).

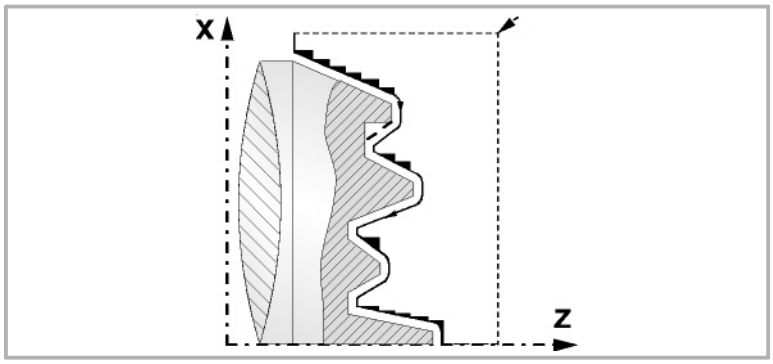
If "D" has been programmed, the 3-4 move is done in rapid (G00); but if "D" has not been programmed, the 3-4 move is done following the profile in G01 at the programmed feedrate (F).

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.

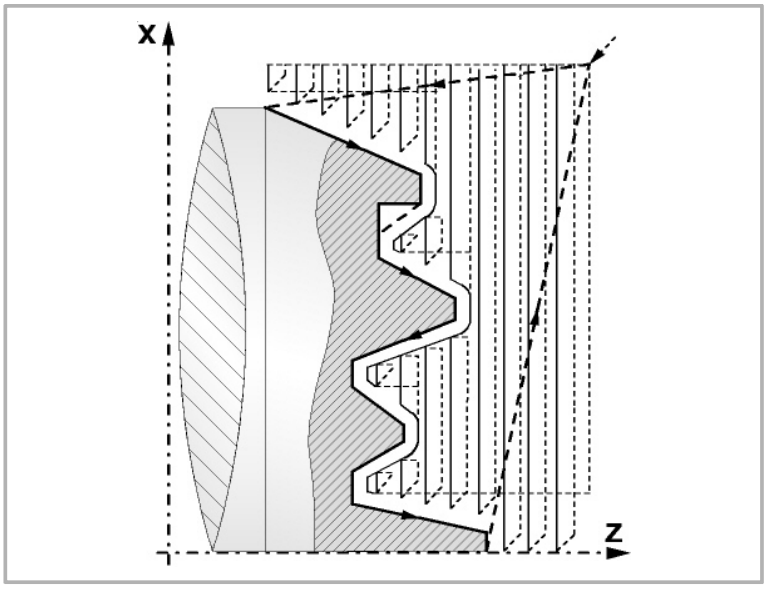


- If a final roughing pass has been programmed, the CNC will run a pass parallel to the profile maintaining an "L" stock at the set feedrate "F". This final roughing pass will eliminate the ridges that were left in the roughing stage.



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

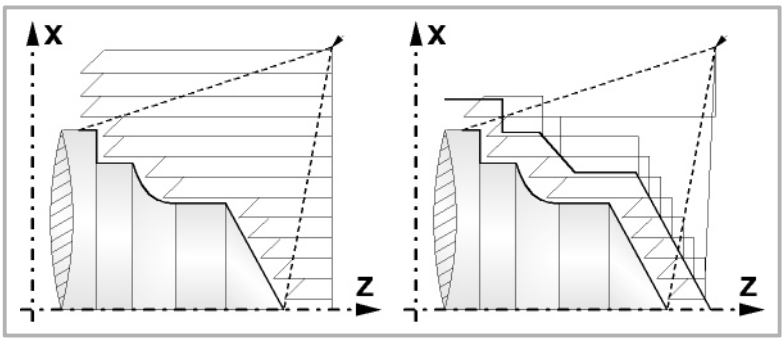
  This profile will coincide or not with the programmed profile depending on whether there are areas not accessible to the selected tool.



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

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

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

G05 Round corner

G07 Square corner

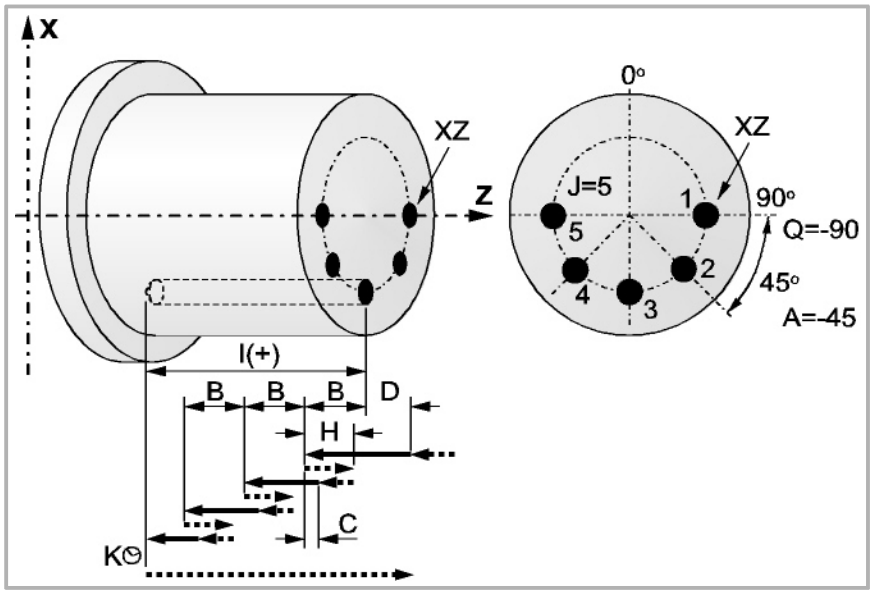
G50 Controlled corner rounding

The F, S, T, D and M functions

1.14 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.

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.

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).

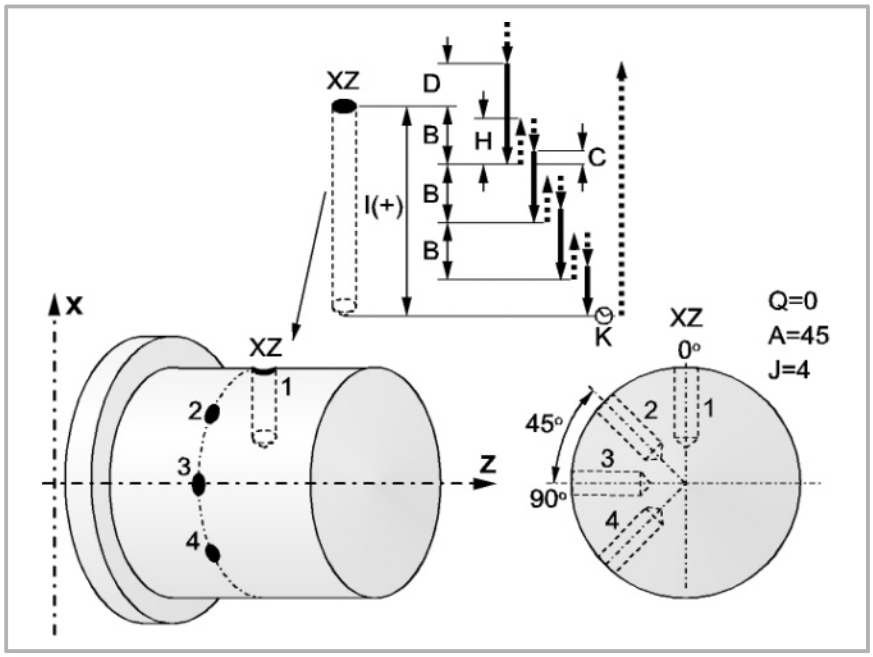
Machining considerations

- 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.

1.15 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.

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.

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).

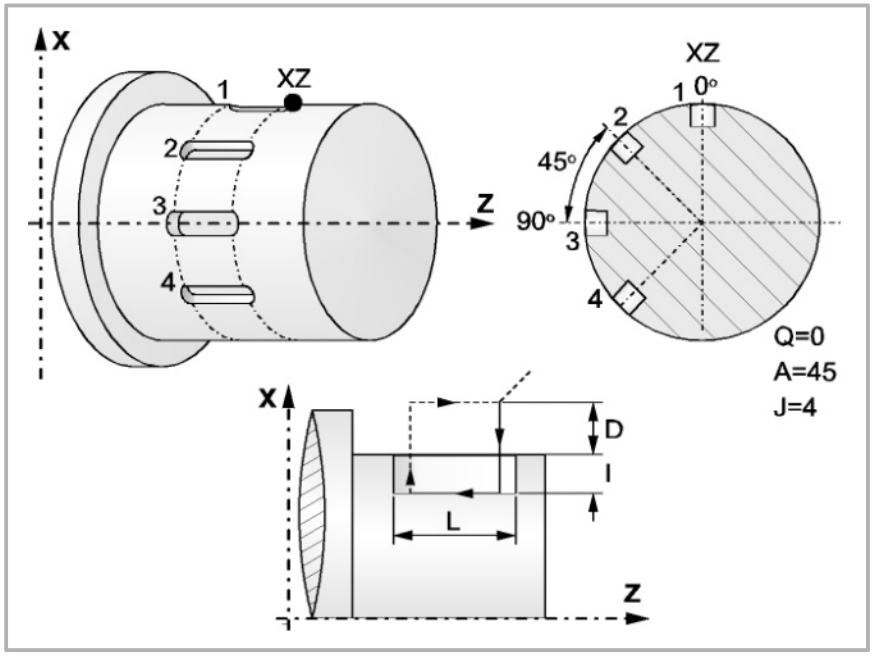
Machining considerations

- 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.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

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.

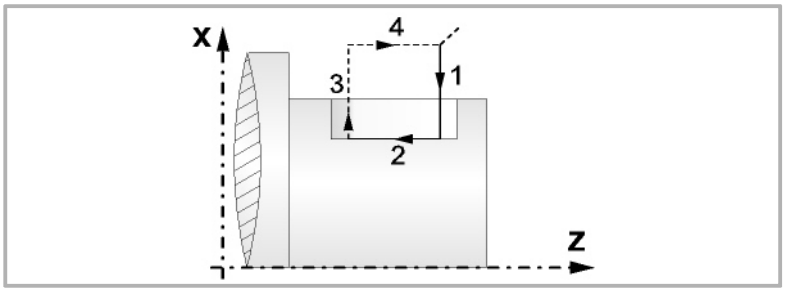
Basic operation

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.

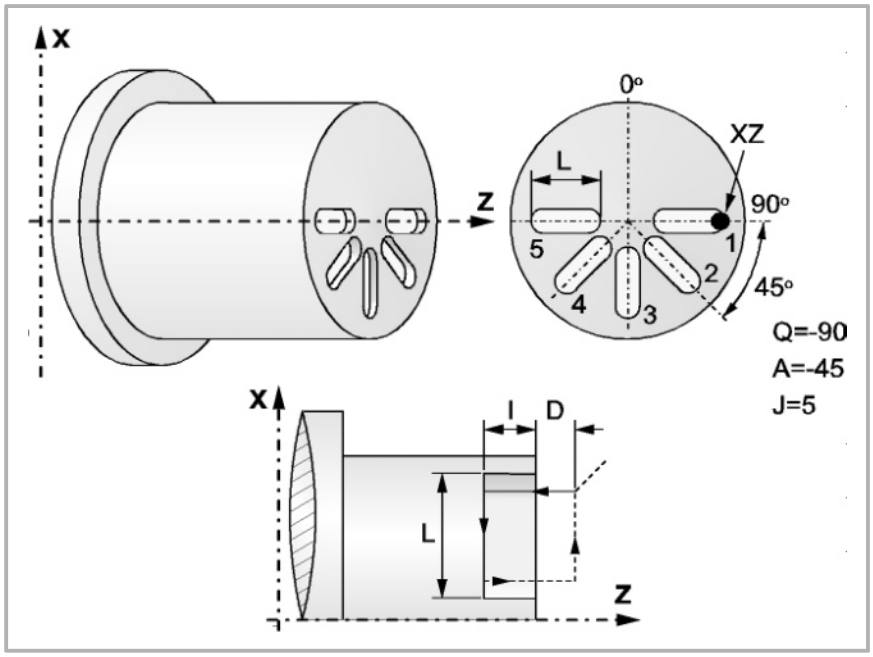
6 The live tool stops.

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".

1.17 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±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, 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±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±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 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±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.

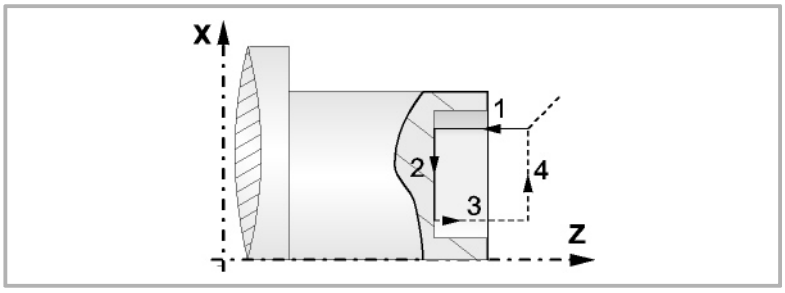
Basic operation

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

FAGOR AUTOMATION

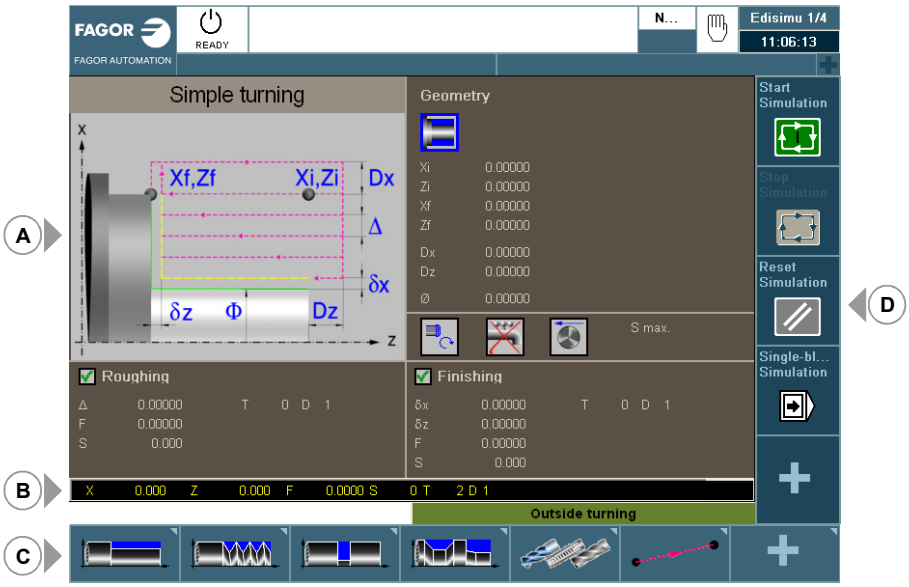
CNCelite
8058 8060
8065

REF. 2109

CYCLE EDITOR

2

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.

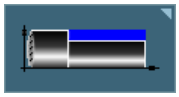
B Teach-in mode.

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.

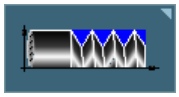
Select the machining cycles.

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.

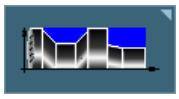


Tapping.

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

Grooving.

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



Profiles.

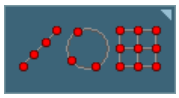
Point-to-point turning, profile turning, ZC plane profile and XC plane profile.

Z axis machining.

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

Positioning.

Positioning and positioning with M functions.



Multiple machining.

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

Activating the Teach-in mode.

The "+" softkey shows the softkey to activate the teach-in mode that allows jogging the axes of the machine and entering the actual (real) position of the axes into the data. See *"2.2 Teach-in mode."* on page 103.

Configuring the cycle editor.

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

Accessing the probe cycles of the lathe model (if available).

The "+" softkey displays the softkey to access the probe cycles of the lathe model.

Accessing the machining cycles.

The "+" softkey displays the softkey to access the machining cycles of the lathe model.

Accessing the machining cycles of the milling model (if available).

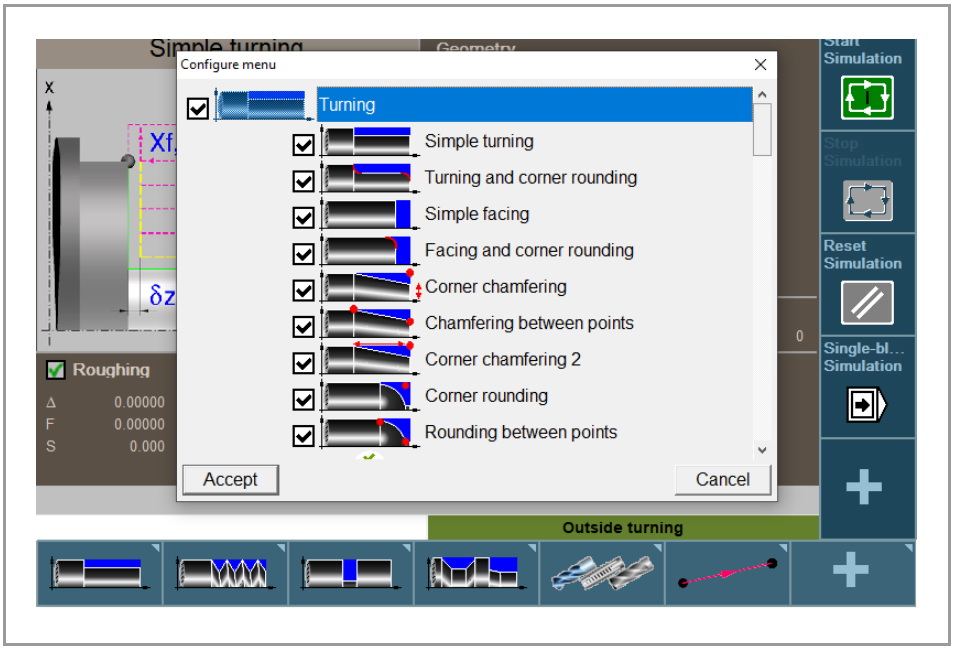
The "+" softkey displays the softkey to access the machining cycles of the milling model. These cycles are only available if “Combined Machine” software option is active.

Accessing the probe cycles of the milling model (if available).

The "+" softkey displays the softkey to access the probe cycles of the milling model. These cycles are only available if “Combined Machine” software option is active.

Hide the Fagor cycles.

The "+" softkey shows the softkey for configuring which Fagor cycles (machining and probe cycles) are visible or not. The menu showing the softkey can also remove the softkeys from the main menu.



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.

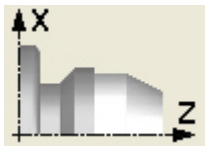
Enable the programming of M functions in the canned cycles, to execute them before each machining operation. This permits, for example, to execute subroutines associated with M functions before the various operations.

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.

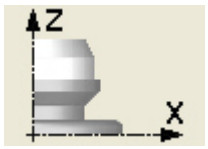
On the cycle screens, the display must be activated in order to see and define the M function data; otherwise, the data will not be displayed.

Select the graphics for a vertical lathe.

Enable the cycles for vertical lathe.

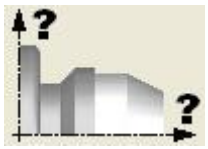


Cycle editor configured for horizontal lathe.



Cycle editor configured for vertical lathe.

Select axis configuration.



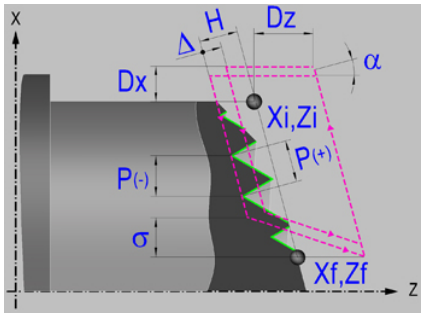
Setting an axis configuration for the cycle editor. The defined axis configuration is only valid for making it easier to edit the cycle because it shows the data related to the coordinates according to the chosen axis configuration.

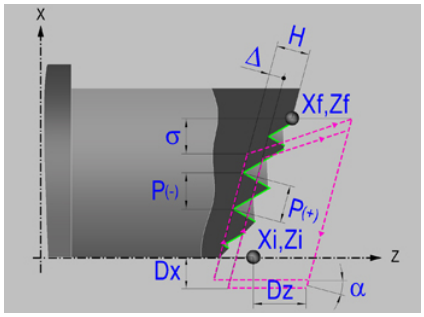
The canned cycles have no work plane associated with them, they are executed in the current active work plane.

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.

2.3 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.

2.4 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.

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 are for the graphics. When quitting the graphics, it will assume as the new display area the one calculated last.

Window for simulation and data editing

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

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

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

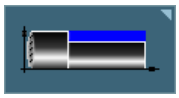
Summary of the hotkeys while simulating a cycle.

[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.

CANNED CYCLES OF THE EDITOR. TURNING AND FACING.

3

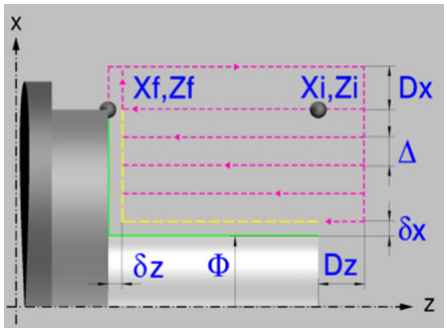
When pressing the softkey, the editor shows the cycle used last. When pressing the same softkey again, the menu shows all the cycles of the group.

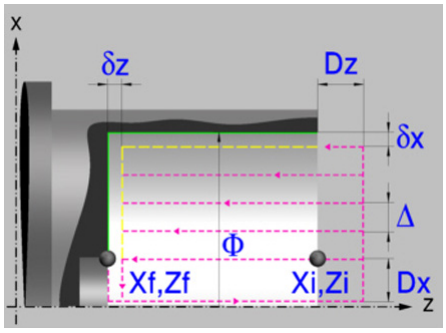


- Simple turning.
- Turning with vertex rounding
- Simple facing.
- Facing with vertex rounding
- Vertex chamfer.
- Chamfer between points.
- Vertex rounding.
- Rounding between points.

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.

Xi, Zi Coordinates of the starting point

Xf, Zf Coordinates of the end point. By default, Xf assumes the value set for Xi.

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.

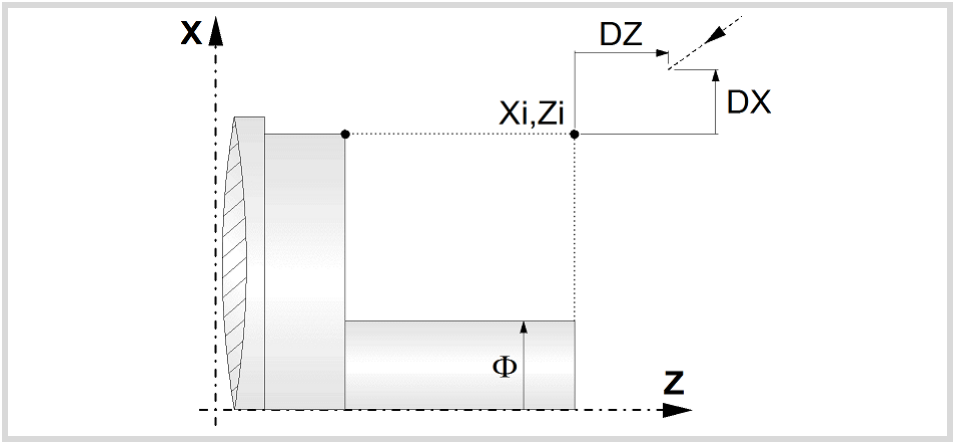
When the surface to be machined is not fully cylindrical, the CNC analyzes the coordinates on X of the starting and end points and takes as starting point on X the coordinate farthest from the final diameter.

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 (Xi, Zi).

Dx, Dz Safety distance.

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.

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.

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.

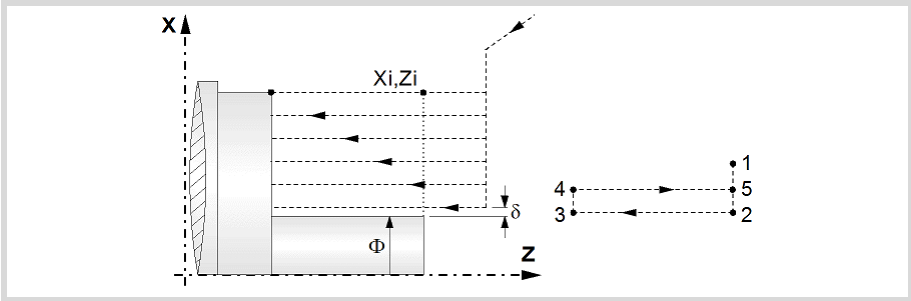
2 The spindle starts with the speed selected and in the direction stated.

3 The tool approaches the starting point (Xi, Zi) in rapid, keeping the selected safety distance (Dx, Dz) 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.

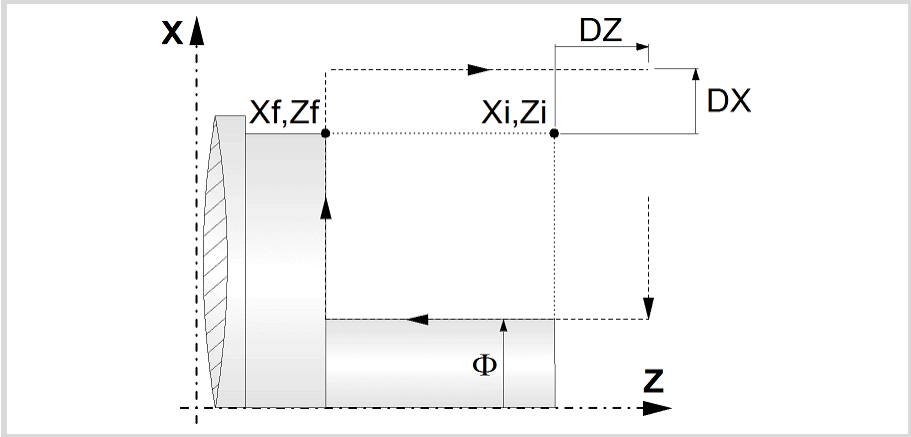
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



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.



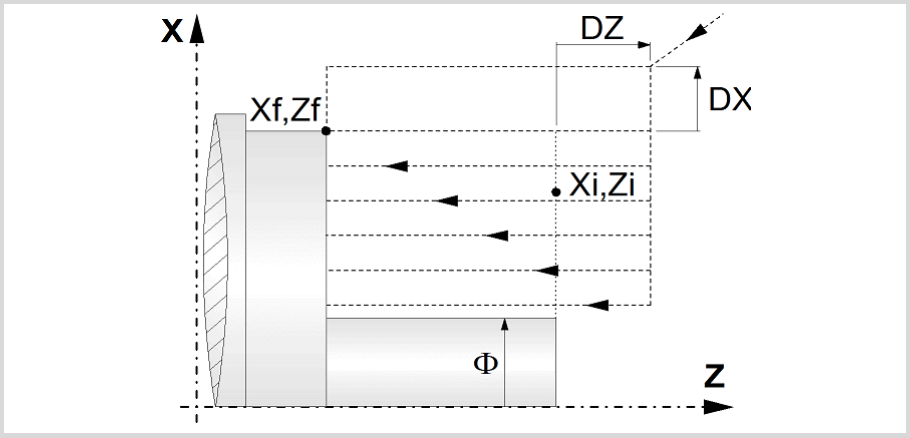
6 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance).

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).

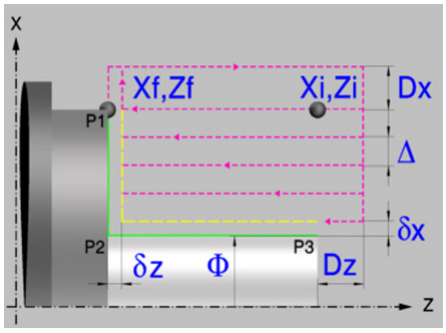
- When the surface to be machined is not fully cylindrical, the CNC analyzes the coordinates on X of the starting and end points and takes as starting point on X the coordinate farthest from the final diameter.

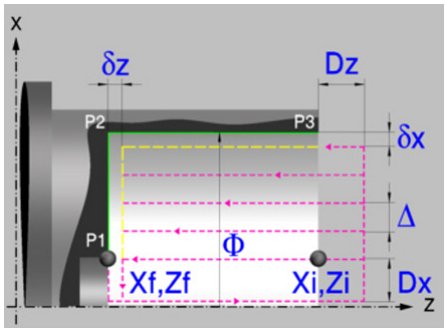


- The roughing operation is carried out in G05, the rounding radius may be modified using the instruction #ROUNDPAR. If #ROUNDPAR is not programmed, it assumes the default rounding radius set by machine parameters.
- The finishing operation is carried out in G07.

3.2 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.

Xi, Zi Coordinates of the starting point

Xf, Zf Coordinates of the end point. By default, Xf assumes the value set for Xi.

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.

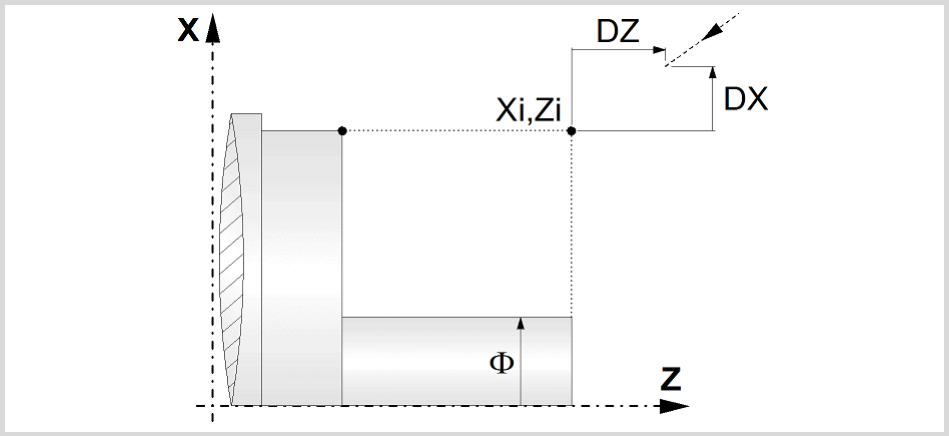
c Size of the chamfer of the pocket corners.

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 (Xi, Zi).

Dx, Dz Safety distance.

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.

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.

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.

3.2.1 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.

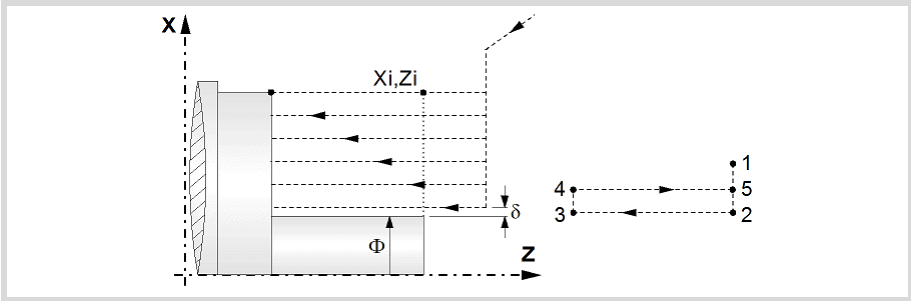
2 The spindle starts with the speed selected and in the direction stated.

3 The tool approaches the starting point (Xi, Zi) in rapid, keeping the selected safety distance (Dx, Dz) 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.

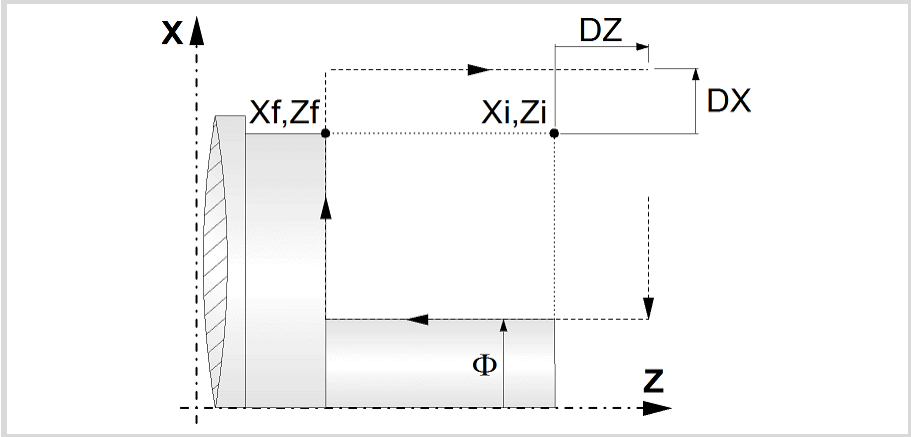
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



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.



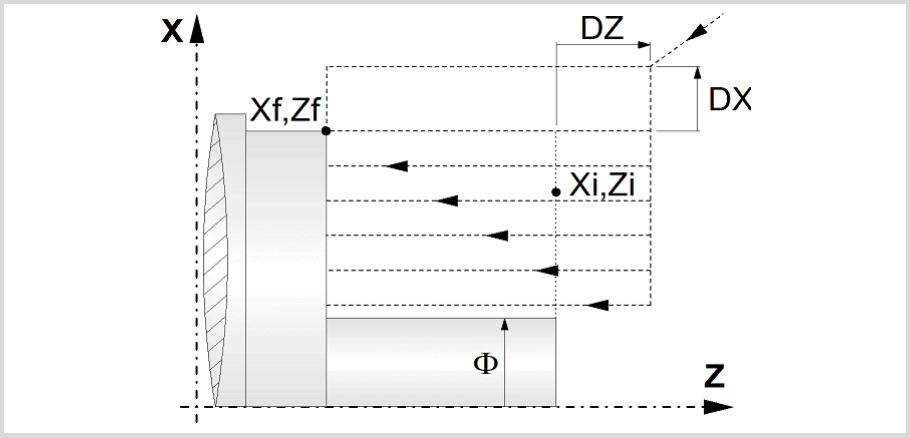
6 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance).

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.

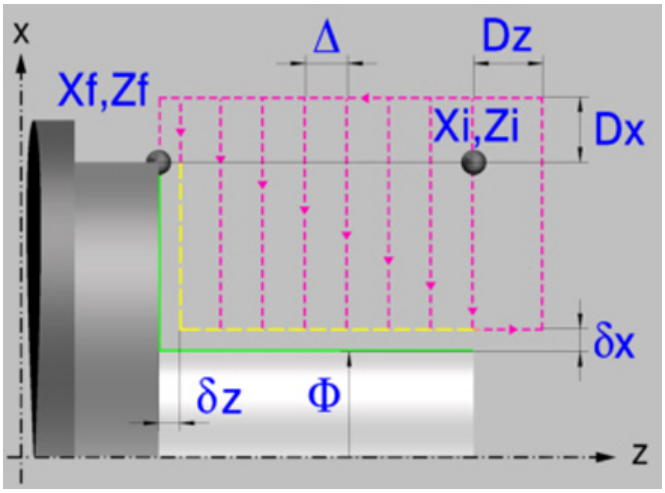
- If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.
- If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the finishing operation, the tool will move to the safety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

- 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.

3.3 Simple facing cycle.



Geometric parameters.

Coordinates of the profile's starting and final points.

Xi, Zi Coordinates of the starting point

Xf, Zf Coordinates of the end point. By default, Xf assumes the value set for Xi.

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.

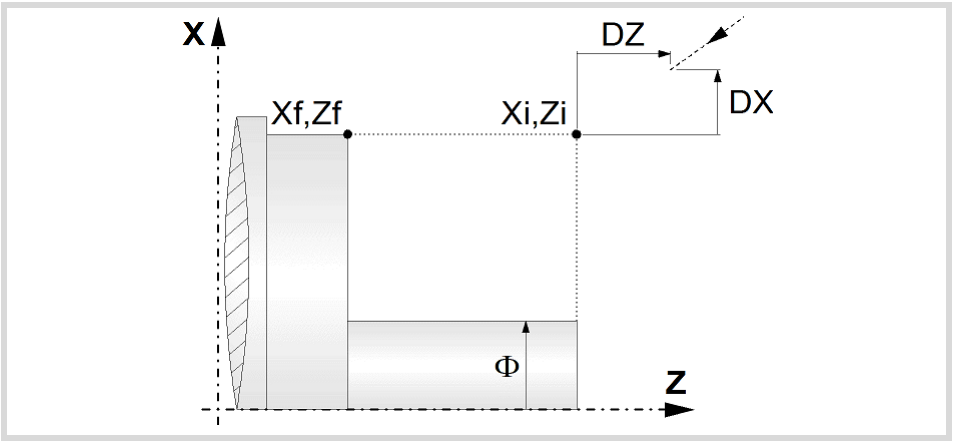
When the surface to be machined is not fully cylindrical, the CNC analyzes the coordinates on X of the starting and end points and takes as starting point on X the coordinate farthest from the final diameter.

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 (Xi, Zi).

Dx, Dz Safety distance.

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.

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.

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.

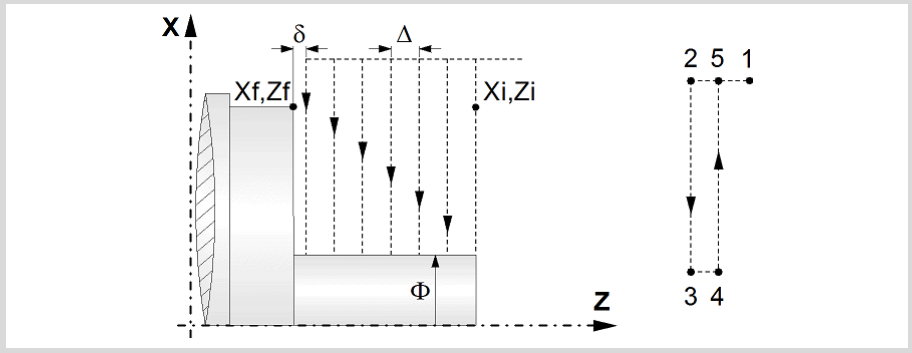
3.3.1 Basic operation.

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 (Xi, Zi) in rapid, keeping the selected safety distance (Dx, Dz) 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.

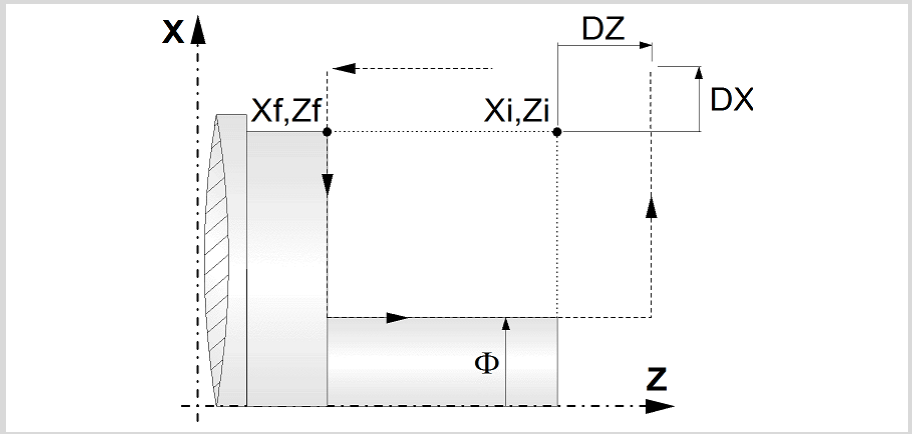
   Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



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.

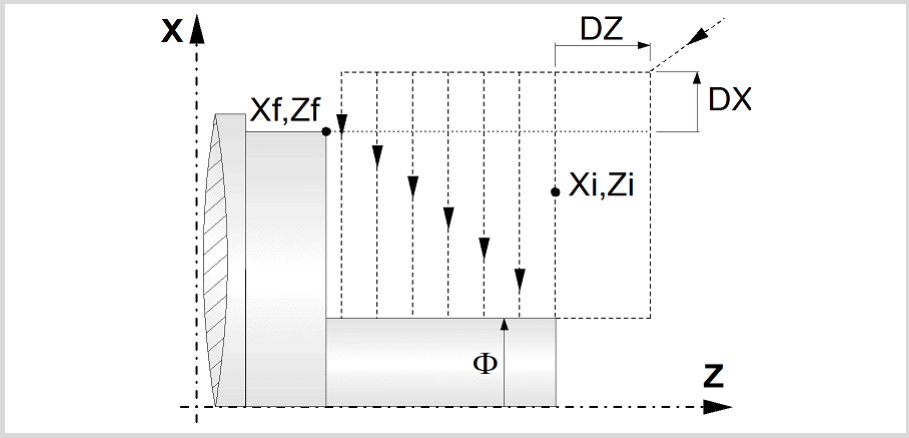


6. Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance).
7. The CNC will not stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

Considerations.

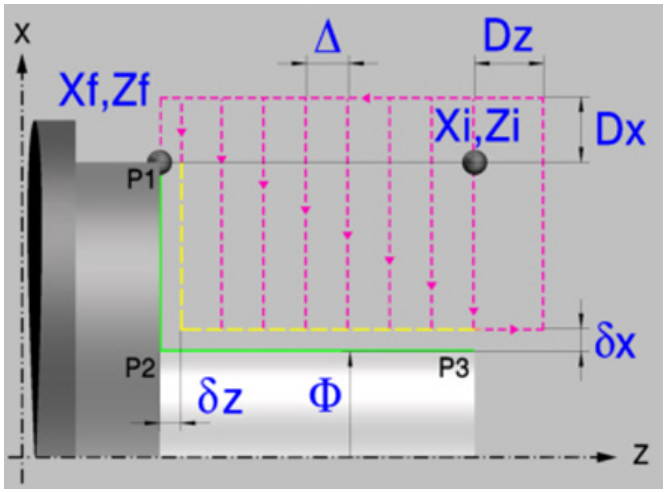
- If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.
- If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the finishing operation, the tool will move to the safety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

- 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.

3.4 Facing cycle with vertex rounding.



Geometric parameters.

Coordinates of the profile's starting and final points.

Xi, Zi Coordinates of the starting point

Xf, Zf Coordinates of the end point. By default, Xf assumes the value set for Xi.

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.

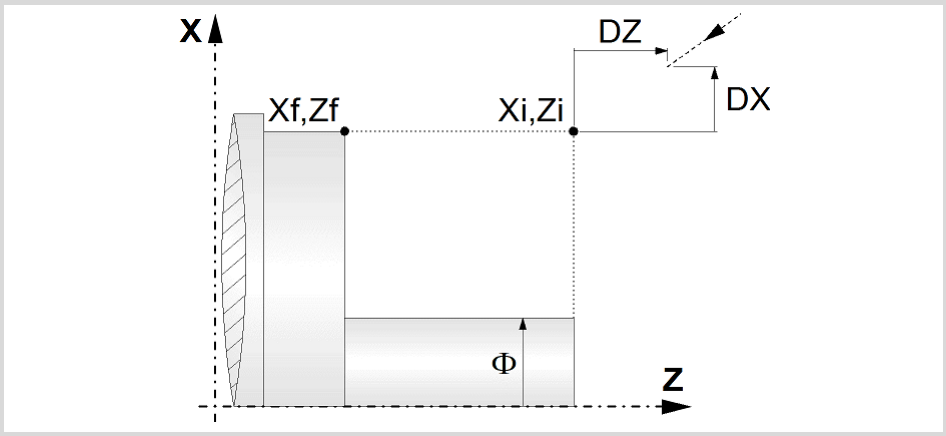
c Size of the chamfer of the pocket corners.

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 (Xi, Zi).

Dx, Dz Safety distance.

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.

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.

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.

3.4.1 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.

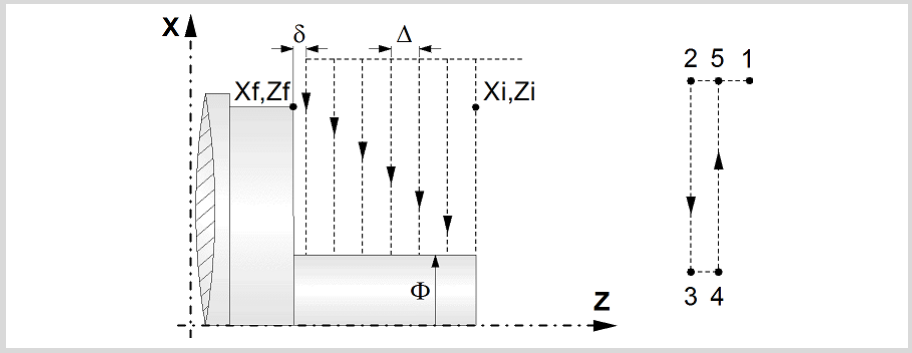
2 The spindle starts with the speed selected and in the direction stated.

3 The tool approaches the starting point (Xi, Zi) in rapid, keeping the selected safety distance (Dx, Dz) 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.

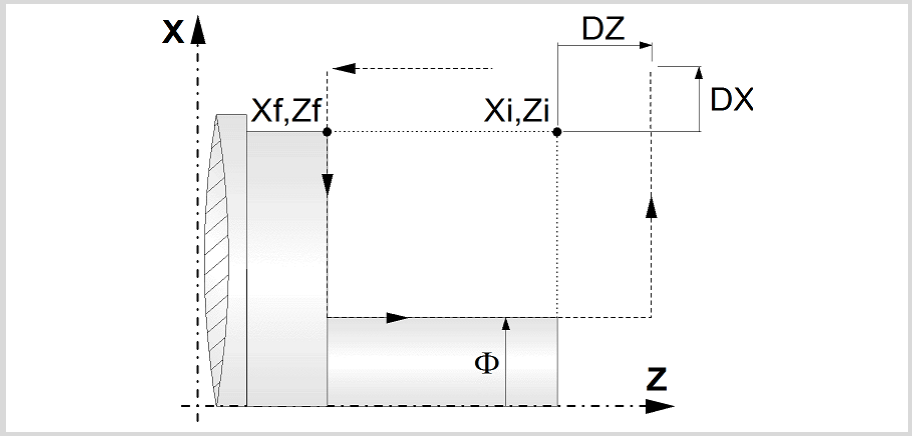
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



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.



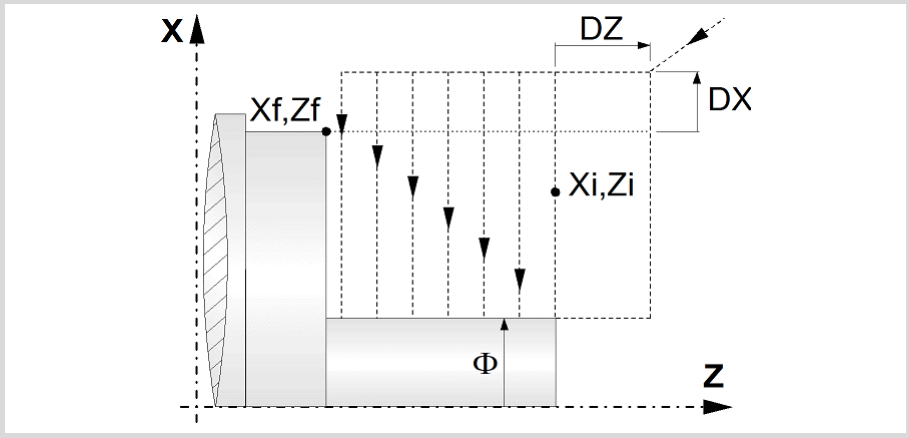
6 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance).

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).

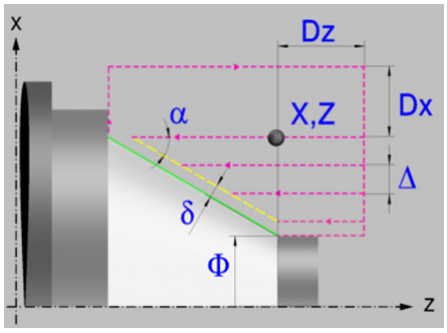
- When the surface to be machined is not fully cylindrical, the CNC analyzes the coordinates on X of the starting and end points and takes as starting point on X the coordinate farthest from the final diameter.

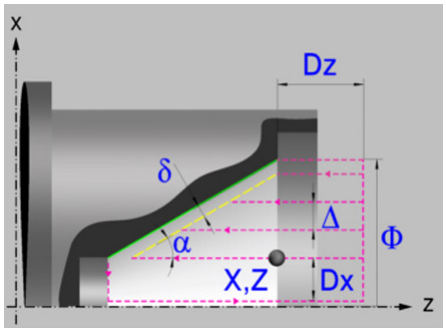


- The roughing operation is carried out in G05, the rounding radius may be modified using the instruction #ROUNDPAR. If #ROUNDPAR is not programmed, it assumes the default rounding radius set by machine parameters.
- The finishing operation is carried out in G07.

3.5 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.

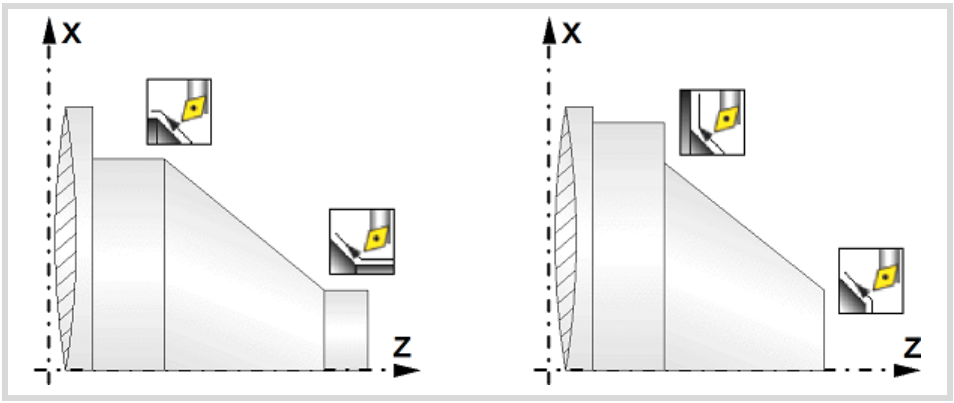
Initial concave corner.

Initial convex corner.

Shape of the part after the tapered segment.

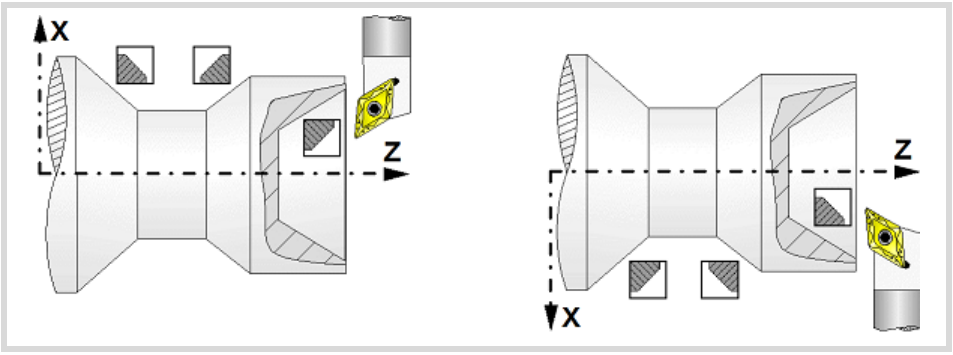
Final concave corner.

Final convex corner.



Work quadrant.

Orientation of the corner to be machined.



Coordinates of the starting point

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

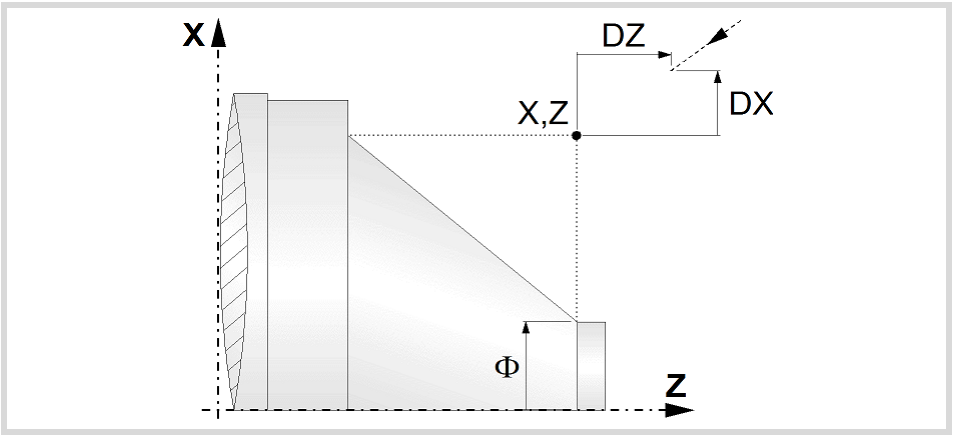
The X axis coordinate must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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.

Coolant status.

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

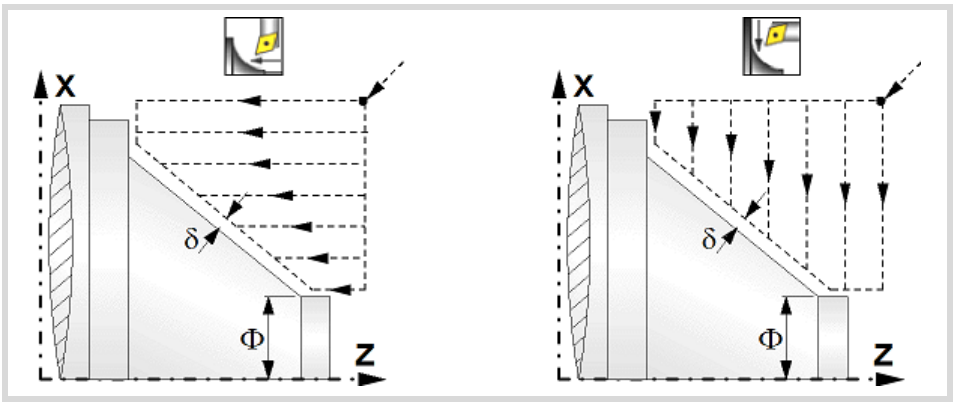
Coolant OFF. The CNC outputs the M9 function to the PLC.

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.

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.

3.5.1 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.

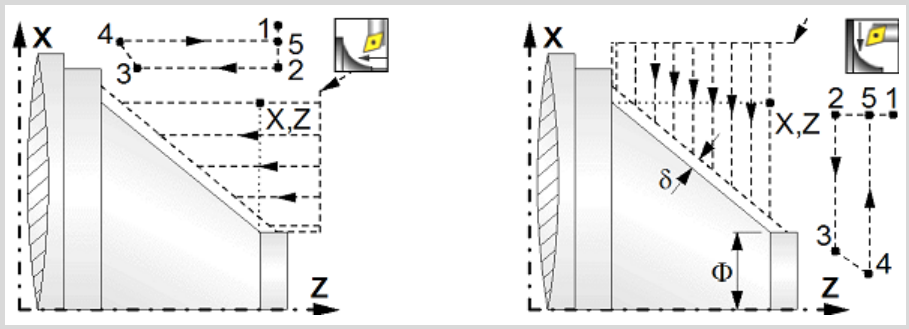
2 The spindle starts with the speed selected and in the direction stated.

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.

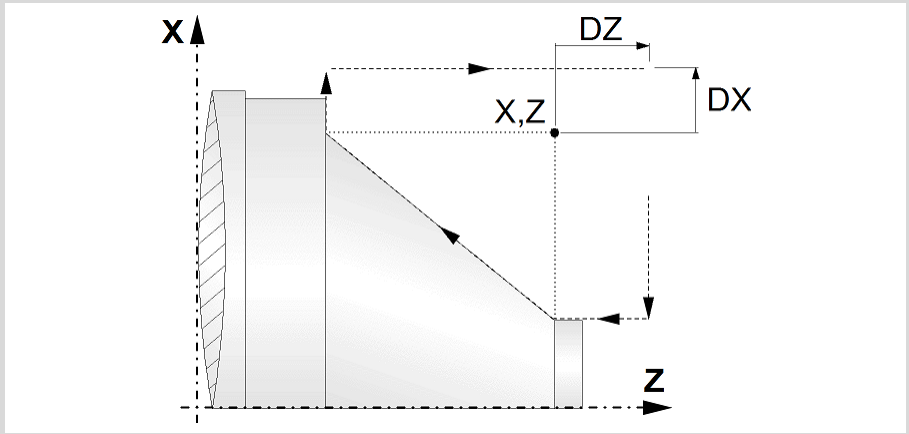
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



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.

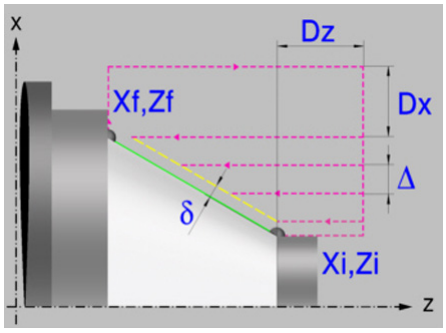
- If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.
- If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the finishing operation, the tool will move to the safety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

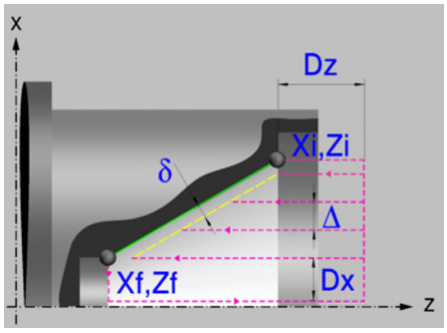
- The roughing operation is carried out in G05, the rounding radius may be modified using the instruction #ROUNDPAR. If #ROUNDPAR is not programmed, it assumes the default rounding radius set by machine parameters.
- The finishing operation is carried out in G07.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

3.6 "Chamfering between points" cycle.

The cycle allows outside and inside machining.





Geometric parameters.

Inside or outside machining.

Outside machining.

Inside machining.

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

Shape of the part before the tapered segment.

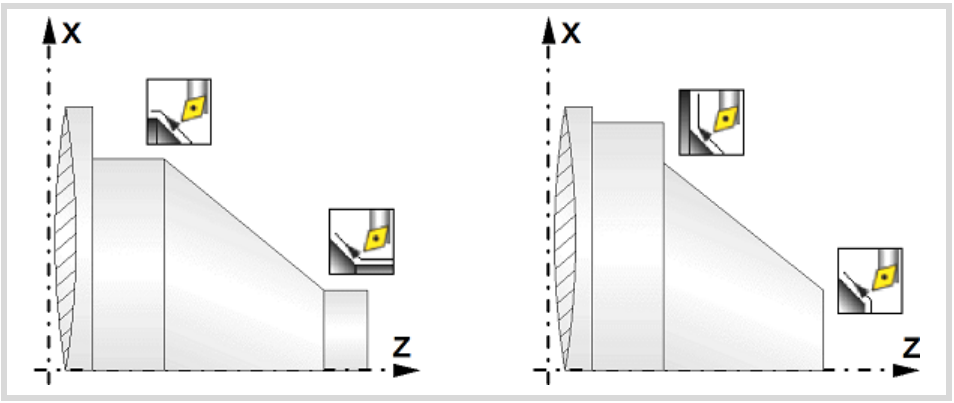
Initial concave corner.

Initial convex corner.

Shape of the part after the tapered segment.

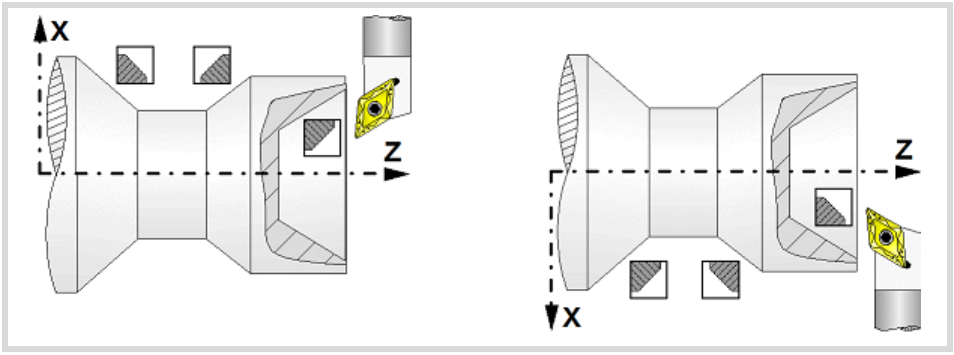
Final concave corner.

Final convex corner.



Work quadrant.

Orientation of the corner to be machined.



Coordinates of the profile's starting and final points.

Xi, Zi Coordinates of the starting point

Xf, Zf Coordinates of the end point.

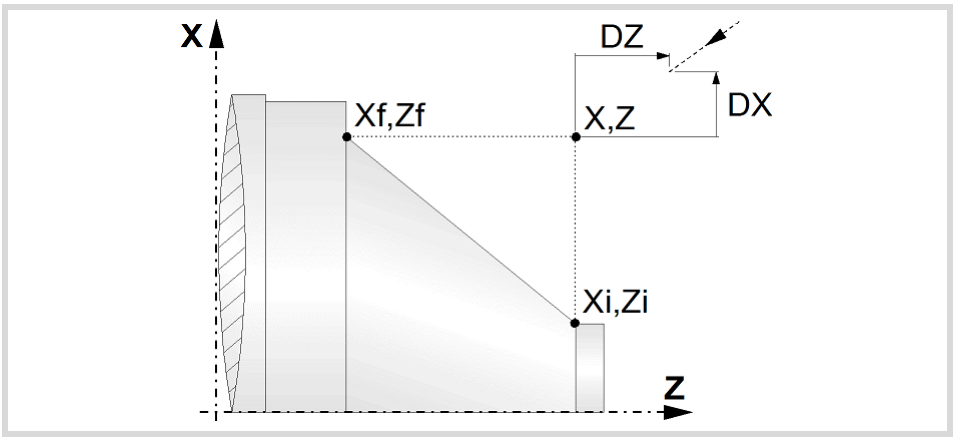
The X axis coordinates must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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.

Dx, Dz 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.

Coolant status.

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

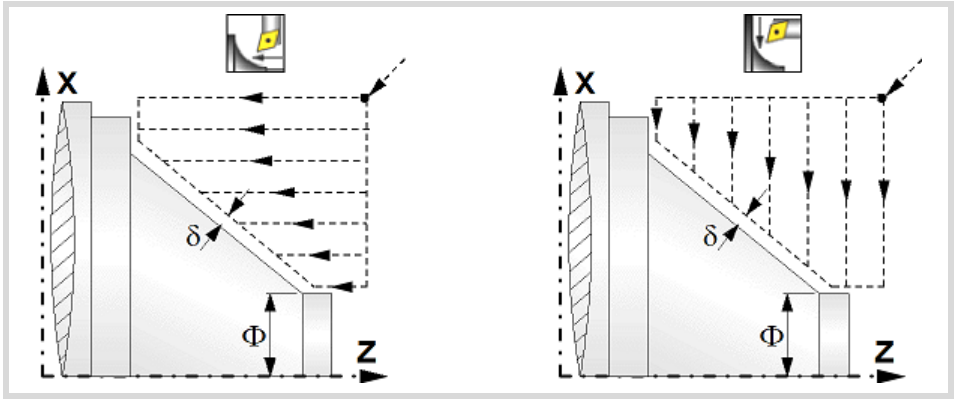
Coolant OFF. The CNC outputs the M9 function to the PLC.

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.

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.

3.6.1 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.

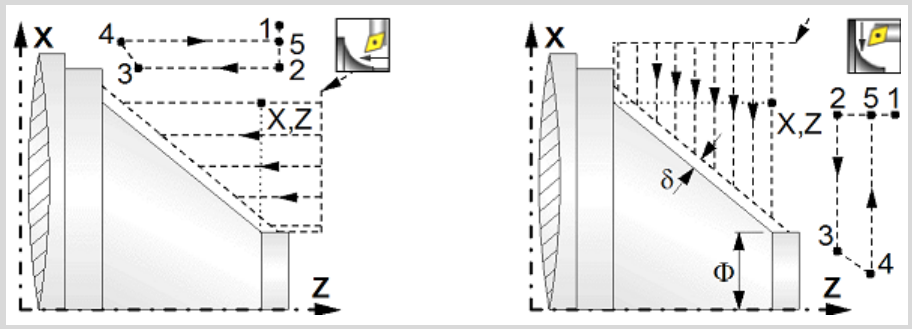
2 The spindle starts with the speed selected and in the direction stated.

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.

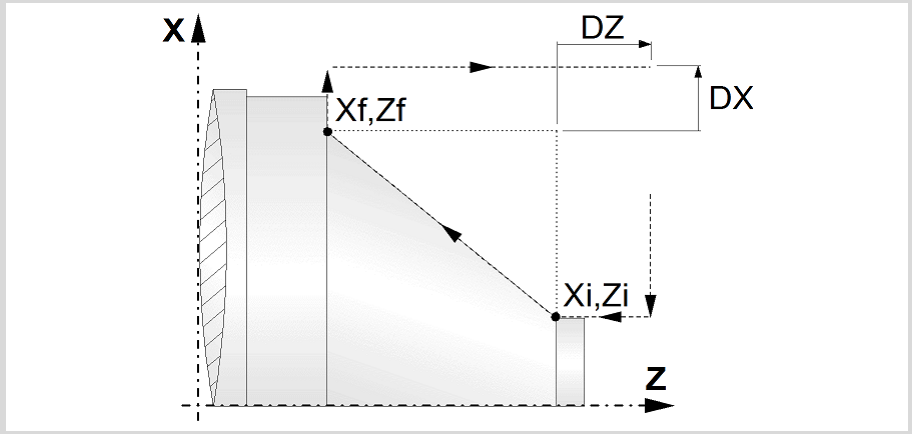
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



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.

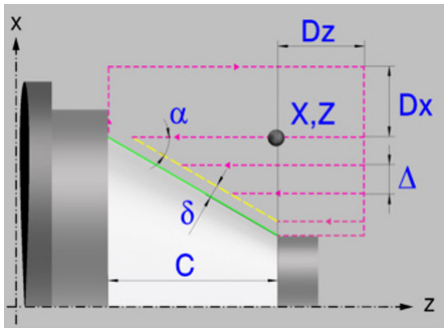
- If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.
- If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the finishing operation, the tool will move to the safety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

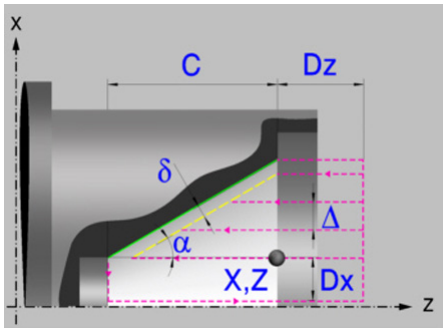
- The roughing operation is carried out in G05, the rounding radius may be modified using the instruction #ROUNDPAR. If #ROUNDPAR is not programmed, it assumes the default rounding radius set by machine parameters.
- The finishing operation is carried out in G07.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

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.

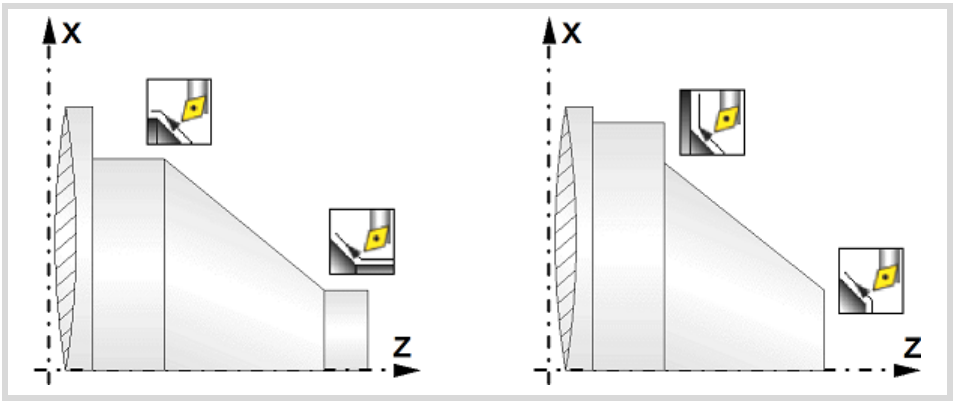
Initial concave corner.

Initial convex corner.

Shape of the part after the tapered segment.

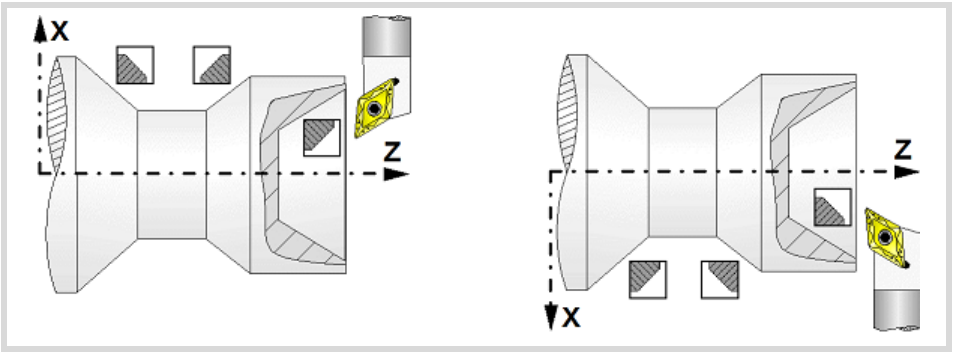
Final concave corner.

Final convex corner.



Work quadrant.

Orientation of the corner to be machined.



Coordinates of the starting point

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

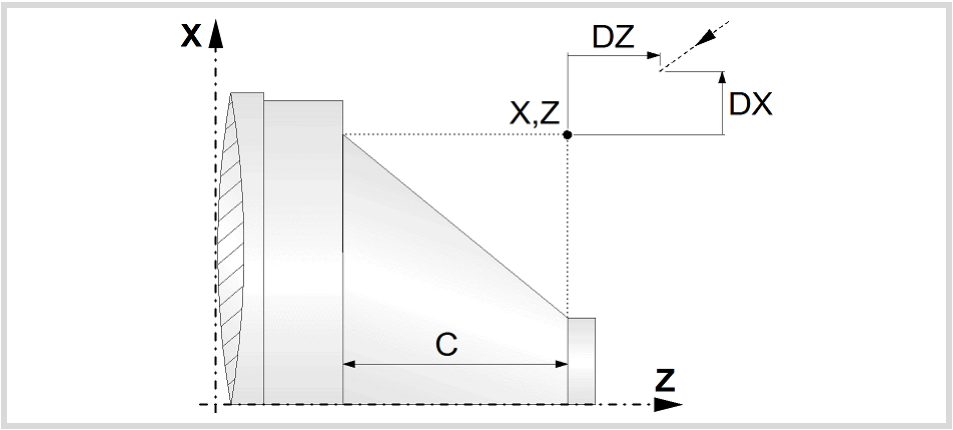
The X axis coordinate must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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.

Spindle clockwise.

Spindle counterclockwise.

Coolant status.

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

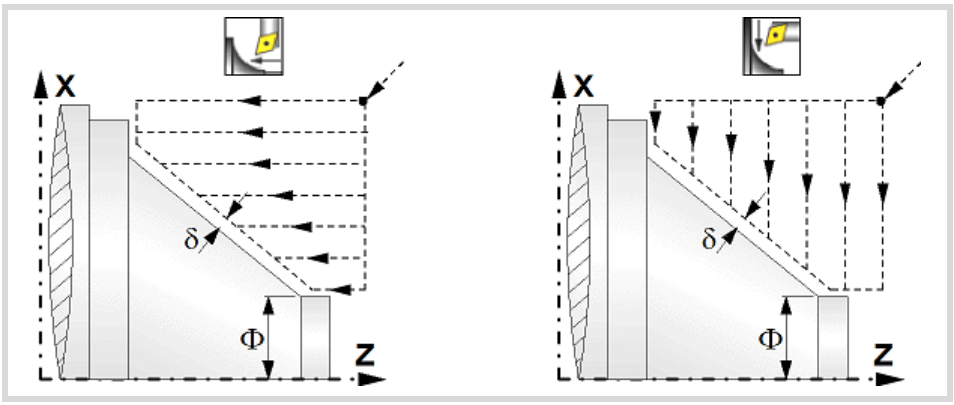
Coolant OFF. The CNC outputs the M9 function to the PLC.

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.

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.

3.7.1 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.

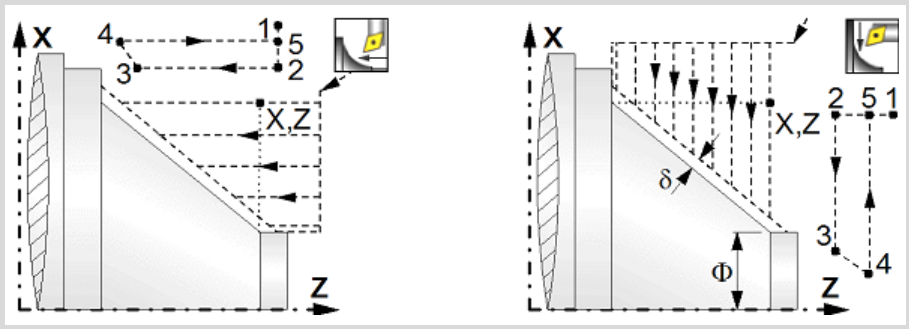
2 The spindle starts with the speed selected and in the direction stated.

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.

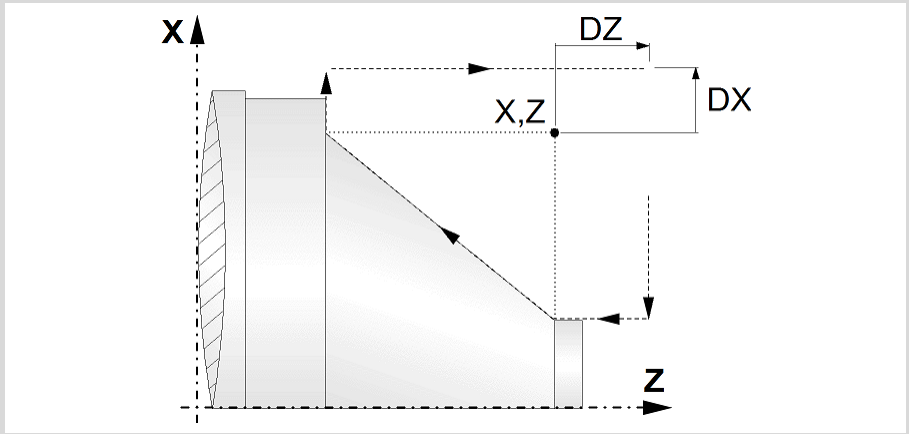
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



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.

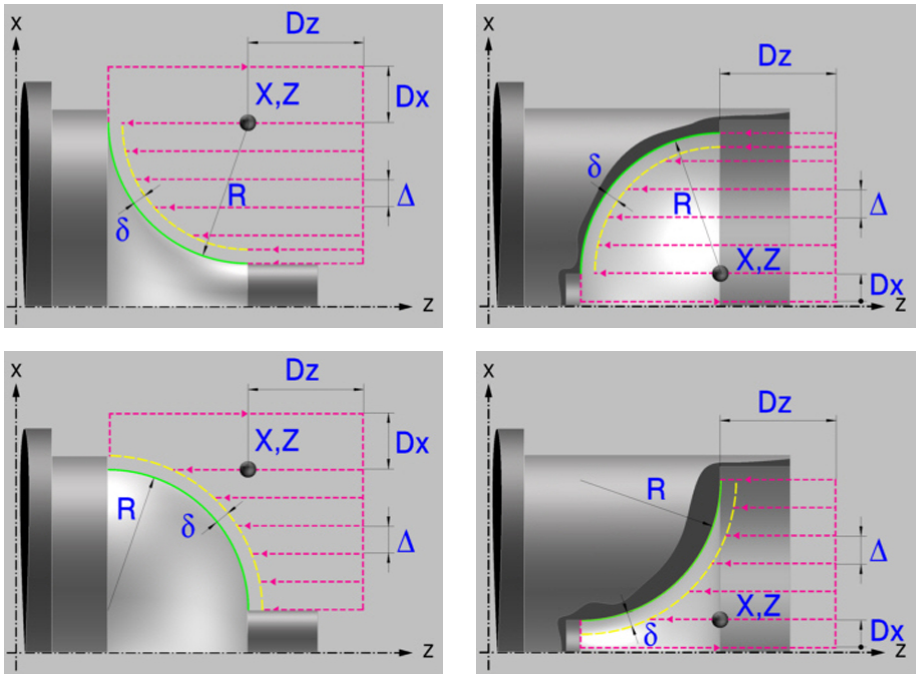
- If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.
- If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the finishing operation, the tool will move to the safety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

- 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.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

3.8 Vertex rounding 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.

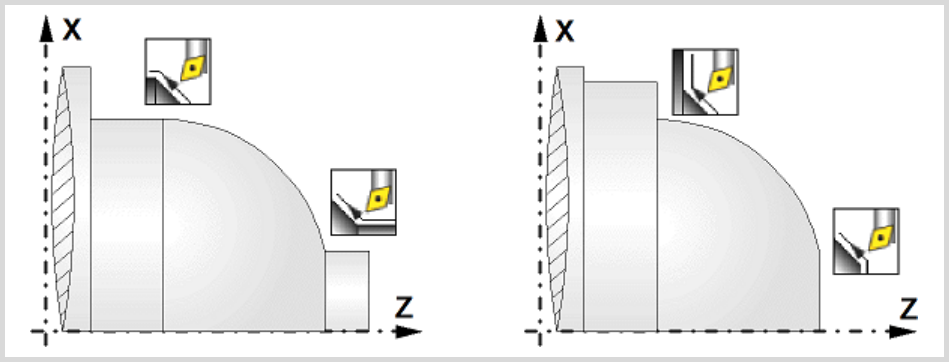
Initial concave corner.

Initial convex corner.

Shape of the part after the tapered segment.

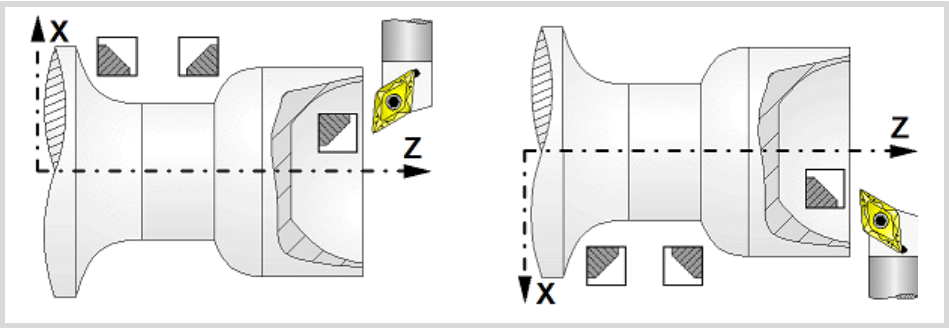
Final concave corner.

Final convex corner.



Work quadrant.

Orientation of the corner to be machined.



Coordinates of the starting point

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

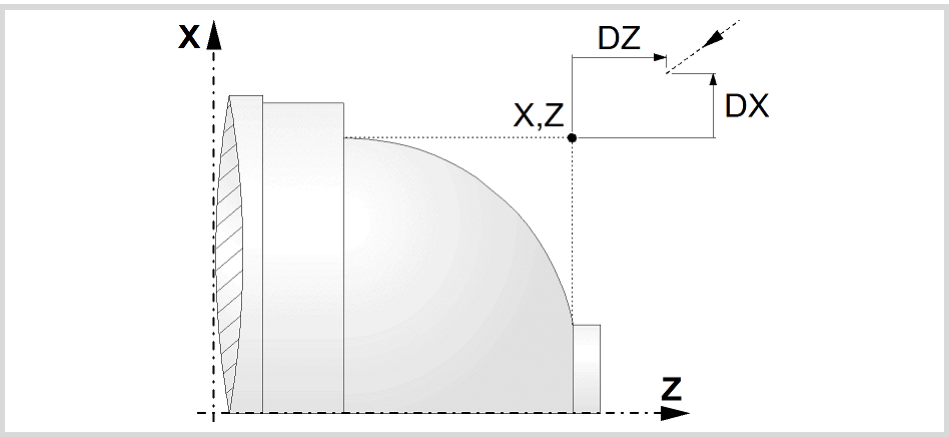
The X axis coordinate must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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.

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.

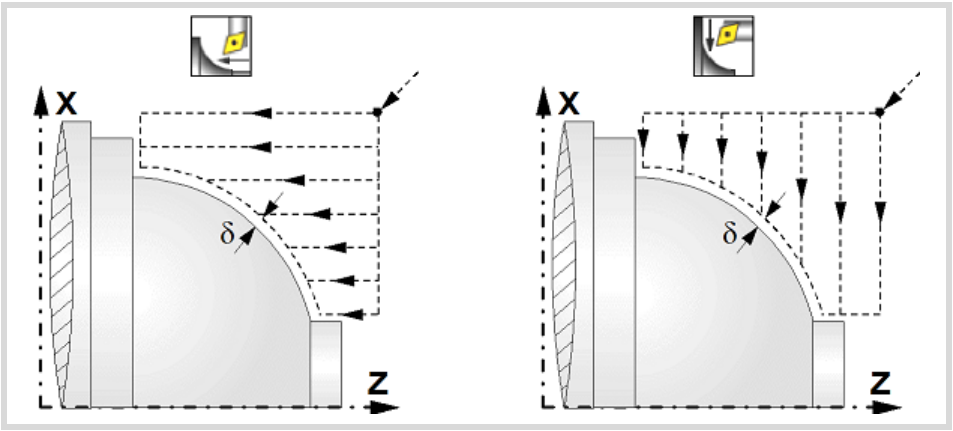
Coolant OFF. The CNC outputs the M9 function to the PLC.

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.

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.

3.8.1 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.

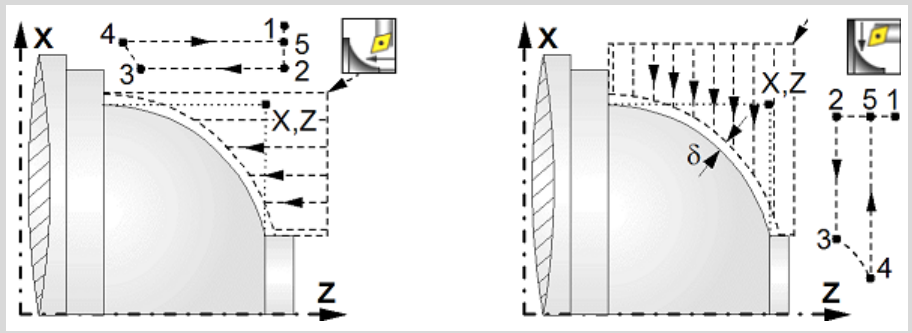
2 The spindle starts with the speed selected and in the direction stated.

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.

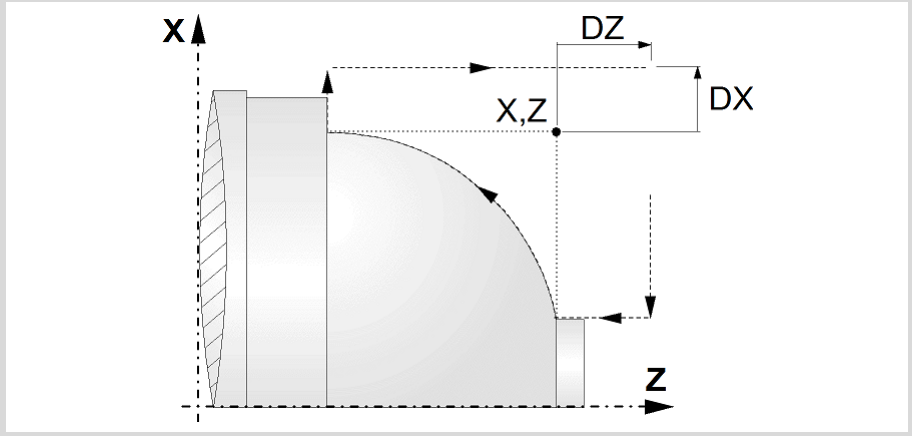
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



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).

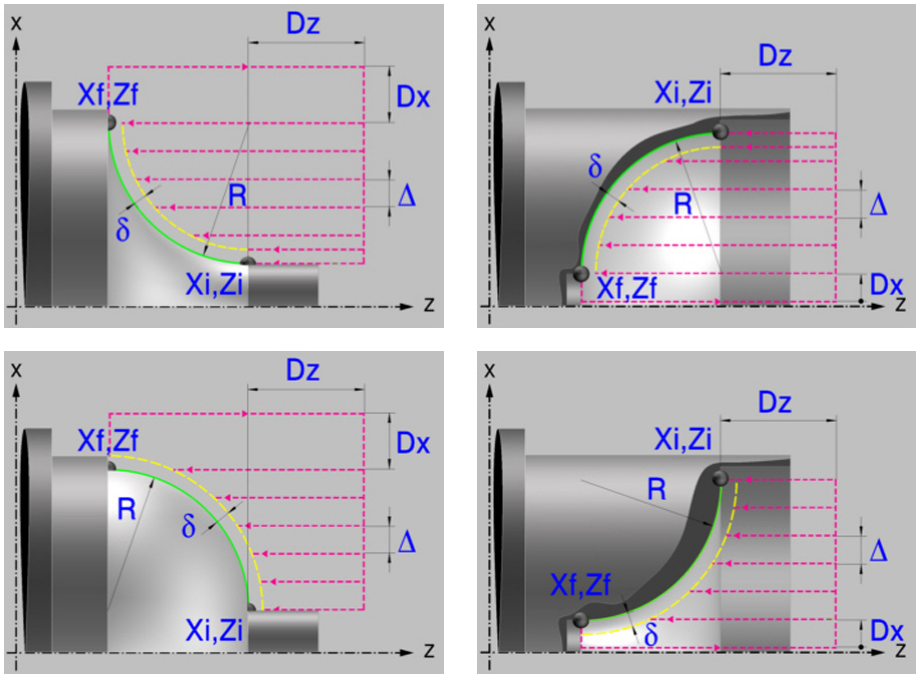
Considerations.

- 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).

- 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.

3.9 "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.

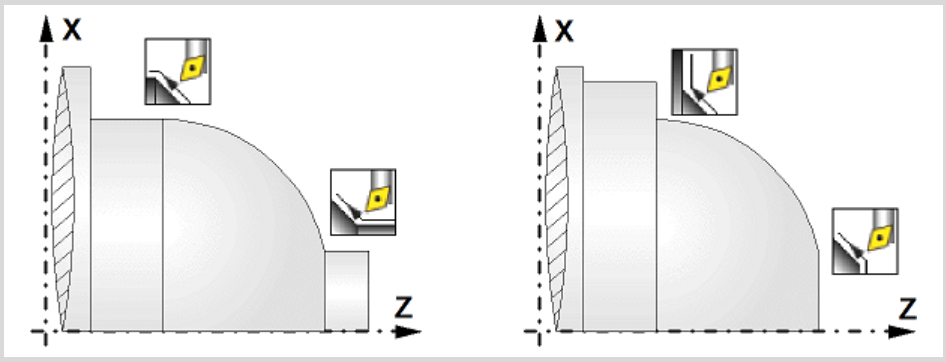
Initial concave corner.

Initial convex corner.

Shape of the part after the tapered segment.

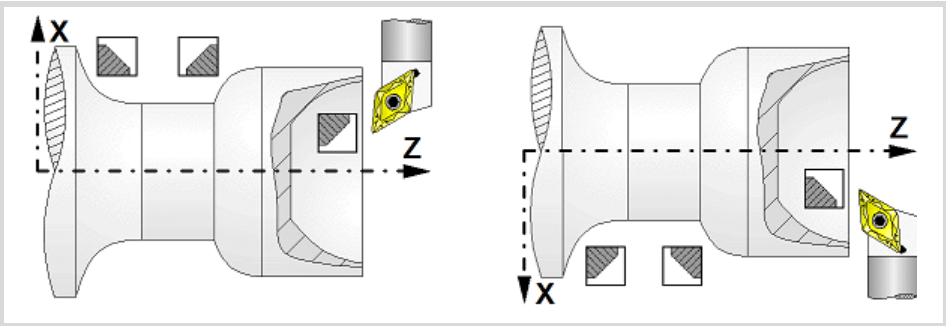
Final concave corner.

Final convex corner.



Work quadrant.

Orientation of the corner to be machined.



Coordinates of the profile's starting and final points.

Xi, Zi Coordinates of the starting point

Xf, Zf Coordinates of the end point.

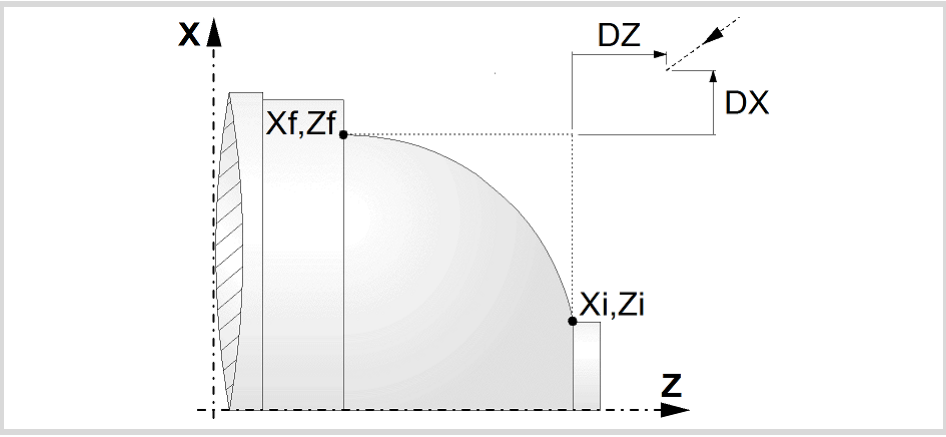
The X axis coordinates must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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.

Dx, Dz Safety distance.

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



Taper length.

R Rounding radius.

Spindle parameters.

Spindle turning direction

The cycle lets select the spindle turning direction.

Spindle clockwise.

Spindle counterclockwise.

Coolant status.

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

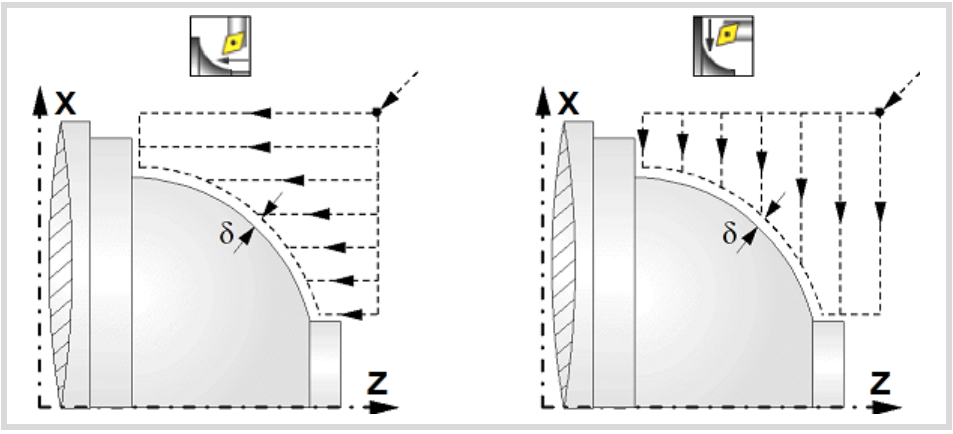
Coolant OFF. The CNC outputs the M9 function to the PLC.

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

Machining direction.

Machining in the turning direction.

Machining in the facing direction.



Spindle work mode; rpm or Constant Surface Speed.

Constant turning speed.

Constant surface speed.

Maximum spindle speed.

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

Spindle range (gear).

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

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.

3.9.1 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.

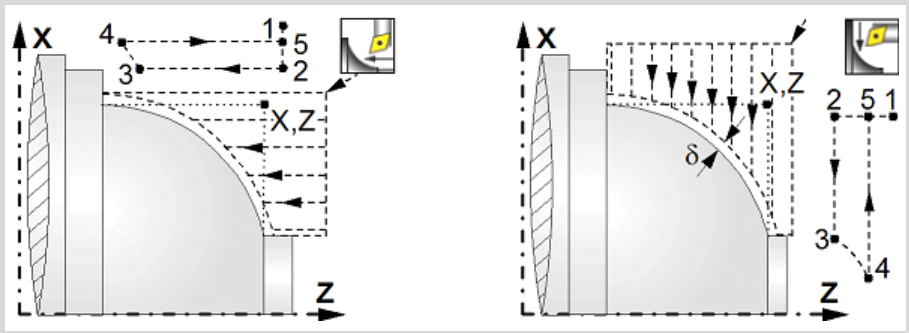
2 The spindle starts with the speed selected and in the direction stated.

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.

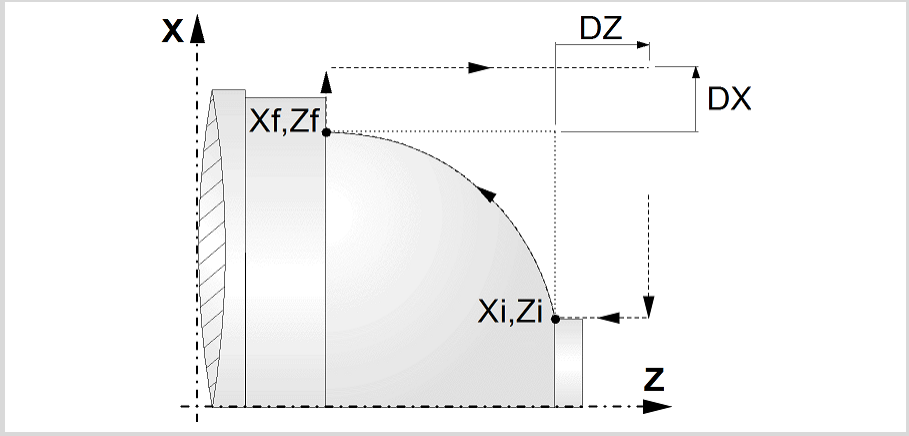
Each roughing pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".



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).

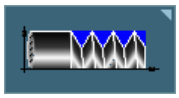
- The roughing operation is carried out in G05, the rounding radius may be modified using the instruction #ROUNDPAR. If #ROUNDPAR is not programmed, it assumes the default rounding radius set by machine parameters.
- The finishing operation is carried out in G07.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

CANNED CYCLES OF THE EDITOR. TAPPING.

4

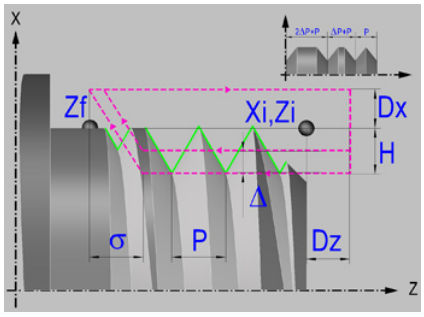
When pressing the softkey, the editor shows the cycle used last. When pressing the same softkey again, the menu shows all the cycles of the group.

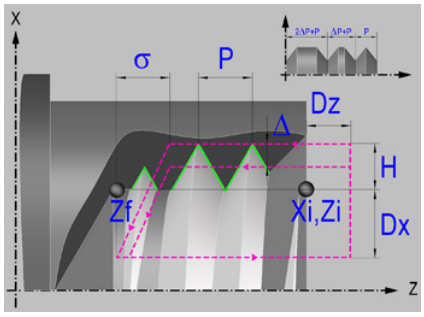


- Longitudinal threading.
- Taper threading.
- Face threading.
- Thread repair.
- Multi-entry (start) threading.

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.

Xi, Zi Coordinates of the starting point

Zf Z coordinate of the end point.

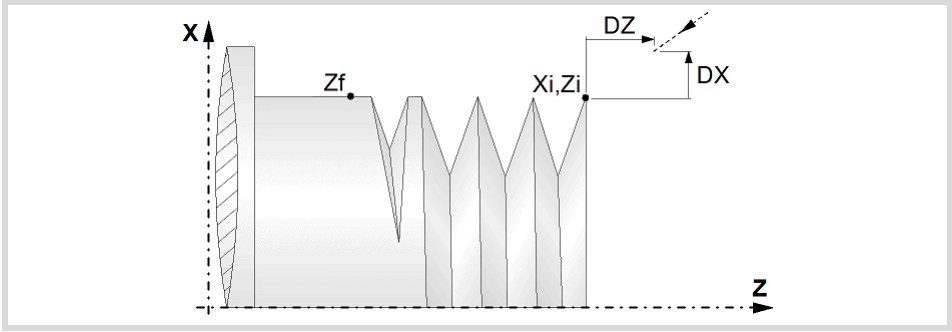
The X axis coordinate must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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 (Xi, Zi).

Dx, Dz 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.

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).

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.

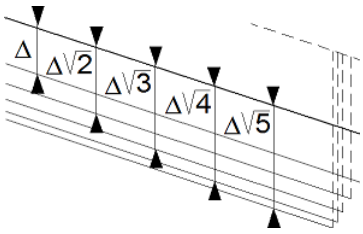
S Spindle speed.

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, Δ√2, Δ√3, Δ√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 σ=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 σ=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.

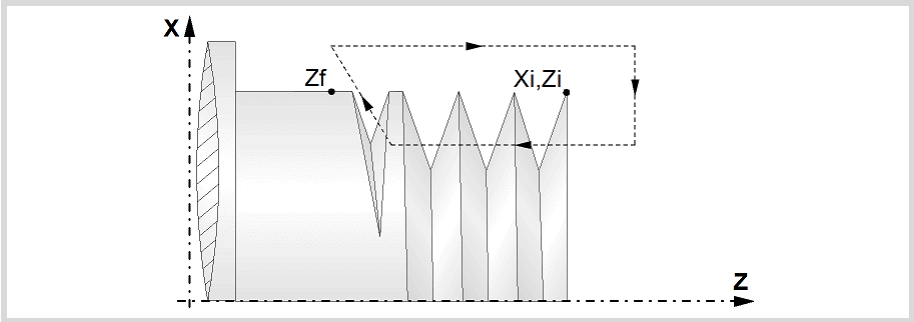
Deactivating the execution of M functions before machining.

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

4.1.1 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.

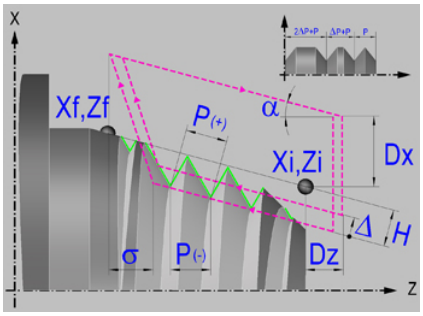
The electronic threading is carried out at 100% of the calculated feedrate and these values cannot be modified from the CNC's operator panel or via PLC. If the OEM allows it, (parameter THREADOVR), the user can modify the speed override from the operator panel and, in that case, the CNC will adapt the feedrate automatically respecting the thread pitch. In order to be able to modify the override, the active feed forward must be higher than 90%.

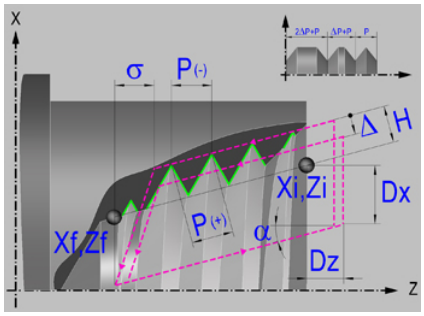
Neither the feedrate override nor the spindle speed override can be changed in the last pass; this pass will be run with the override set in the previous pass.

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).

4.2 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.

Xi, Zi	Coordinates of the starting point
Xf, Zf	Coordinates of the end point. By default, Xf assumes the value set for Xi.
α	Thread angle.
Δz	Thread length.

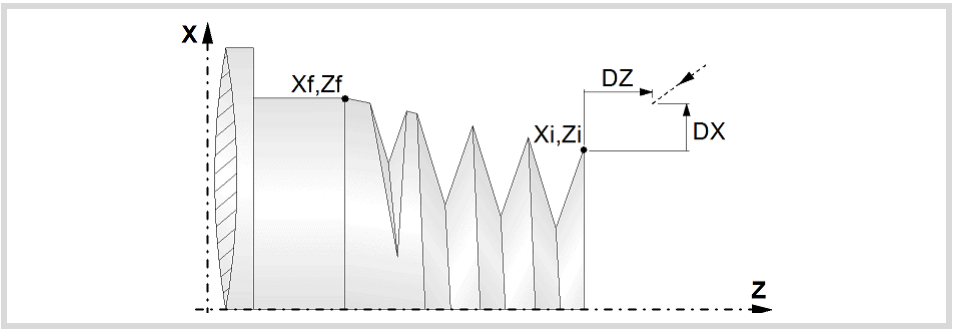
The X axis coordinates must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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 (Xi, Zi).

Dx, Dz	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.

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.

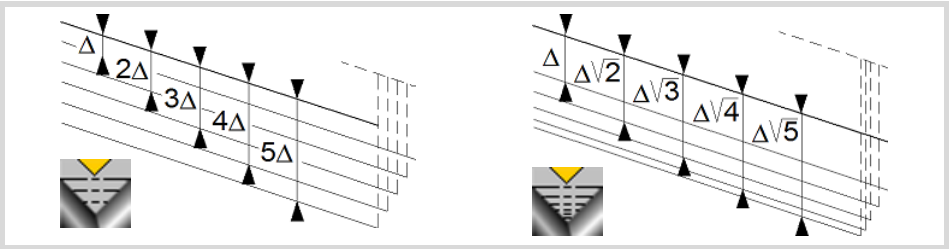
Spindle speed.

S Spindle speed.

Depth of pass.

Δ Maximum penetration step.

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



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 σ=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 σ=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).

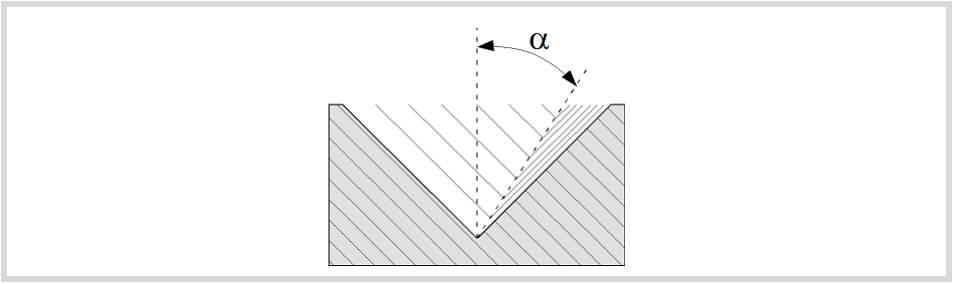
Penetration in zigzag, alternates between radial and final flank. The CNC will ask for the cutter's penetration angle (α).

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.

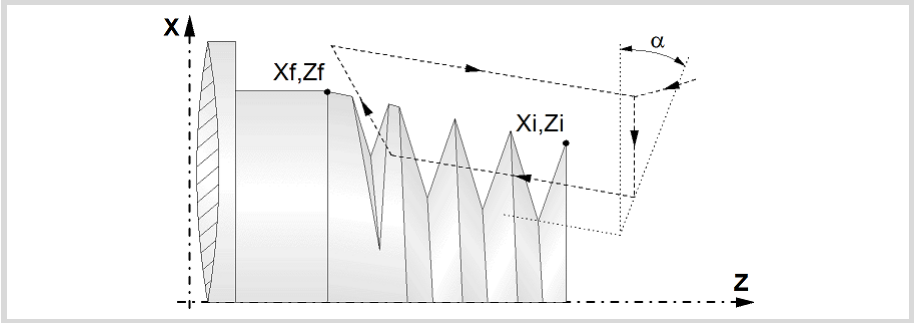
Deactivating the execution of M functions before machining.

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

4.2.1 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 is 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:



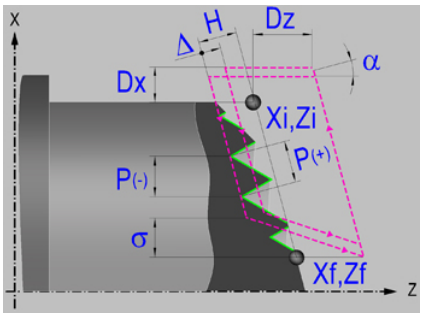
First, rapid move to the corresponding programmed depth. This positioning is done according to the selected tool penetration angle (α). Then, threading of the programmed section up to the distance from the end of the thread (normal thread) or up to the final point (blind thread) and exit threading up to the final coordinate. And last, withdrawal in rapid to the approach point.

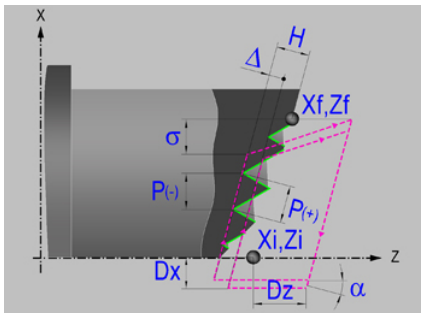
The electronic threading is carried out at 100% of the calculated feedrate and these values cannot be modified from the CNC's operator panel or via PLC. If the OEM allows it, (parameter THREADOVR), the user can modify the speed override from the operator panel and, in that case, the CNC will adapt the 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).

4.3 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.

Xi, Zi Coordinates of the starting point

Xf, Zf Coordinates of the end point. By default, Zf assumes the value set for Zi.

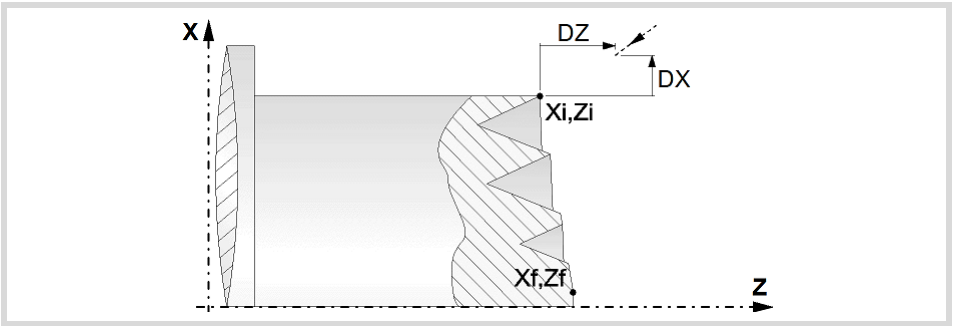
The X axis coordinates must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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 (Xi, Zi).

Dx, Dz 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.

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.

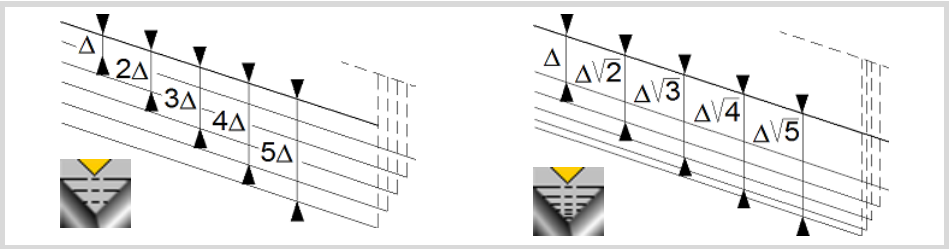
Spindle speed.

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 σ=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 σ=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).

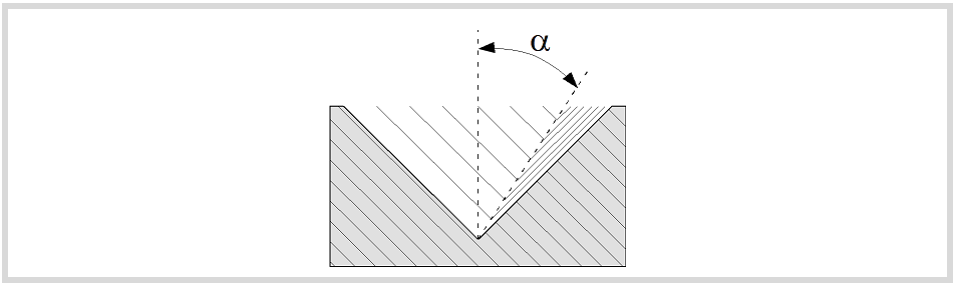
Penetration by final flank. The CNC will ask for the cutter's penetration angle (α).

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.

4.3.1 Basic operation.

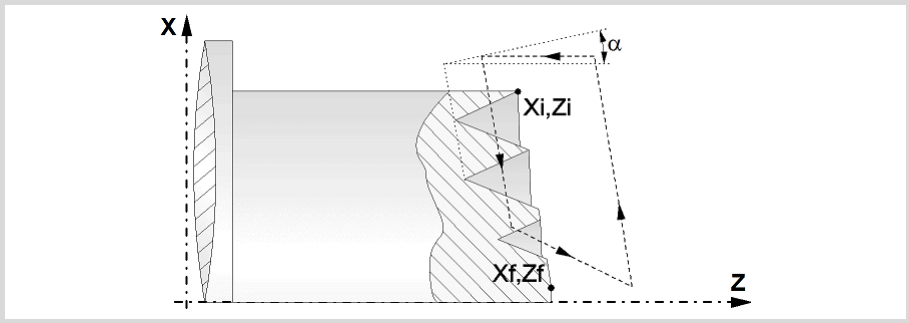
The machining steps in this cycle are as follows:

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 is 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.

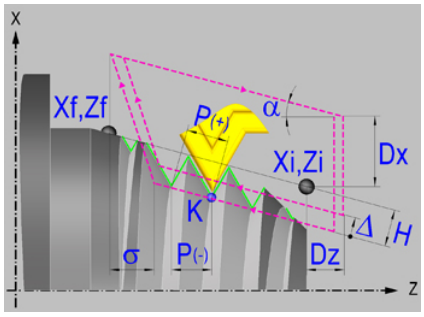
Neither the feedrate override nor the spindle speed override can be changed in the last pass; this pass will be run with the override set in the previous pass.

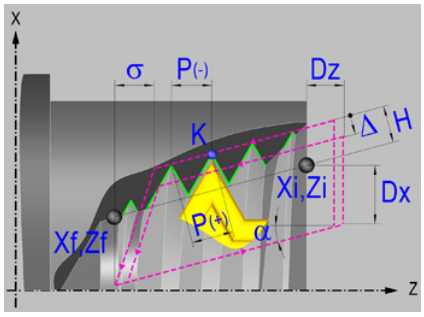
5 Once the operation or cycle has ended, the tool will return to the safety point. When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.

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).

4.4 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 (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.

Xi, Zi	Coordinates of the starting point
Xf, Zf	Coordinates of the end point. By default, Xf assumes the value set for Xi.
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.

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.

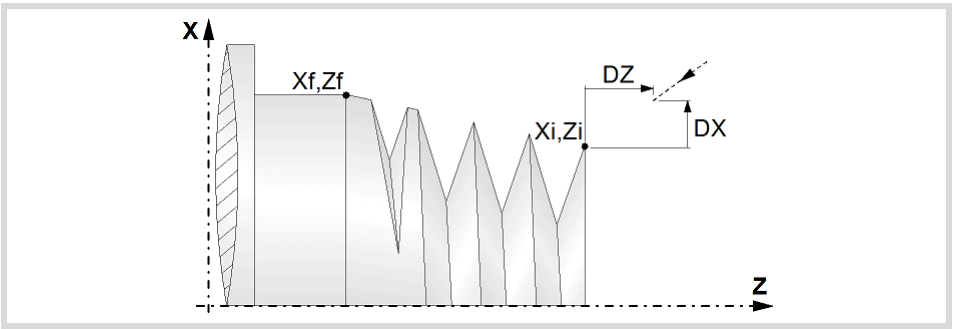
In partial thread repair, the starting point and the final (end) point correspond to the segment to be machined. The entry to the thread (start) is calculated automatically.

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 (Xi, Zi).

Dx, Dz 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.

Type of thread; free-pitch thread o standard thread (the standard threads are not available in face thread repair).

The cycle lets select the following types of standard threads. It is possible to select a taper thread and also a standard thread, in this case, it will calculate the pitch and depth that would correspond to the standard cylindrical thread and it will be assigned to that taper thread.

P-H.	Free-pitch thread.
M (S.I.)	Metric thread with normal pitch (international system).
M (S.I.F.)	Metric thread with fine pitch (international system).
B.S.W. (W)	Whitworth thread with normal pitch.
B.S.F.	Whitworth thread with fine pitch.
U.N.C.	Unified American thread with normal pitch.
U.N.F.	Unified American thread with fine pitch.
B.S.P.	Whitworth gas thread.
API	API thread (oil pipe).

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

Geometry of the thread (free-pitch threads).

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

P — Thread pitch in mm, inches or number of threads per inch. The thread pitch may be set along the taper of the thread or along its associated axis.

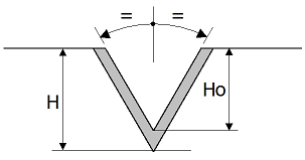
- P > 0; When programming the pitch along the taper of the thread.
- P < 0; To program the pass along according to the associated axis.

H — Total thread depth. The total depth must be programmed in radii and with positive value.

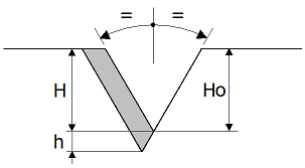
h — Penetration step in thread repair maintaining the geometry. This parameter must be programmed in radius and with a positive value. This parameter is useful for asymmetrical cutters. The penetrating direction is defined by the cutting angle.

- When programming parameter "h", the tool data (cutter angle and the cutting angle) must be correctly defined in the tool table.

Starting with a thread whose depth is Ho, the following examples show the difference between increasing the value of parameter H and programming parameter h.

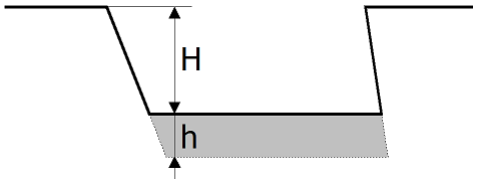


When programming a depth (H) greater than the initial one (Ho), the cycle increases the depth of the thread ignoring the existing flanks.

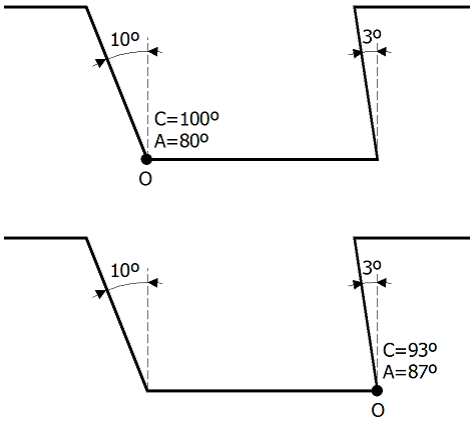


When programming the original depth (H=Ho) plus and increment (h), the cycle increases the depth but maintaining the entry (start) flank.

- 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.



O = Calibrated tip. C = Cutter angle. A = Cutting angle.



In either case, the entry angle (α) must be between -3º and -10º (angle of the edges).

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.

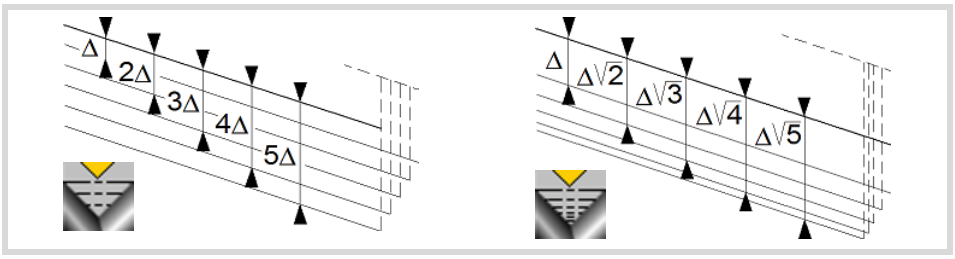
Spindle speed.

S Spindle speed.

Depth of pass.

Δ Maximum penetration step.

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



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, Δ√2, Δ√3, Δ√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 σ=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 σ=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).

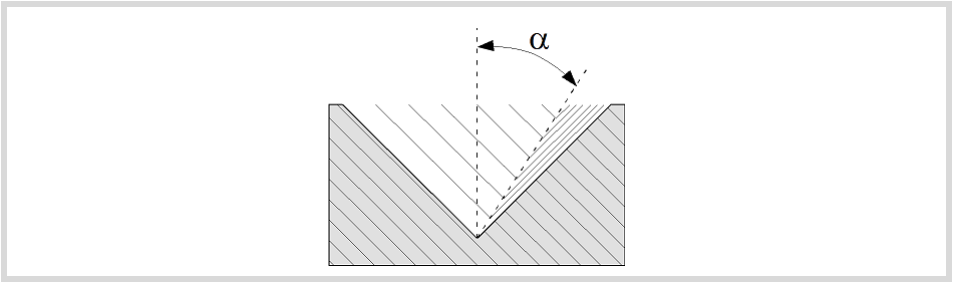
Penetration by final flank. The CNC will ask for the cutter's penetration angle (α).

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.

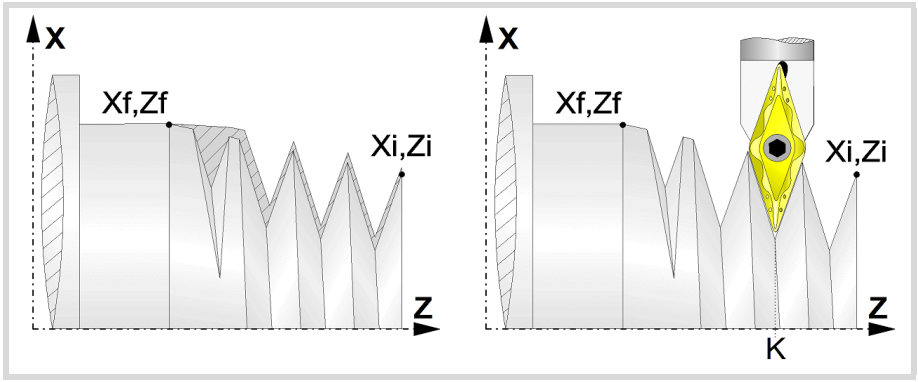
Deactivating the execution of M functions before machining.

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

4.4.1 Basic operation.

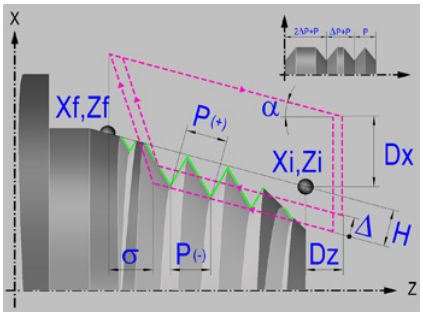
The machining steps in this cycle are as follows:

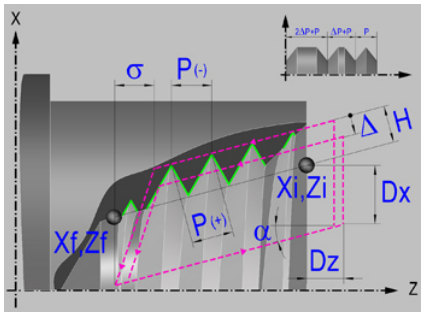
1. Define the dimensions of the thread like at the rest of the levels and the coordinates of one of the roots. To define the coordinates of the root, the CNC must know the spindle position; for that, just orient the spindle (M19) once after power-up.
2. With the spindle stopped, take the tool to be used 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.



4.5 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.

Xi, Zi	Coordinates of the starting point
Xf, Zf	Coordinates of the end point. By default, Xf assumes the value set for Xi.
α	Thread angle.
Δz	Thread length.

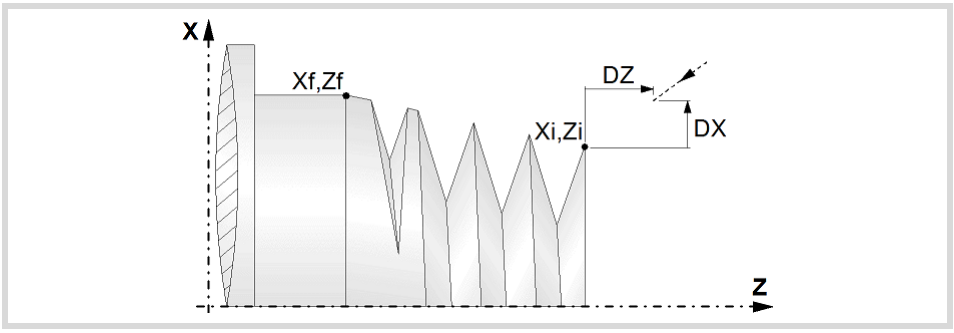
The X axis coordinates must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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 (Xi, Zi).

Dx, Dz	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.

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.

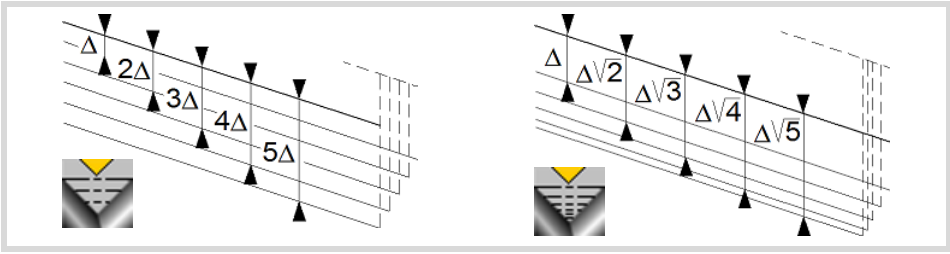
Spindle speed.

S — Spindle speed.

Depth of pass.

Δ — Maximum penetration step.

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 σ=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 σ=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).

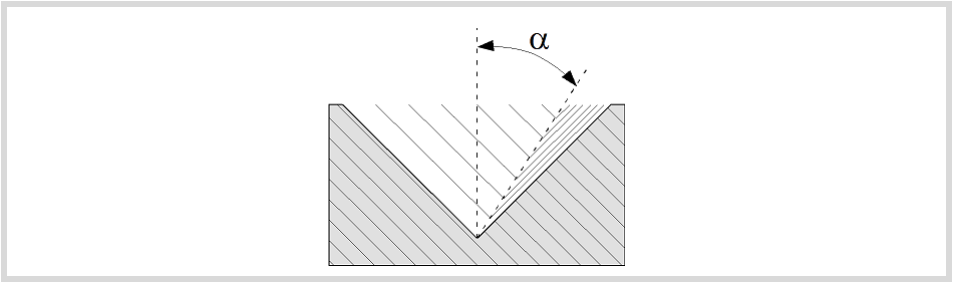
Penetration in zigzag, alternates between radial and final flank. The CNC will ask for the cutter's penetration angle (α).

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.

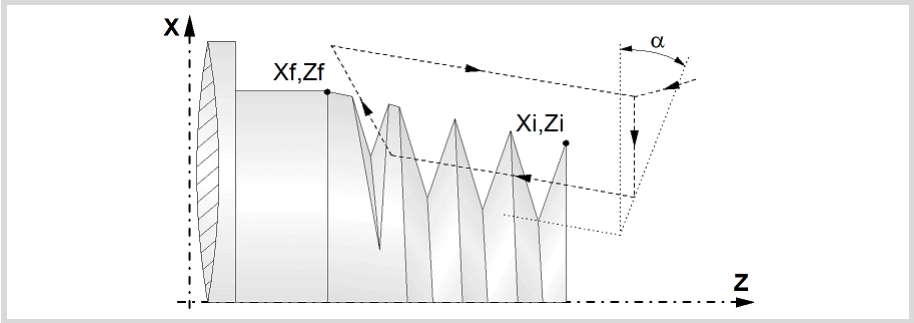
Deactivating the execution of M functions before machining.

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

4.5.1 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 is 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.

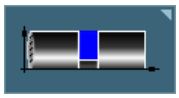
Neither the feedrate override nor the spindle speed override can be changed in the last pass; this pass will be run with the override set in the previous pass.

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).

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.



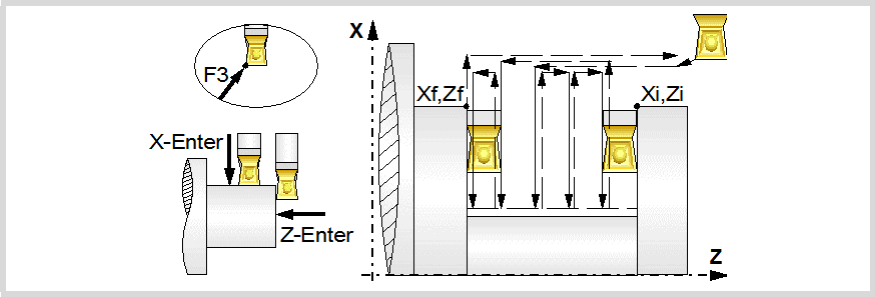
- Simple longitudinal grooving.
- Simple face grooving.
- Inclined (angled) longitudinal grooving.
- Inclined (angled) face grooving.
- Cut-off.

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

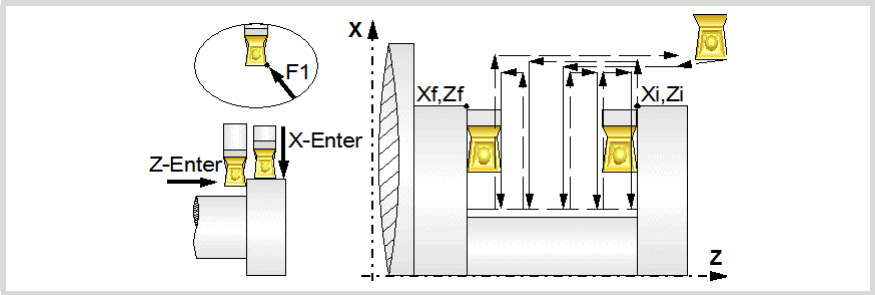
5.1.1 Outside longitudinal grooving.

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:

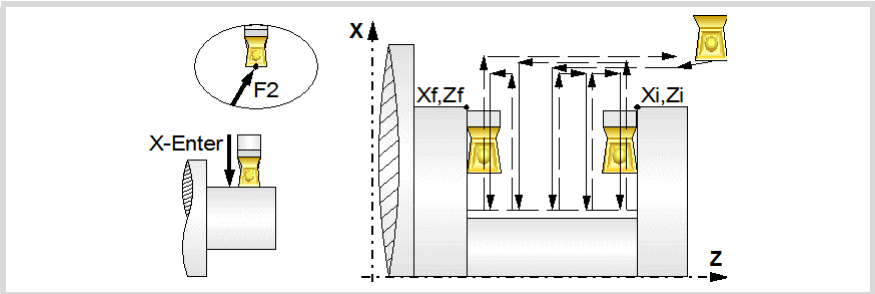
- The lower left-hand corner of the cutter is calibrated. Location code F3.



- The lower right-hand corner of the cutter is calibrated. Location code F1.

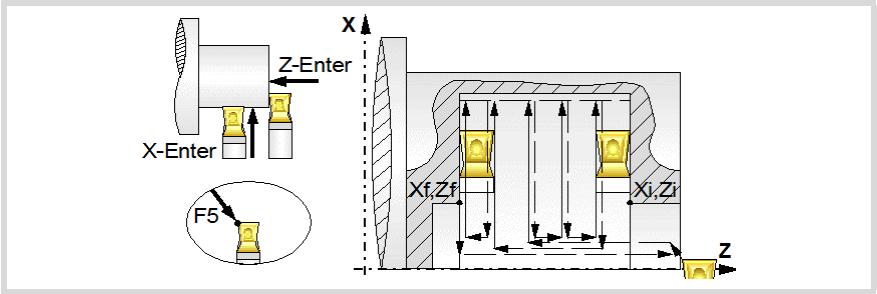


- Calibration is made only according to axis X, the CNC assumes the lower center of the cutter as the calibrated point. Location code F2.

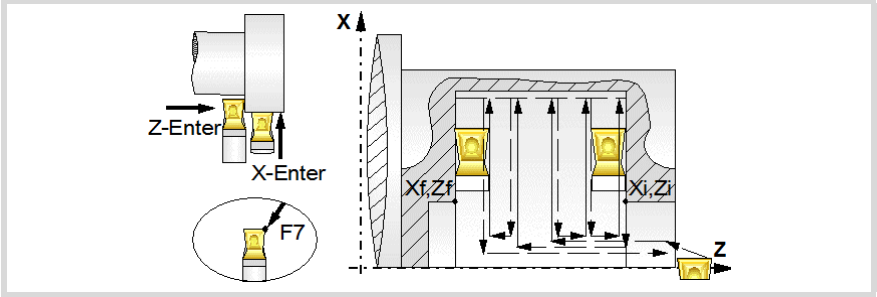


5.1.2 Inside longitudinal grooving.

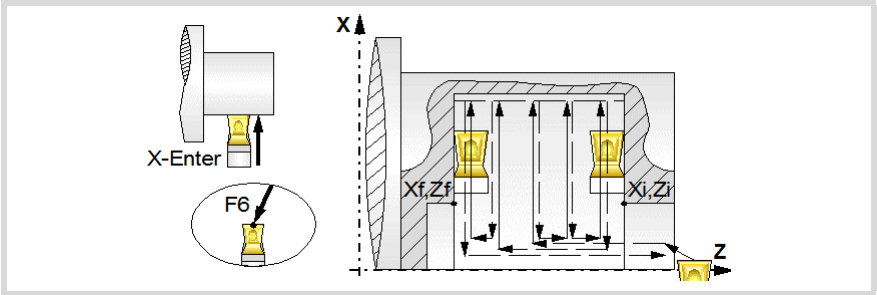
- The upper left-hand corner of the cutter is calibrated. Location code F5.



- The upper right-hand corner of the cutter is calibrated. Location code F7.



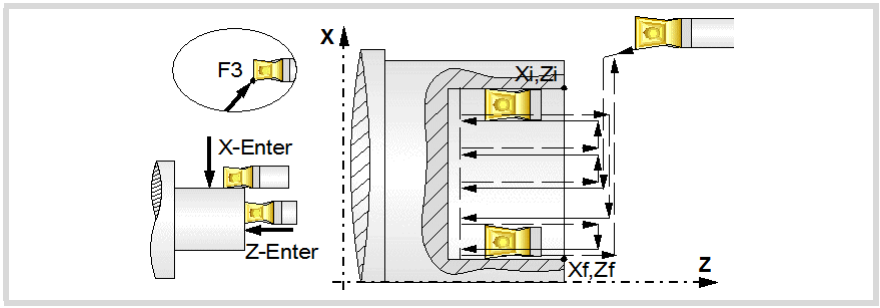
- Calibration is made only according to axis X, the CNC assumes the upper center of the cutter as the calibrated point. Location code F6.



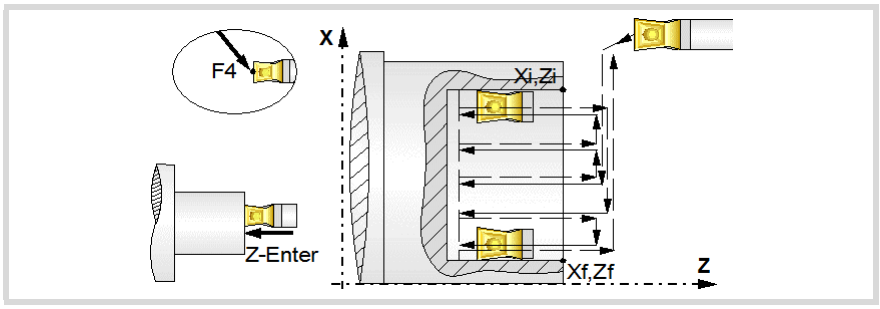
5.1.3 Face grooving.

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:

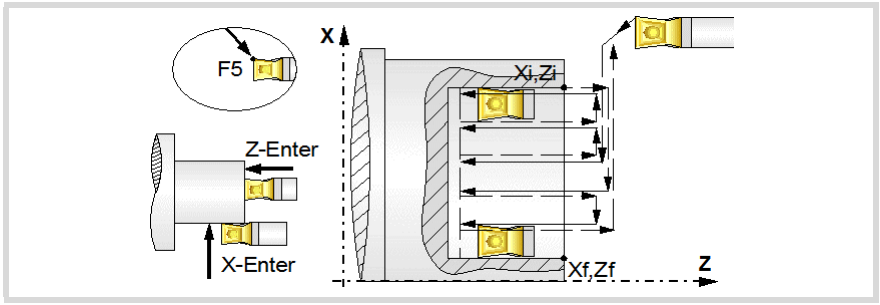
- The lower left-hand corner of the cutter is calibrated. Location code F3.



- 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.

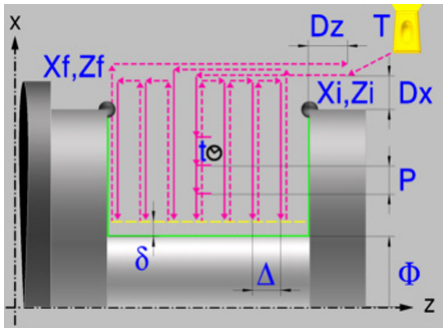


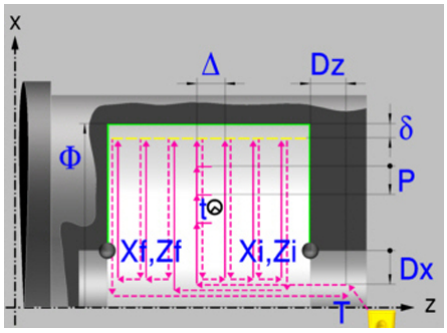
- The upper left-hand corner of the cutter is calibrated. Location code F5.



5.2 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 196.

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.

Xi, Zi Coordinates of the starting point

Xf, Zf Coordinates of the end point. By default, Xf assumes the value set for Xi.

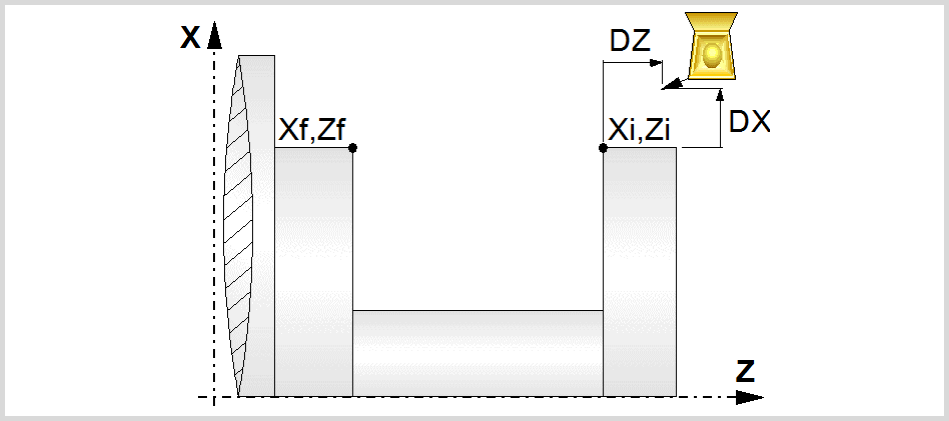
The X axis coordinates must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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 (Xi, Zi).

Dx, Dz Safety distance.

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



Final diameter.

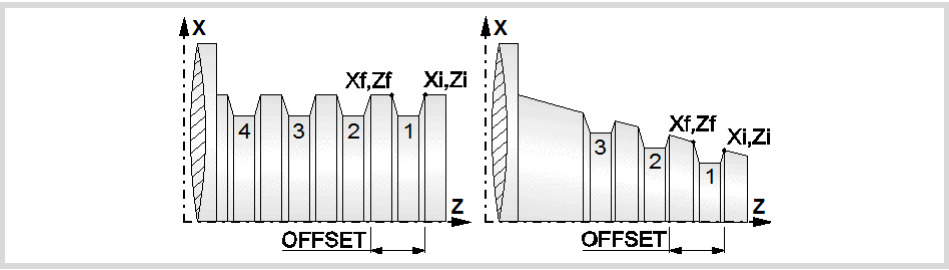
Φ Final diameter.

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 (Xi different from Xf) that taper is maintained for the rest of the grooves.



Spindle parameters.

Spindle turning direction

The cycle lets select the spindle turning direction.

Spindle clockwise.

Spindle counterclockwise.

Coolant status.

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

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

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

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.

Penetration (icon for inner and outer machining).

Penetration without chip removal. Option recommended for materials with small chips.

Penetration with chip removal. Option recommended for materials with large chips.

Penetration with chip removal and first penetration in zig-zag. Option recommended for materials with large chips.

Two more parameters must be defined when the cycle runs the grooving operation with chip removal.

P Penetration step.

t Dwell for chip removal.

Chip removal (icon).

Waiting time for chip removal only in the first penetration.

Waiting time for chip removal in all penetrations.

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 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.

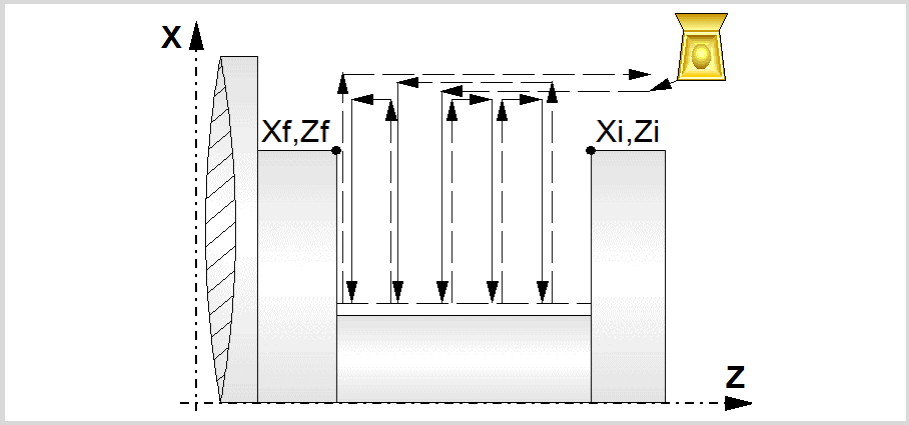
2 The spindle starts with the speed selected and in the direction stated.

3 The tool approaches the starting point (Xi, Zi) in rapid, keeping the selected safety distance (Dx, Dz) 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.

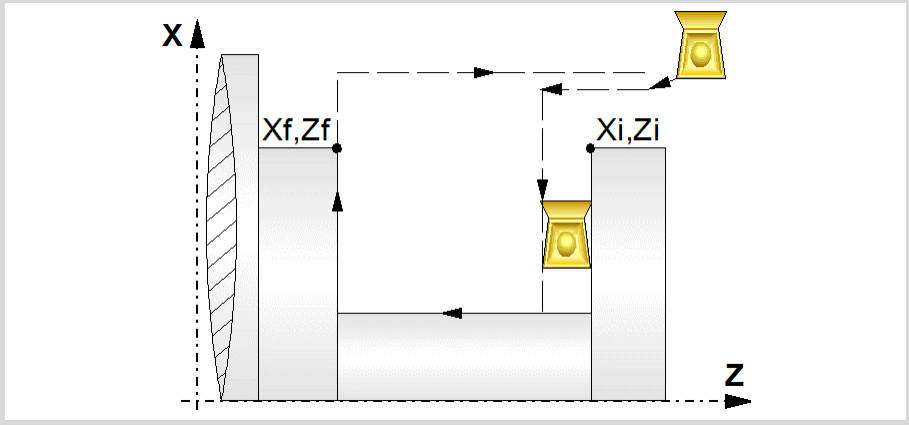
The first penetration pass may be done in two ways; either continuously or in intervals depending on the programmed pass and with a dwell at the bottom of each pass.



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.

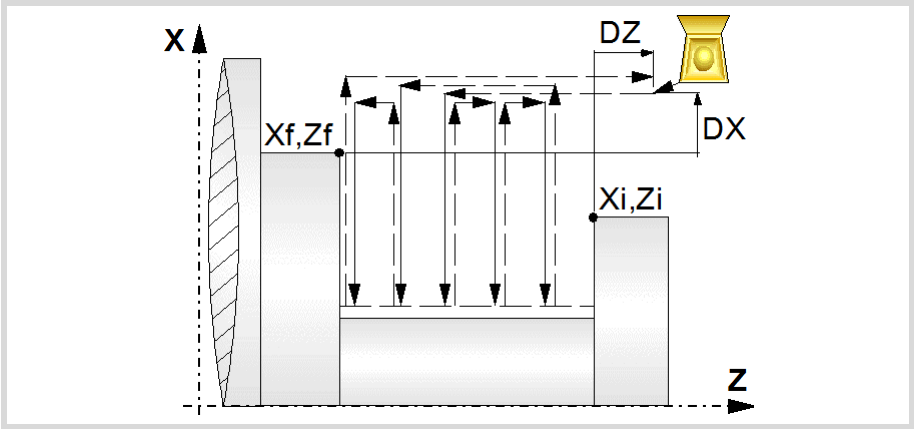


6 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.

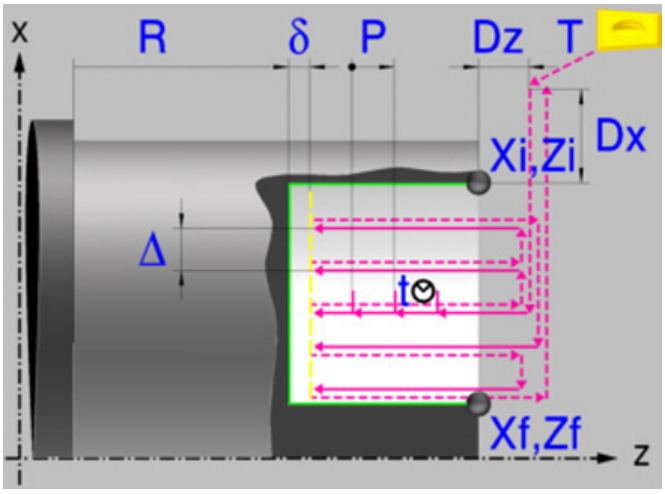
7 The CNC will not stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

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.



5.3 Simple face grooving cycle



Calibration of the grooving tool.

When calibrating the grooving tool proper indication should be made of the location codes for the corner that has been calibrated. For this cycle, the same tool may be calibrated in three different ways, both for outside machining and for inside machining See *"5.1 Calibrate the grooving tool (location code, shape)."* on page 196.

Geometric parameters.

Coordinates of the profile's starting and final points.

Xi, Zi Coordinates of the starting point

Xf, Zf Coordinates of the end point. By default, Zf assumes the value set for Zi.

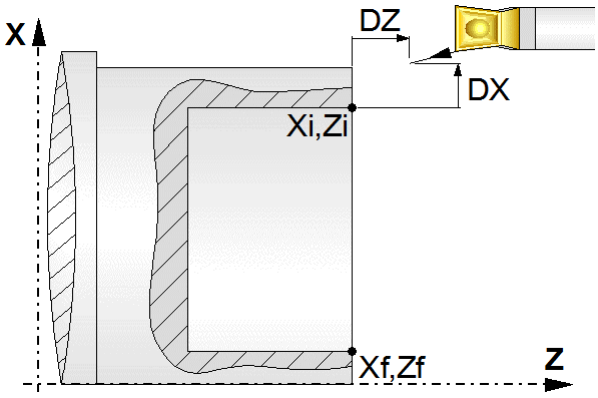
The X axis coordinates must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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 (Xi, Zi).

Dx, Dz Safety distance.

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



Coordinate of the bottom of the groove.

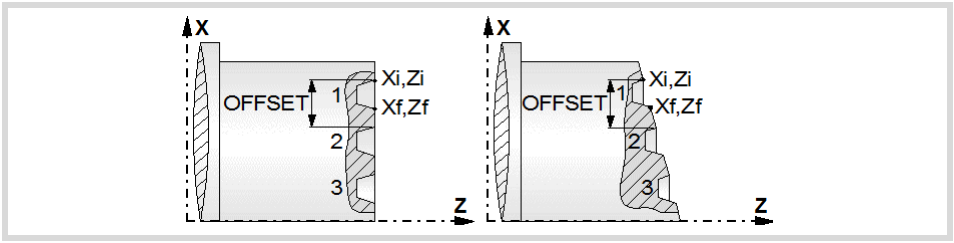
R Coordinate of the bottom of the groove

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 (Xi different from Xf) that taper is maintained for the rest of the grooves.



Spindle parameters.

Spindle turning direction

The cycle lets select the spindle turning direction.

Spindle clockwise.

Spindle counterclockwise.

Coolant status.

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

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

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

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 (icon).

Penetration without chip removal. Option recommended for materials with small chips.

Penetration with chip removal. Option recommended for materials with large chips.

Penetration with chip removal and first penetration in zig-zag. Option recommended for materials with large chips.

Two more parameters must be defined when the cycle runs the grooving operation with chip removal.

P Penetration step.

t Dwell for chip removal.

Chip removal (icon).

Waiting time for chip removal only in the first penetration.

Waiting time for chip removal in all penetrations.

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 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.

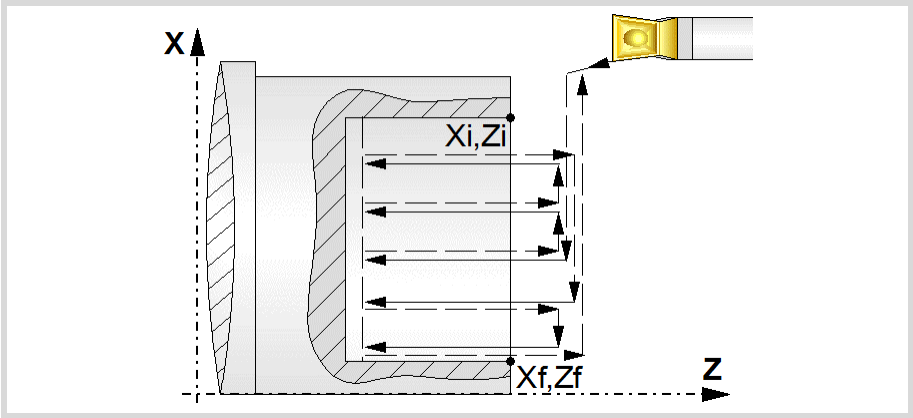
2 The spindle starts with the speed selected and in the direction stated.

3 The tool approaches the starting point (Xi, Zi) in rapid, keeping the selected safety distance (Dx, Dz) 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.

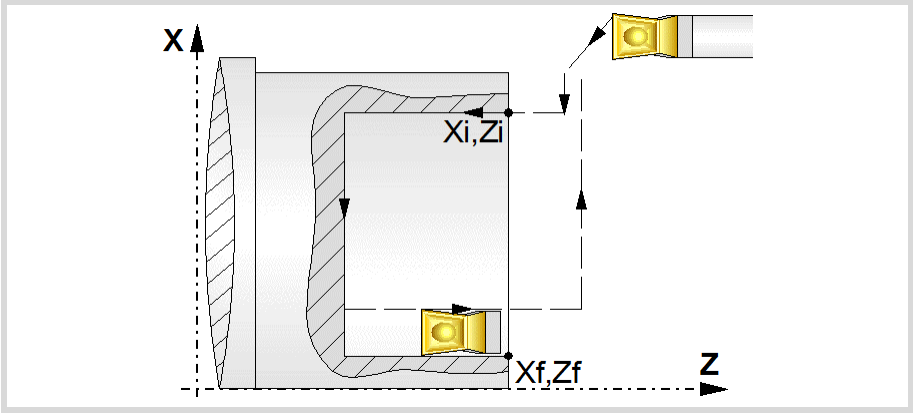
The first penetration pass may be done in two ways; either continuously or in intervals depending on the programmed pass and with a dwell at the bottom of each pass.



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.

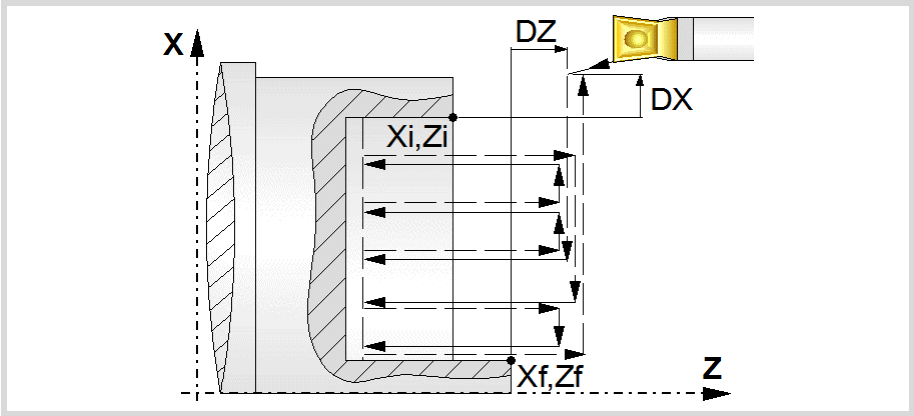


6 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.

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).

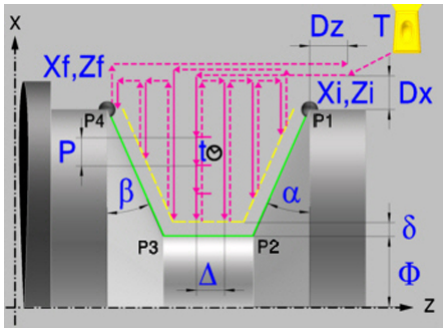
Considerations.

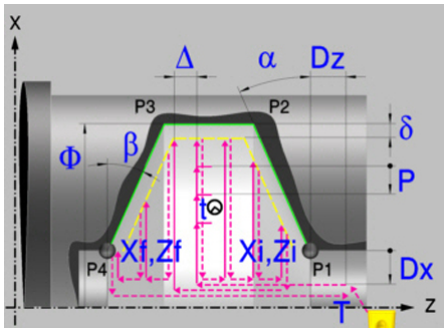
- If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.
- If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the finishing operation, the tool will move to the safety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



5.4 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 196.

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.

Xi, Zi Coordinates of the starting point

Xf, Zf Coordinates of the end point. By default, Xf assumes the value set for Xi.

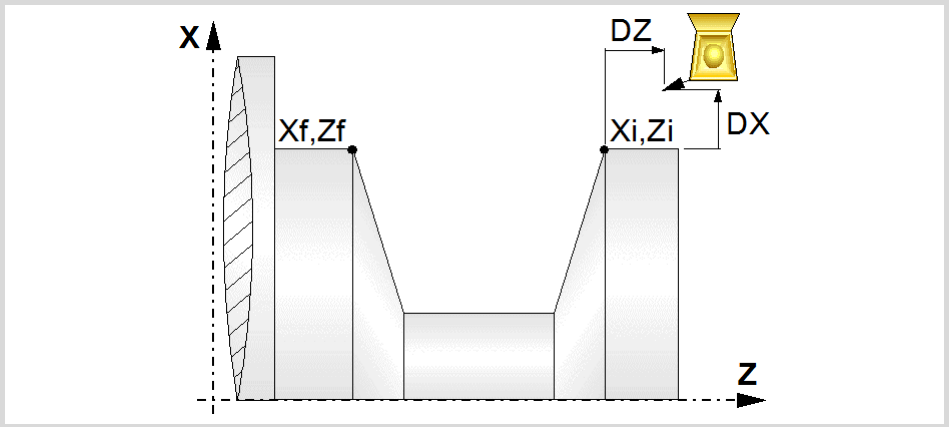
The X axis coordinates must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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 (Xi, Zi).

Dx, Dz Safety distance.

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.

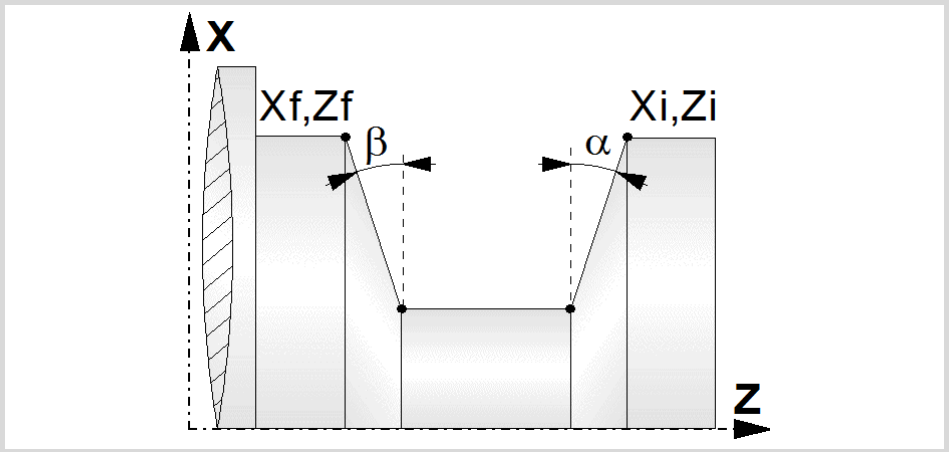
r Rounding radius of the pocket corners.

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.

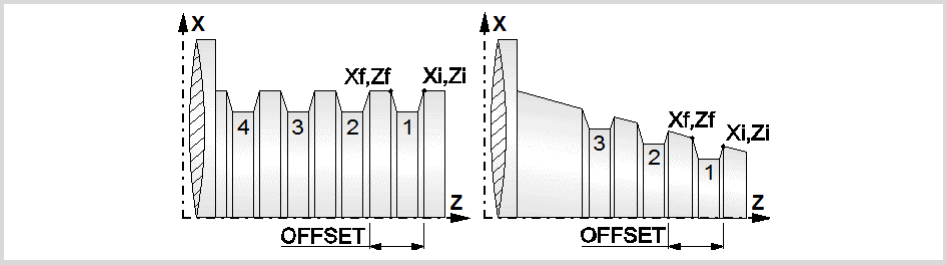


Groove repetition.

N Number of grooves. If is equal to "0" or "1", the cycle will only run one grooving operation.

I Distance between grooves.

With this data, the same groove may be repeated along the Z axis on longitudinal grooving or along the X axis on face grooving. If the initial groove is tapered (Xi different from Xf) that taper is maintained for the rest of the grooves.



Spindle parameters.

Spindle turning direction

The cycle lets select the spindle turning direction.

Spindle clockwise.

Spindle counterclockwise.

Coolant status.

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

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

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

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.

Penetration (icon for inner and outer machining).

Penetration without chip removal. Option recommended for materials with small chips.

Penetration with chip removal. Option recommended for materials with large chips.

Penetration with chip removal and first penetration in zig-zag. Option recommended for materials with large chips.

Two more parameters must be defined when the cycle runs the grooving operation with chip removal.

P Penetration step.

t Dwell for chip removal.

Chip removal (icon).

Waiting time for chip removal only in the first penetration.

Waiting time for chip removal in all penetrations.

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).

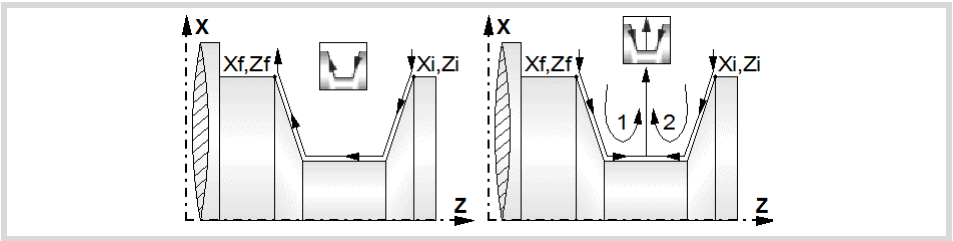
Finishing stock.

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.

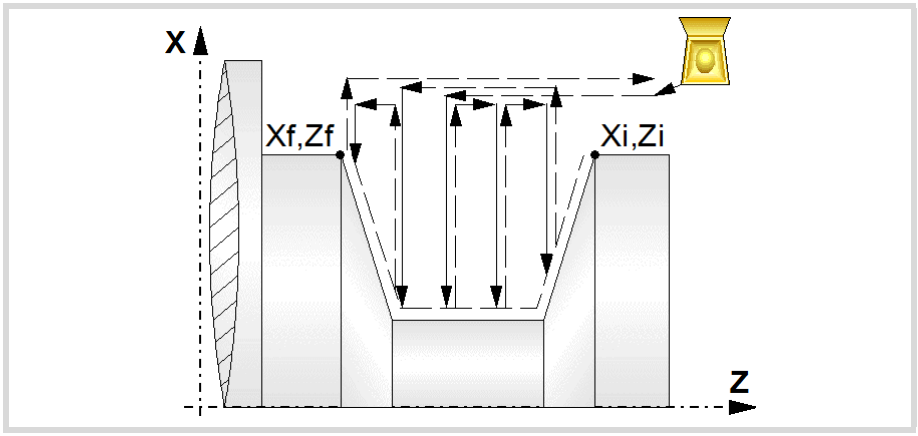
5.4.1 Basic operation

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 (Xi, Zi) in rapid, keeping the selected safety distance (Dx, Dz) 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.

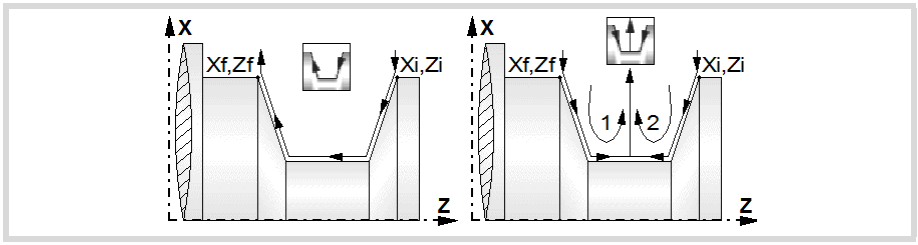
   The first penetration pass may be done in two ways; either continuously or in intervals depending on the programmed pass and with a dwell at the bottom of each pass.



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.

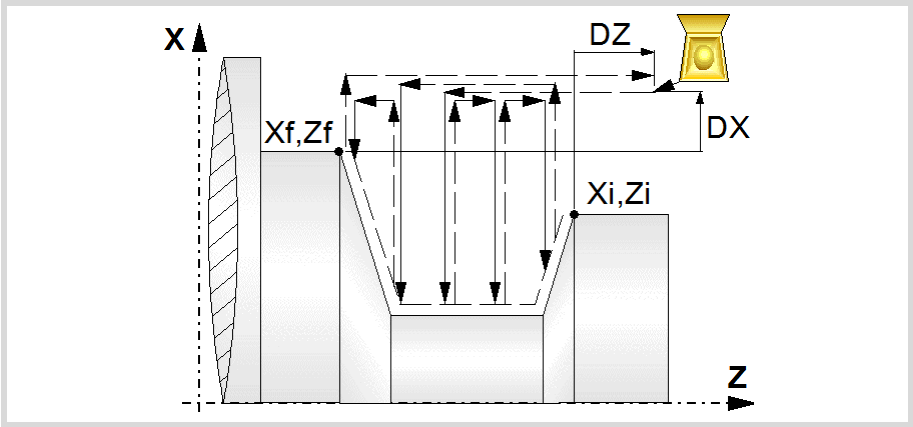
   The cycle runs a single pass to remove the finishing stock. This operation is done with the conditions set for the finishing operation.



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).

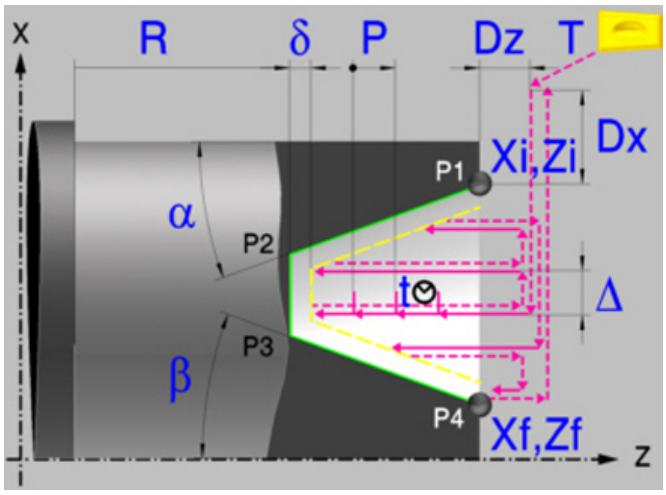
Considerations.

- 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.

5.5 Inclined face grooving cycle.



Calibration of the grooving tool.

When calibrating the grooving tool proper indication should be made of the location codes for the corner that has been calibrated. For this cycle, the same tool may be calibrated in three different ways, both for outside machining and for inside machining See *"5.1 Calibrate the grooving tool (location code, shape)."* on page 196.

Geometric parameters.

Coordinates of the profile's starting and final points.

Xi, Zi Coordinates of the starting point

Xf, Zf Coordinates of the end point. By default, Zf assumes the value set for Zi.

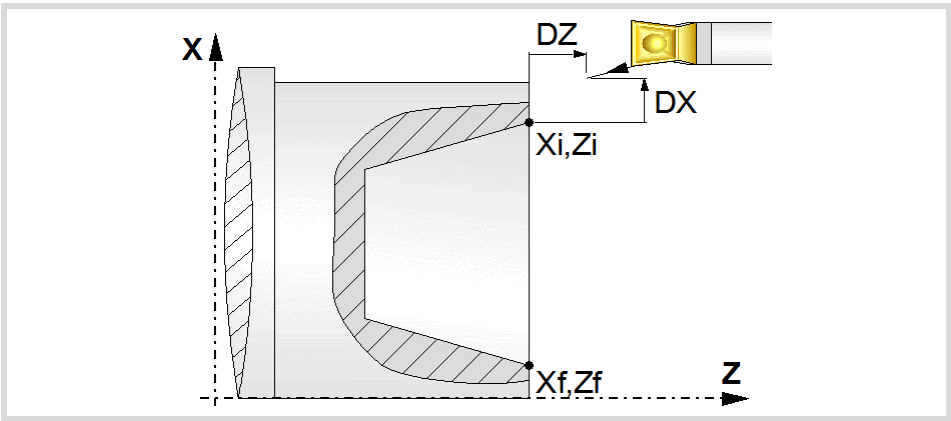
The X axis coordinates must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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 (Xi, Zi).

Dx, Dz Safety distance.

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



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.

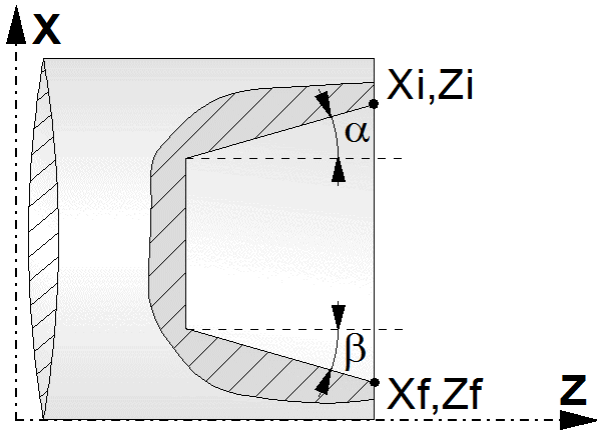
r Rounding radius of the pocket corners.

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.

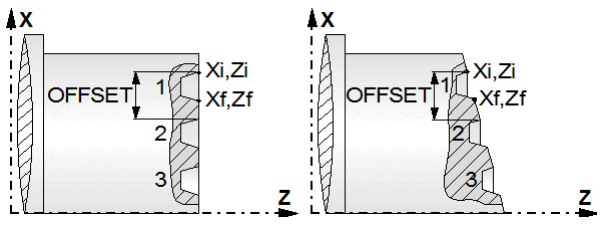


Groove repetition.

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 (Xi different from Xf) that taper is maintained for the rest of the grooves.



Spindle parameters.

Spindle turning direction

The cycle lets select the spindle turning direction.

Spindle clockwise.

Spindle counterclockwise.

Coolant status.

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

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

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

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 (icon).

Penetration without chip removal. Option recommended for materials with small chips.

Penetration with chip removal. Option recommended for materials with large chips.

Penetration with chip removal and first penetration in zig-zag. Option recommended for materials with large chips.

Two more parameters must be defined when the cycle runs the grooving operation with chip removal.

P Penetration step.

t Dwell for chip removal.

Chip removal (icon).

Waiting time for chip removal only in the first penetration.

Waiting time for chip removal in all penetrations.

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).

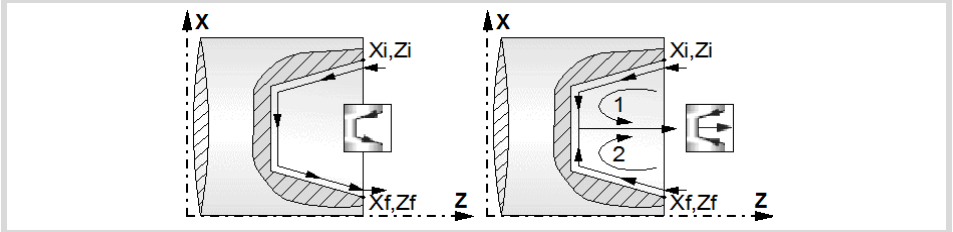
Finishing stock.

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.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

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.

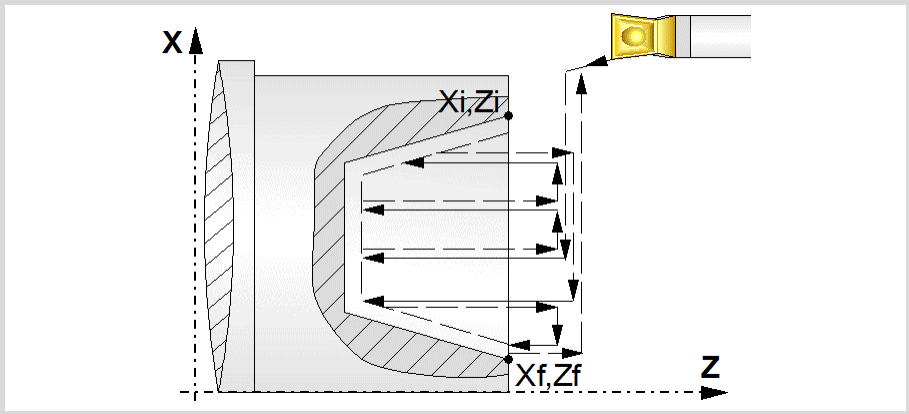
2 The spindle starts with the speed selected and in the direction stated.

3 The tool approaches the starting point (Xi, Zi) in rapid, keeping the selected safety distance (Dx, Dz) 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.

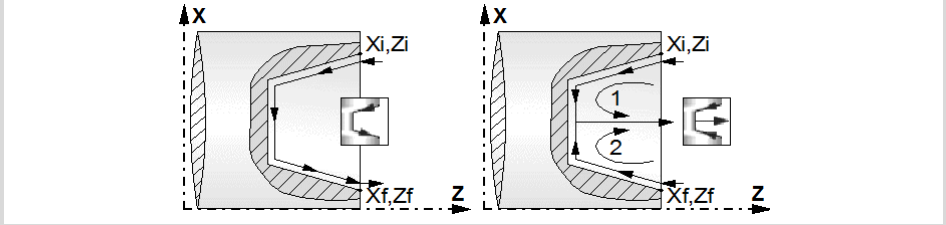
The first penetration pass may be done in two ways; either continuously or in intervals depending on the programmed pass and with a dwell at the bottom of each pass.



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.

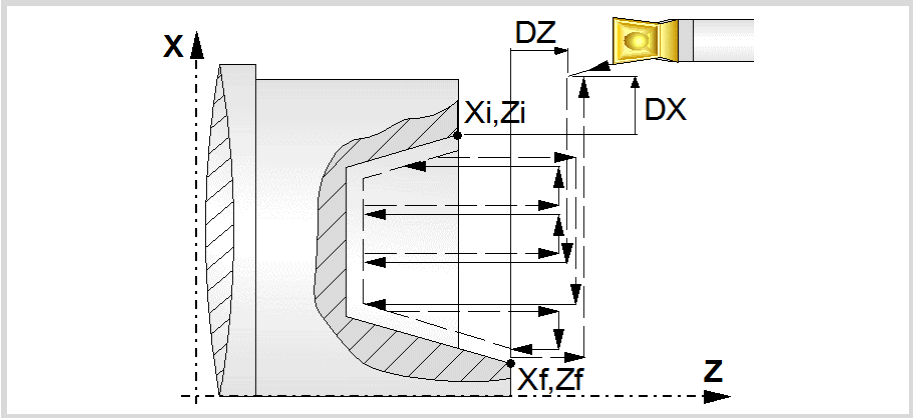


6 Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.

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).

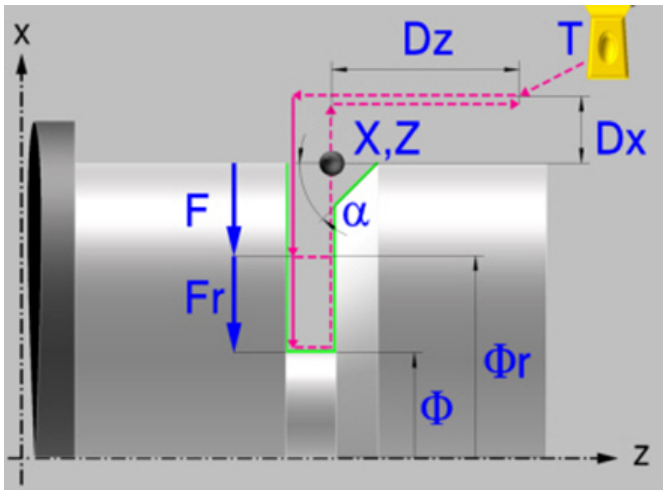
Considerations.

- 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.

5.6 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 196.

Geometric parameters.

Coordinates of the profile's starting and final points.

Xi, Zi Coordinates of the starting point

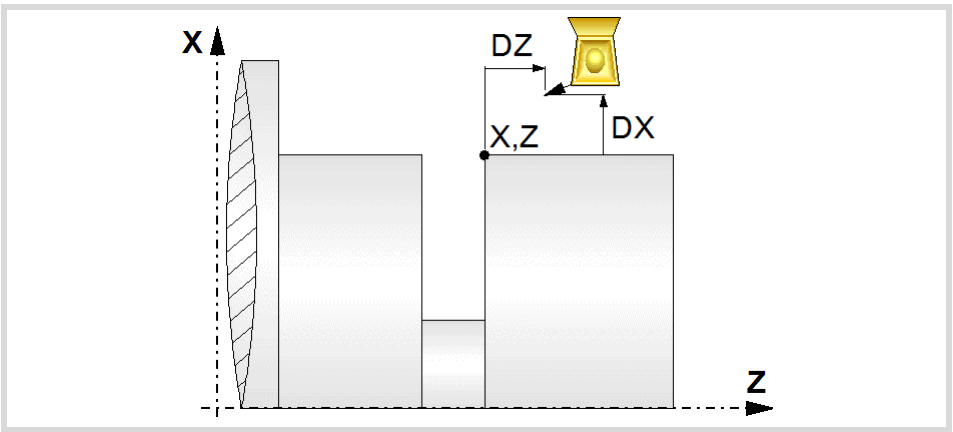
The X axis coordinate must be programmed in the active units, radius or diameter. To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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 (Xi, Zi).

Dx, Dz Safety distance.

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 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.

5.6.1 Basic operation

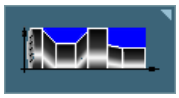
The machining steps in this cycle are as follows:

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 (Xi, Zi) in rapid, keeping the selected safety distance (Dx, Dz) 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 Φr, from then on, it slows down until the final diameter is reached at a feedrate Fr.
6. The part finishing is done with the set condition; axis feedrate (F), slower feedrate (Fr), 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).

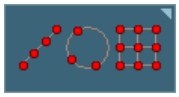
CANNED CYCLES OF THE EDITOR. PROFILES.

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-emptied 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-emptied pocket in the XC/XY plane.
- 2D profile pocket in the XC/XY plane.



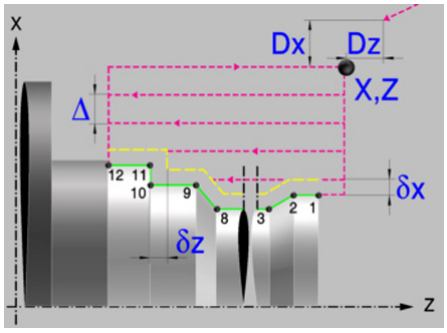
Multiple machining may be added to pocket cycles so the cycle is repeated in several points. The available multiple machining operations are the following. See chapter *"9 Canned cycles of the editor. Multiple positioning."*.

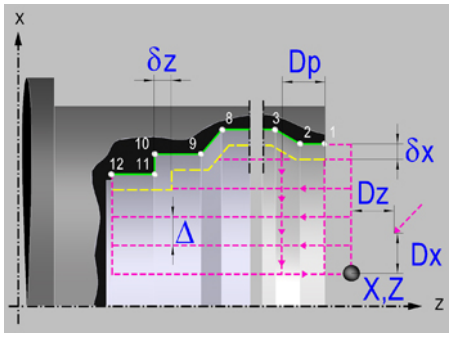
- Points in line.
- Points in arc.
- Points in rectangle.
- Points in grid.
- Random points.

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.

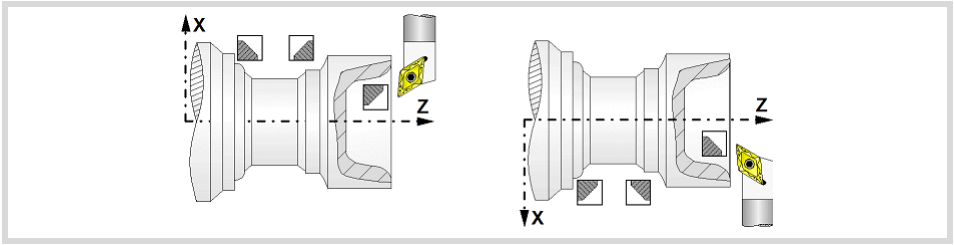
Outside machining.

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.

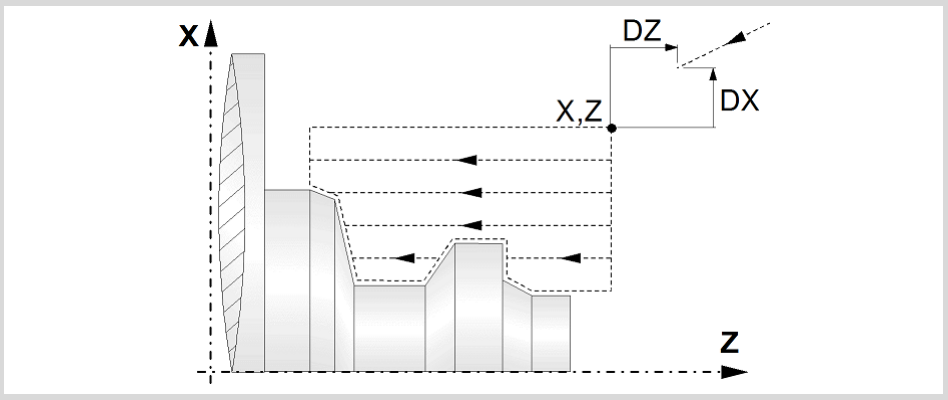
To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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).

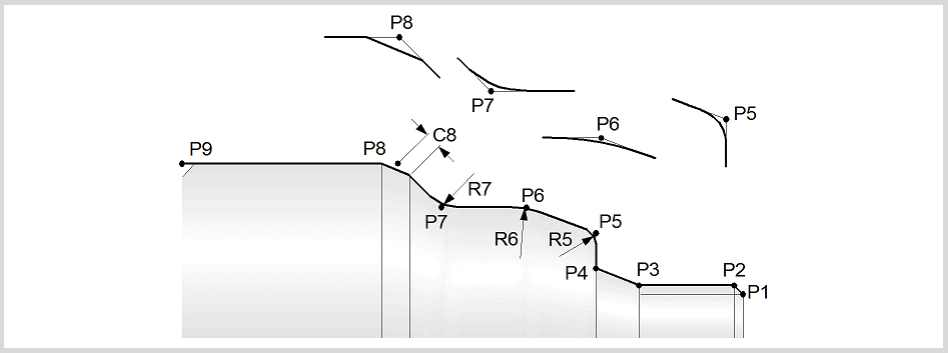
Dx, Dz Safety distance.

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



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.

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.

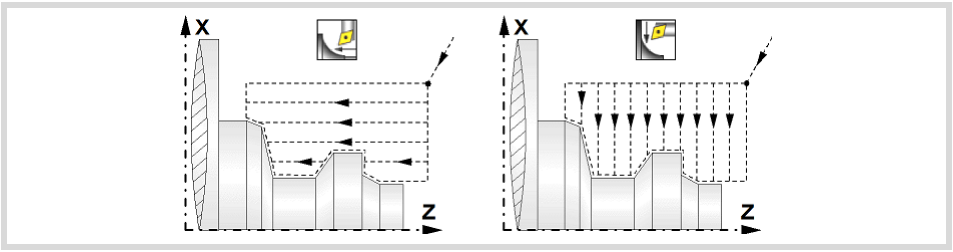
Coolant OFF. The CNC outputs the M9 function to the PLC.

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.

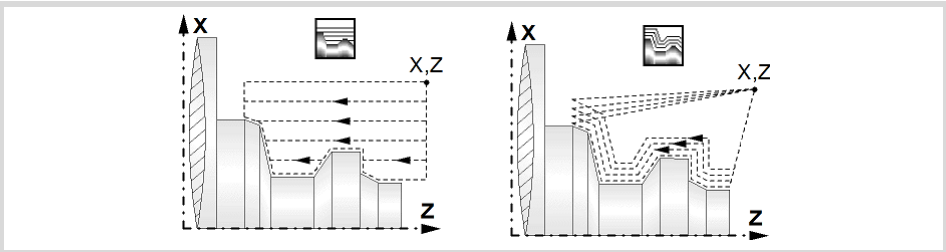
Machining type; paraxial or pattern repeat.

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.

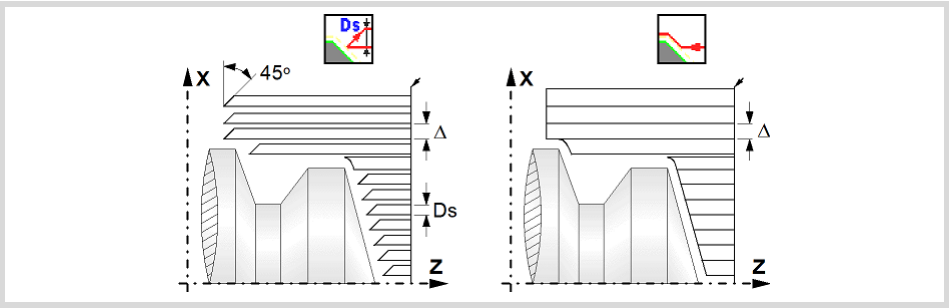
When paraxial roughing, the cycle offers the choice to exit at 45º at the end of each roughing pass; otherwise, each roughing pass ends following the profile.

Tool withdrawal following the profile.

Tool withdrawal with a 45º exit.

Ds 45º 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 45º exits. If not programmed or programmed with a 0 value, the cycle does not run this pass.



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.

6.1.1 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 45º), the cycle will run one last pass parallel to the profile to remove the peaks left by the exits at 45º.
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.

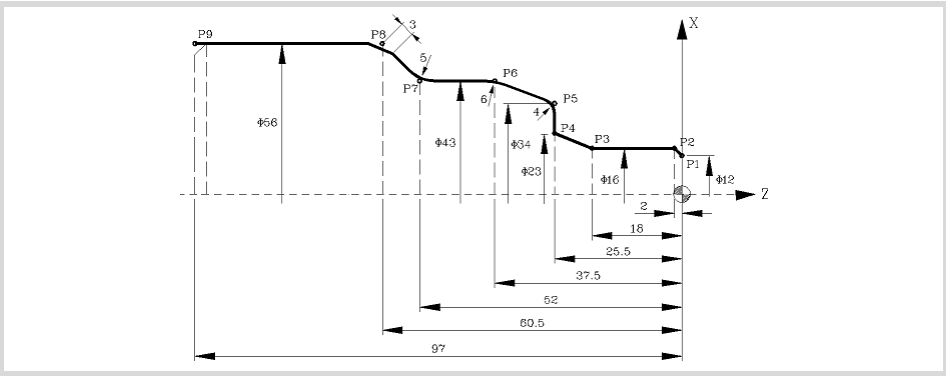
   The cycle runs a single pass until reaching the final profile, eliminating the finishing stocks. This operation is done with the conditions set for the finishing operation.
6. Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When 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).

6.1.2 Programming example.



Geometric parameters.

Outside machining.

Work quadrant.

X 65.0000 Starting point.
Z 10.0000

Dx 1.0000 Safety distance.
Dz 1.0000

Define the part profile.

P1	ABS	X	12.0000	Z	- 0.0000			
P2	ABS	X	16.0000	Z	- 2.0000			
P3	ABS	X	16.0000	Z	- 18.0000			
P4	ABS	X	23.0000	Z	- 25.5000			
P5	ABS	X	34.0000	Z	- 25.5000	R	4.0000	
P6	ABS	X	43.0000	Z	- 37.5000	R	6.0000	
P7	ABS	X	43.0000	Z	- 52.0000	R	5.0000	
P8	ABS	X	56.0000	Z	- 60.5000	C	3.0000	
P9	ABS	X	56.0000	Z	- 97.0000			
P10	ABS	X	56.0000	Z	- 97.0000			

Spindle parameters.

Machining in the turning direction.

Constant turning speed.

Machining parameters. Roughing.

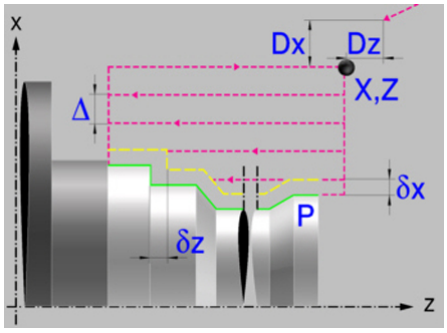
Δ 2 F 0.8 S 1000 T 3

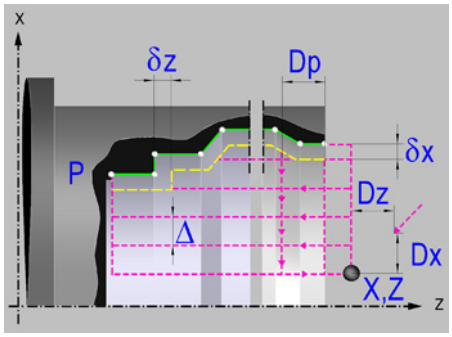
Machining parameters. Finishing.

δ 0.25 F 0.4 S 1000 T 3

6.2 Profile turning cycle.

The cycle allows outside and inside machining.





Geometric parameters.

Inside or outside machining.

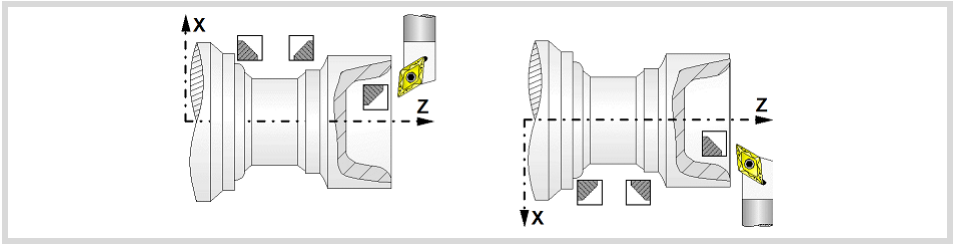
Outside machining.

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.

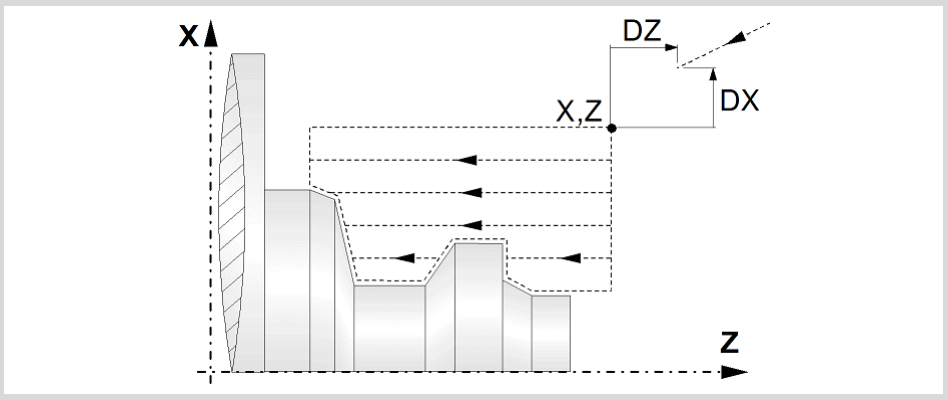
To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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.



Define a profile.

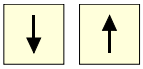
This icon lets define the profile using the profile editor; it may be an existing profile or a new one.

Profile definition.

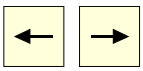
P Name of the program with the profile (between 0 and 999).

- 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.

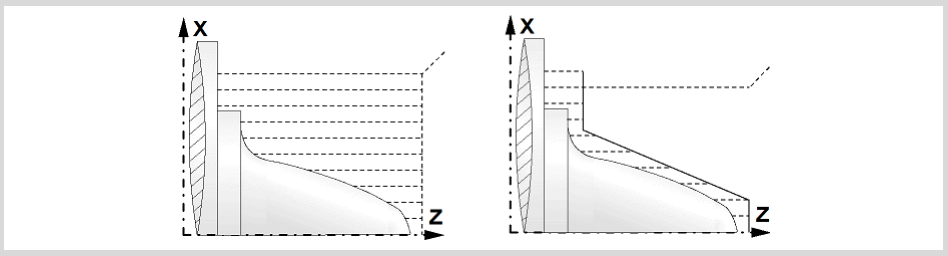
Modify an existing profile.

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.

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.

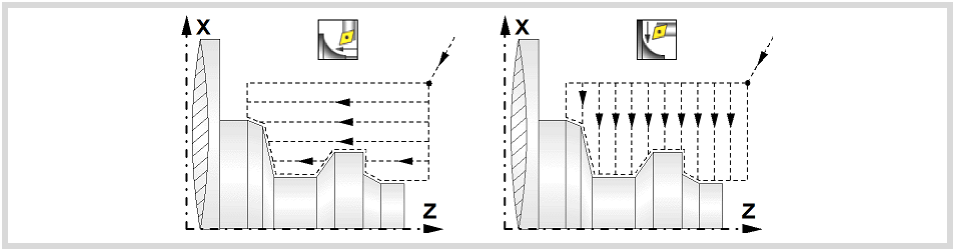
Coolant OFF. The CNC outputs the M9 function to the PLC.

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.

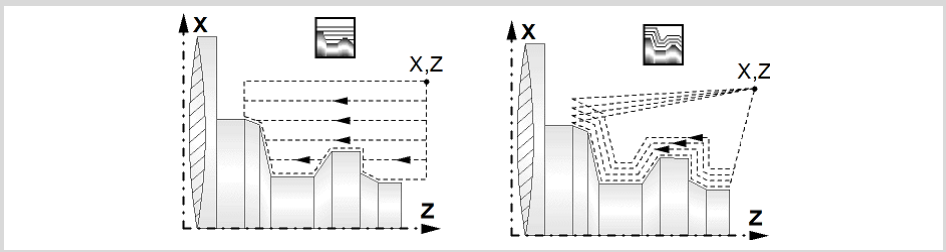
Machining type; paraxial or pattern repeat.

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.

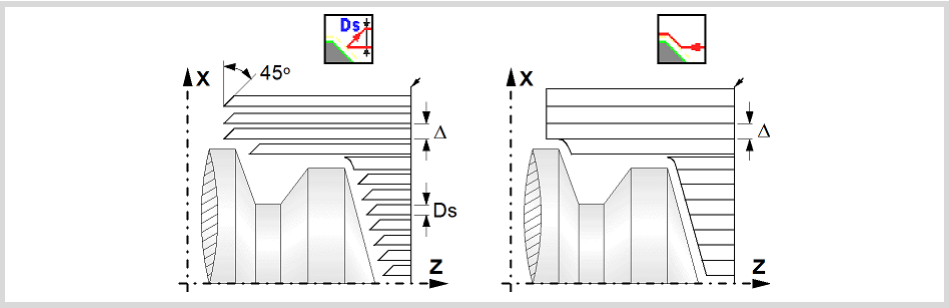
When paraxial roughing, the cycle offers the choice to exit at 45º at the end of each roughing pass; otherwise, each roughing pass ends following the profile.

Tool withdrawal following the profile.

Tool withdrawal with a 45º exit.

Ds — 45º 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 45º exits. If not programmed or programmed with a 0 value, the cycle does not run this pass.



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.

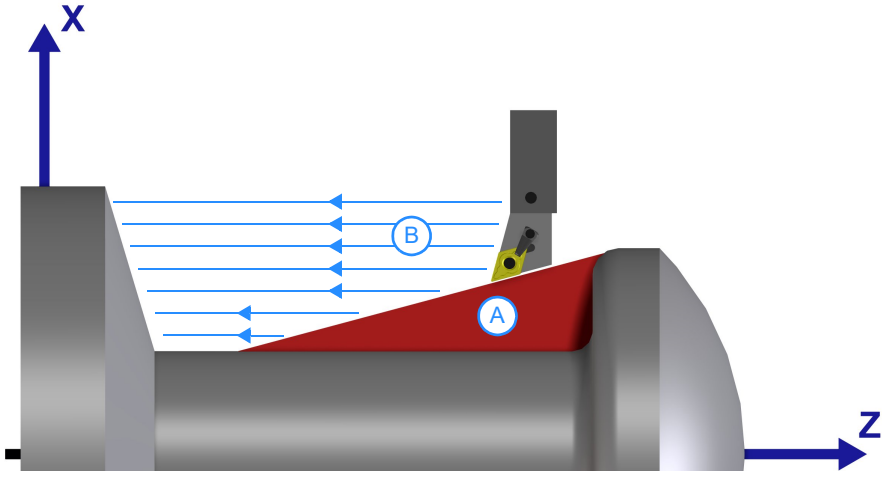
Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

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.

Residual material.

Roughing operation.



A.Machining error.
B.Machining direction.

When the roughing operation cannot remove all the specified material or profile due to the geometry of the profile itself (valleys) or the tool, the cycle saves the information for the non-machined part and finishes the roughing without displaying any warning.

If the finishing operation detects that there is material not removed in the previous operation, it first performs a few passes to remove this material (as much as the geometry of the tool allows) and then performs the finishing pass. This machining process uses its own cutting data. The operator is responsible for ensuring that the tool has the proper geometry to

machine the profile. If the finishing operation cannot remove all the specified material or profile, it machines the part that it can and the cycle displays the corresponding warning.

Activating the operation.

To remove residual material from the roughing operation, the corresponding verification checkbox must be activated.

Remove residual material.

Do not remove residual material.

Tool data.

T Tool. If it is set to 0, the cycle does not remove the residual material.

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.

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 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.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

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 45º), the cycle will run one last pass parallel to the profile to remove the peaks left by the exits at 45º.
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.

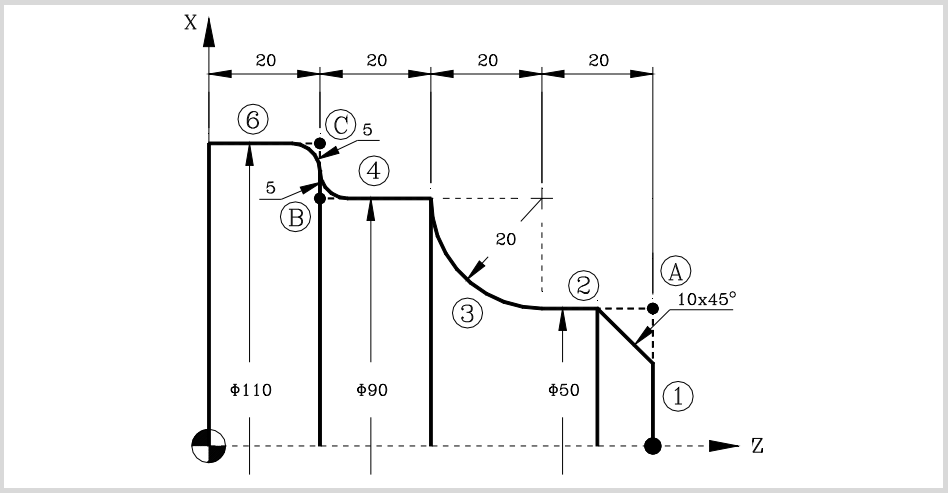
   The cycle runs a single pass until reaching the final profile, eliminating the finishing stocks. This operation is done with the conditions set for the finishing operation.
6. Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When 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).

6.2.2 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.

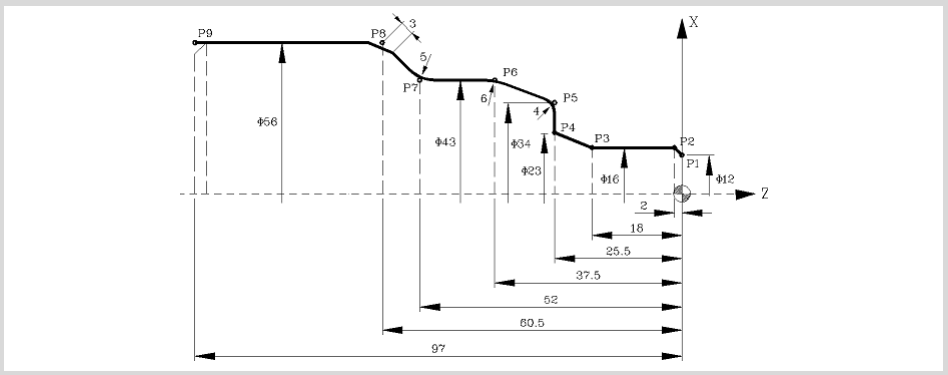
Constant turning speed.

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



Geometric parameters.

	Outside machining.		Work quadrant.
X 65.0000 Z 10.0000	Starting point.	Dx 1.0000 Dz 1.0000	Safety distance.

Define the part profile.

Definition of a profile without rounding or chamfers.

Section.		Geometry.	
	Starting point.	Z = 0	X = 0
(1)	Straight line.	Z = 0	X = 16
(2)	Straight line.	Z = -18	X = 16
(3)	Straight line.	Z = -25.53	X = 23
(4)	Straight line.	Z = -25.5	X = 34
(5)	Straight line.	Z = -37.5	X = 43
(6)	Straight line.	Z = -52	X = 43
(7)	Straight line.	Z = -60.5	X = 56
(8)	Straight line.	Z = -97	X = 56

Definition of roundings and chamfers.

Corners.	Point.	Radius/Size.
Chamfer.	Point "A".	Size = 2
Rounding.	Point "B".	Radius = 4
Rounding.	Point "C".	Radius = 6
Rounding.	Point "D".	Radius = 5
Chamfer.	Point "E".	Size = 3

Spindle parameters.

Machining in the turning direction.

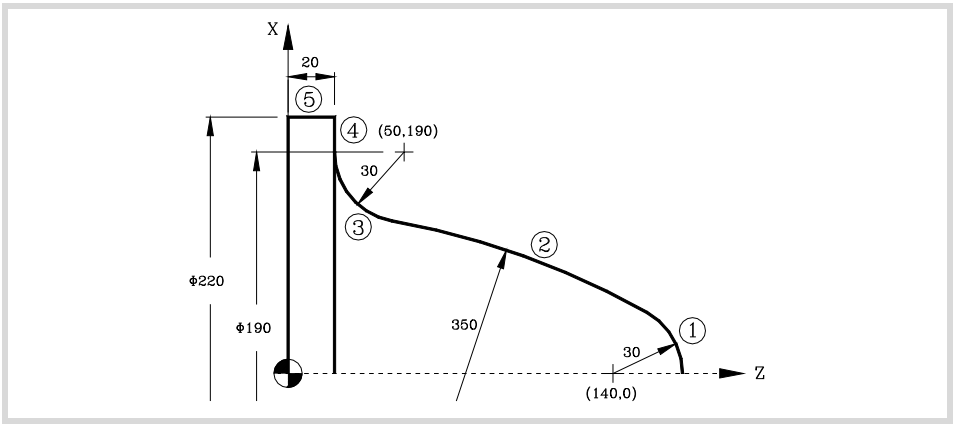
Constant turning speed.

Machining parameters. Roughing.

Δ 2 F 0.8 S 1000 T 3

Machining parameters. Finishing.

δ 0.25 F 0.4 S 1000 T 3



Geometric parameters.

	Outside machining.		Work quadrant.
X 230.0000 Z 180.0000	Starting point.	Dx 1.0000 Dz 1.0000	Safety distance.

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.

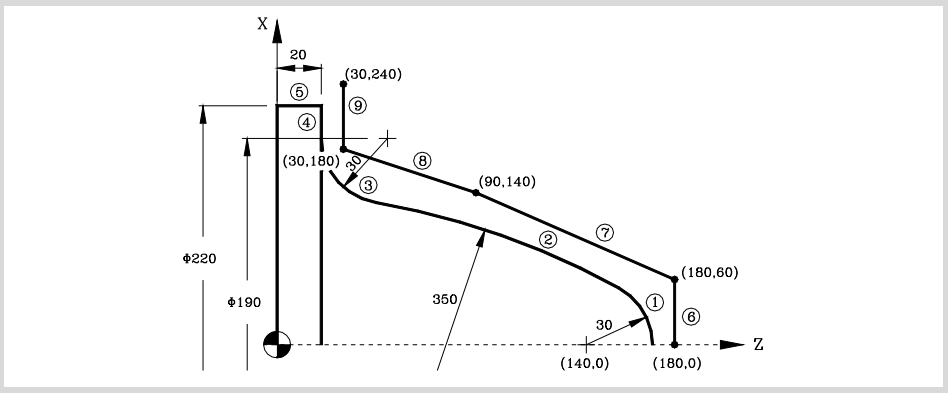
Constant turning speed.

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



Geometric parameters.

	Outside machining.		Work quadrant.
X 230.0000 Z 180.0000	Starting point.	Dx 1.0000 Dz 1.0000	Safety distance.

Define the part profile.

	Section.	Geometry.			
	Starting point.	Z = 170	X = 0		
(1)	Counterclockwise arc.	Zc = 140	Xc = 0	R = 30	
(2)	Counterclockwise arc.	R = 350	Tangency = Yes		
(3)	Clockwise arc.	Zc = 50	Xc = 190	R = 30	Tangency = Yes
• The CNC shows all the possible options for section 2. Select the correct one.					
(4)	Straight line.	Z = 20	X = 220	Tangency = Yes	
• The CNC displays all the possible options between sections 3-4. Select the correct one.					
(5)	Straight line.	Z = 0	X = 220		

Define the raw part profile.

	Section.	Geometry.	
	Starting point.	Z = 180	X = 0
(6)	Straight line.	Z = 180	X = 60
(7)	Straight line.	Z = 90	X = 140
(8)	Straight line.	Z = 30	X = 180
(9)	Straight line.	Z = 30	X = 240

Spindle parameters.

Machining in the turning direction.

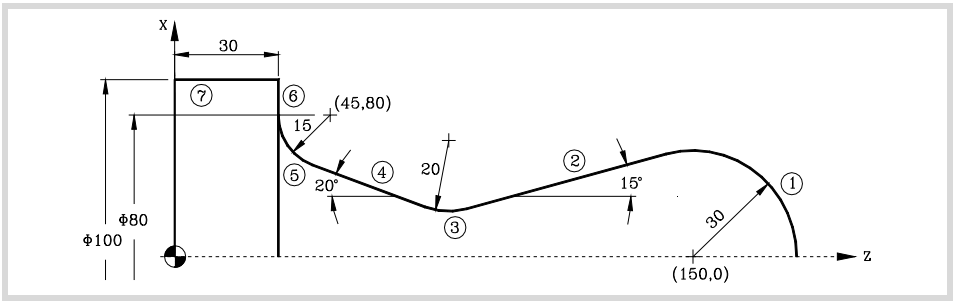
Constant turning speed.

Machining parameters. Roughing.

Δ 2 F 0.8 S 1000 T 3

Machining parameters. Finishing.

δ 0.25 F 0.4 S 1000 T 3



Geometric parameters.

Outside machining.

Work quadrant.

X 110.0000 Starting point.
Z 190.0000

Dx 1.0000 Safety distance.
Dz 1.0000

Define the part profile.

	Section.	Geometry.			
	Starting point.	Z = 180	X = 0		
(1)	Counterclockwise arc.	Zc = 150	Xc = 0	R = 30	
(2)	Straight line.	Angle = 195	Tangency = Yes		
• The CNC displays all the possible options between sections 1-2. Select the correct one.					
(3)	Clockwise arc.	R = 20	Tangency = Yes		
(4)	Straight line.	Angle = 160	Tangency = Yes		
(5)	Clockwise arc.	Z = 30 X = 80	Zc = 45 Xc = 80	R = 15	Tangency = Yes
• The CNC displays all the possible options between sections 4-5. Select the correct one.					
• The CNC shows all the possible options for section 3. Select the correct one.					
(6)	Straight line.	Z = 30	X = 100	Tangency = Yes	
(7)	Straight line.	Z = 0	X = 100		

Spindle parameters.

Machining in the turning direction.

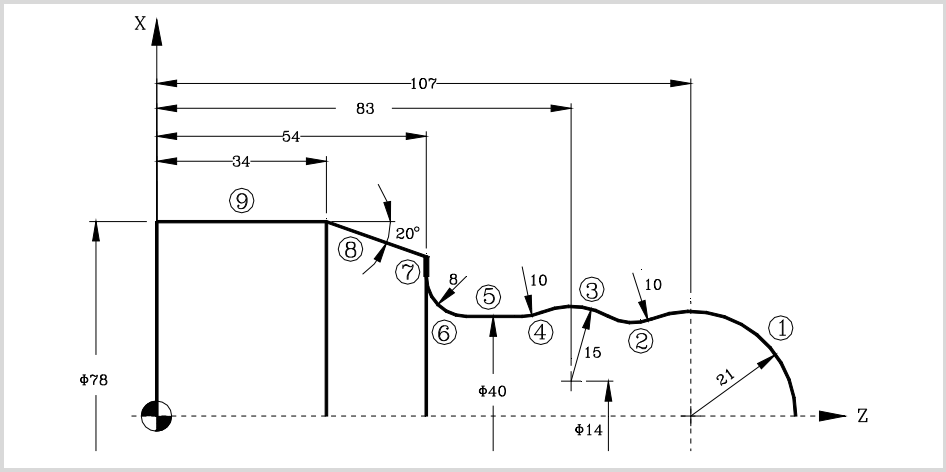
Constant turning speed.

Machining parameters. Roughing.

Δ 2 F 0.8 S 1000 T 3

Machining parameters. Finishing.

δ 0.25 F 0.4 S 1000 T 3



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.

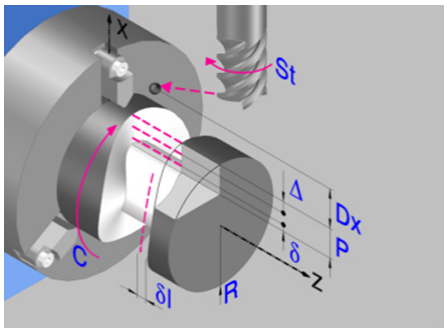
Machining parameters. Roughing.

Δ 2 F 0.8 S 1000 T 3

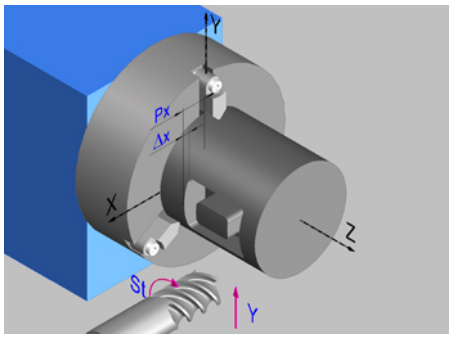
Machining parameters. Finishing.

δ 0.25 F 0.4 S 1000 T 3

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.

Spindle angular position.

ω Angular position of the spindle, in degrees.

Define a profile.

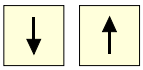
This icon lets define the profile using the profile editor; it may be an existing profile or a new one.

Profile definition.

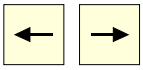
P Name of the program with the profile (between 0 and 999).

- 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, δl) (δ, δl).

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 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 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.

6.3.1 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.

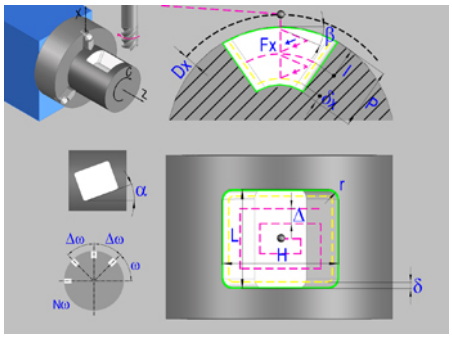
   The cycle eliminates the finishing stocks. If tool radius compensation is active, the cycle runs the indicated number of passes until the final profile is reached; if it is not active, the cycle runs a single pass. This operation is done with the conditions set for the finishing operation.
7. Once the 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.

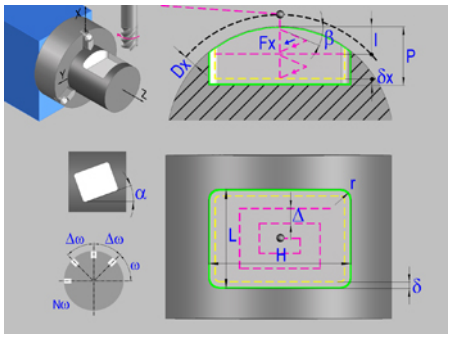
If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.

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).

6.4 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.

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.

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω 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.

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, 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 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.

6.4.1 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 pases 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).

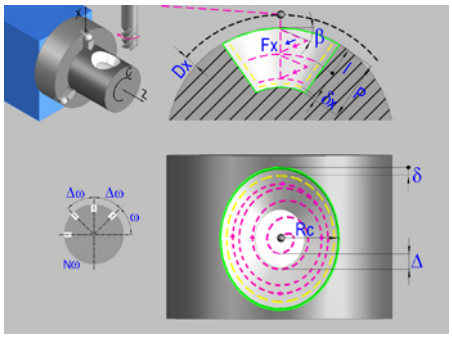
Roughing and finishing in multiple pockets.

If the roughing tool and the finishing tool are different, the cycle executes first all the roughing operations and then all the finishing operations.

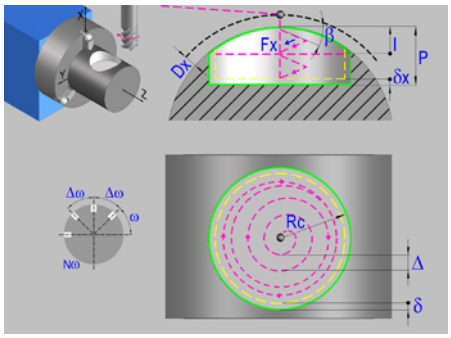
If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- 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.

6.5 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.

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.

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω 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.

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.

6.5.1 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 pases 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).

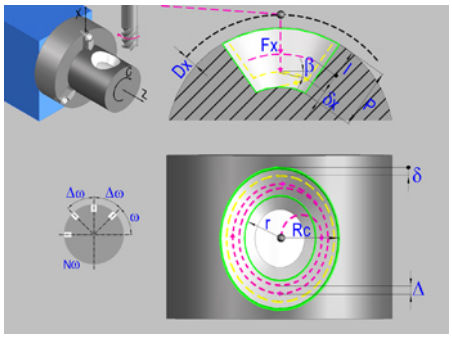
Roughing and finishing in multiple pockets.

If the roughing tool and the finishing tool are different, the cycle executes first all the roughing operations and then all the finishing operations.

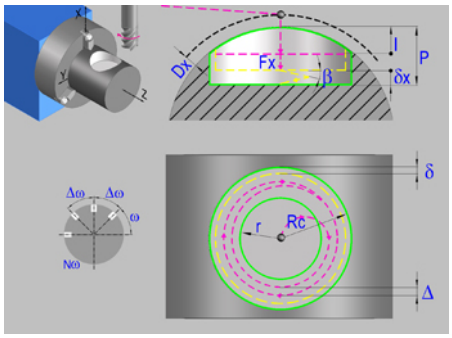
If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- 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.

6.6 Circular pre-emptied pocket cycle in the ZC/YZ plane.



Circular pre-emptied pocket in the ZC plane.



Circular pre-emptied 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.

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.

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω 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-emptied area, it goes in straight in G0, ignoring the penetration angle. If the tool does not fit in the pre-emptied 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.

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 pases 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).

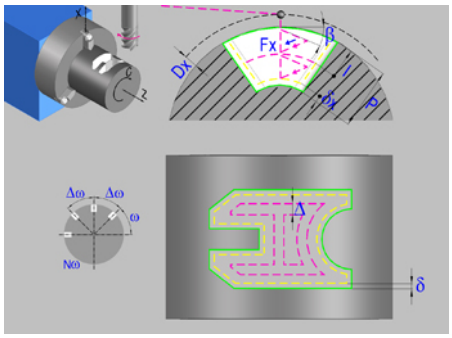
Roughing and finishing in multiple pockets.

If the roughing tool and the finishing tool are different, the cycle executes first all the roughing operations and then all the finishing operations.

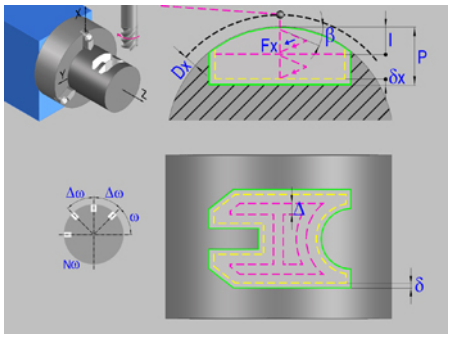
If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- 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.

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



2D profile pocket in the ZC plane.



2D profile pocket in the YZ plane.

Geometric parameters.

Work plane.

ZC plane.

YZ plane.

Name of the program with the profile.

P Name of the program with the profile (between 0 and 999).

Radius of the cylinder (if ZC plane).

R Outside radius of the part.

Coordinates of the plane (if YZ plane).

X X coordinate of the work plane. 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.

Angular position of the machining operations.

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

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω Total number of operations.

Define a profile.

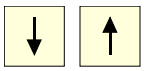
This icon lets define the profile using the profile editor; it may be an existing profile or a new one.

Profile definition.

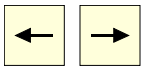
P Name of the program with the profile (between 0 and 999).

- 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.

Fx Penetration feedrate.

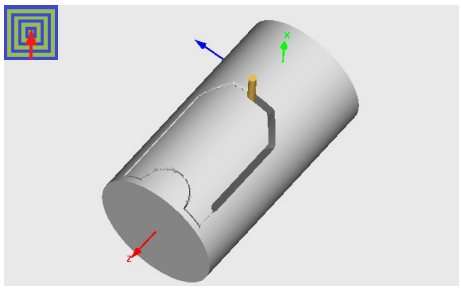
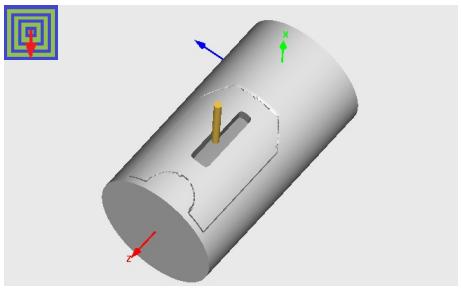
Machining strategy (icon).

Outward-inward concentric machining. The wall of the pocket is machined in the first pass. Depending on the geometry, this option may be faster, but the wear on the tool is greater.

- The starting point is near the initial point of the outer contour, and varies very little if the radius of the roughing tool is changed and does not vary at all if the roughing pass is changed.
- If a drilling operation is programmed before the roughing (or the bottom finishing if there is no roughing operation), the CNC chooses the optimal drilling point, as close as possible to the initial point of the outer contour.

Inward-outward concentric machining. The wall of the pocket is machined in the last pass. Depending on the geometry, this option maybe slower, but there is less wear on the tool. The CNC machines the pocket, attempting to minimize the number of sections where the tool machines over the largest surface, the number of penetrations in part sections and the machining time.

- The starting point does not depend on the initial point of the outer contour and can vary if either the tool radius or the machining pass is changed (milling width). In a pocket without islands, the initial point is found near the center of the pocket. In a pocket with islands, where different areas can be formed that require roughing, the initial point will be located in the center of the largest area.
- If a drilling operation is programmed before the roughing (or the bottom finishing, if there is no roughing operation), the CNC chooses the optimal drilling points (there may be more than one point).

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

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 (icon).

For pockets with islands, the machining of the outer profile is done in the direction indicated by the icon, where as the islands are machined in the opposite 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 (icon).

For pockets with islands, the machining of the outer profile is done in the direction indicated by the icon, where as the islands are machined in the opposite 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.

6.7.1 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 carried out under the conditions set for the roughing operation.
6. Finishing operation (if it has been programmed).

   If the finishing operation has been programmed with another tool, the CNC will change the tool and will move to the change position if so required by the machine.

   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 not 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).

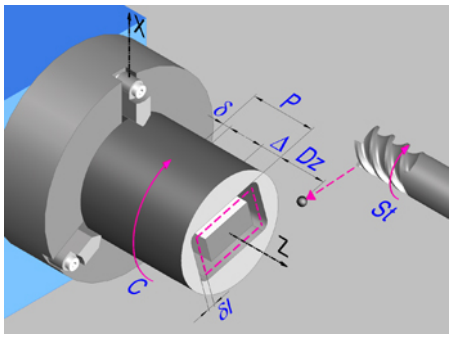
Roughing and finishing in multiple pockets.

If the roughing tool and the finishing tool are different, the cycle executes first all the roughing operations and then all the finishing operations.

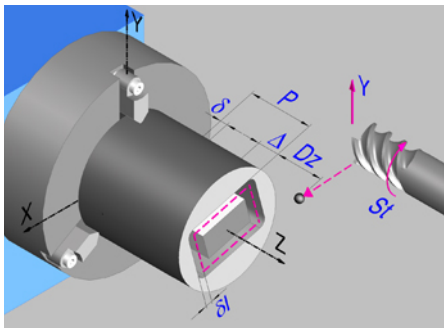
If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- 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.

6.8 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.

Define a profile.

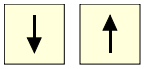
This icon lets define the profile using the profile editor; it may be an existing profile or a new one.

Profile definition.

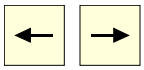
P Name of the program with the profile (between 0 and 999).

- 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, δl) (δ, δl).

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 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 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.

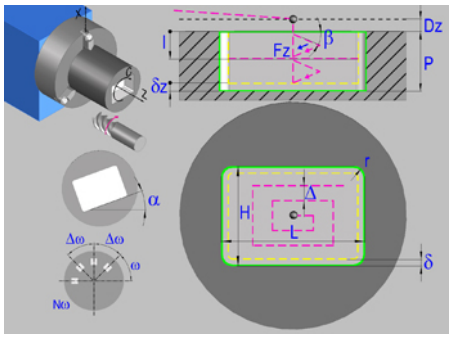
   The cycle eliminates the finishing stocks. If tool radius compensation is active, the cycle runs the indicated number of passes until the final profile is reached; if it is not active, the cycle runs a single pass. This operation is done with the conditions set for the finishing operation.
7. Once the 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.

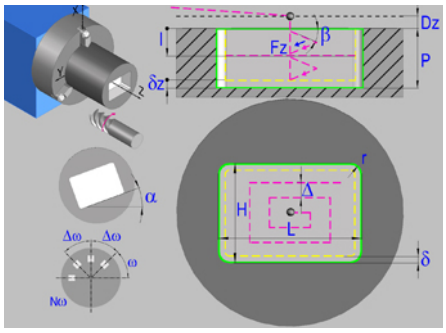
If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.

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).

6.9 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.

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.

$N\omega$ 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.

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.

6.9.1 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 pases 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).

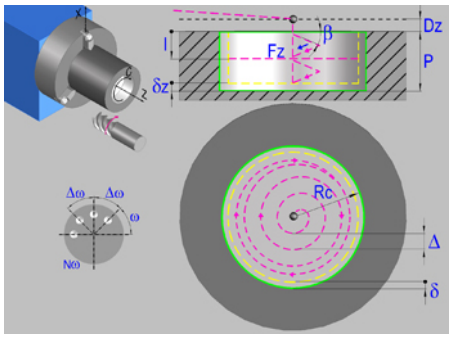
Roughing and finishing in multiple pockets.

If the roughing tool and the finishing tool are different, the cycle executes first all the roughing operations and then all the finishing operations.

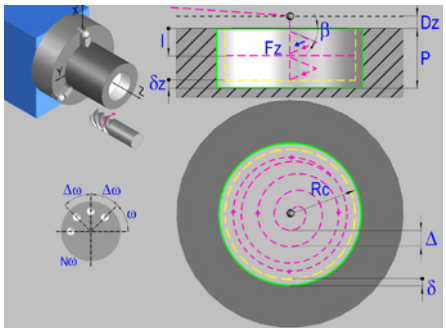
If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- 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.

6.10 Circular pocket cycle in the XC/XY plane.



Circular pocket in the XC plane.



Circular pocket in the XY plane.

Geometric parameters.

Work plane.

XC plane.

XY plane.

Coordinates of the center of the pocket.

Xc, Cc Coordinates of the center of the pocket (if XC plane). 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.

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.

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω 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.

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.

6.10.1 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 pases 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).

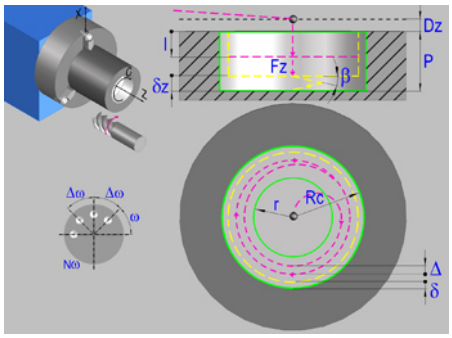
Roughing and finishing in multiple pockets.

If the roughing tool and the finishing tool are different, the cycle executes first all the roughing operations and then all the finishing operations.

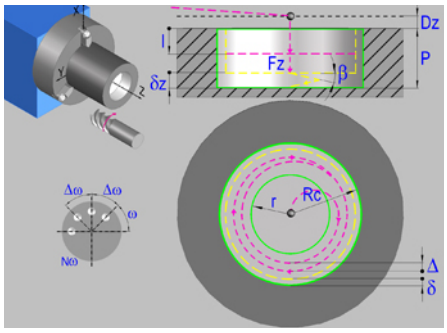
If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- 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.

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



Circular pre-emptied pocket in the XC plane.



Circular pre-emptied 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.

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.

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω 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-emptied area, it goes in straight in G0, ignoring the penetration angle. If the tool does not fit in the pre-emptied 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.

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 pases 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).

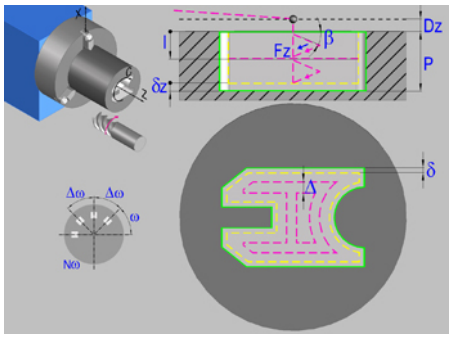
Roughing and finishing in multiple pockets.

If the roughing tool and the finishing tool are different, the cycle executes first all the roughing operations and then all the finishing operations.

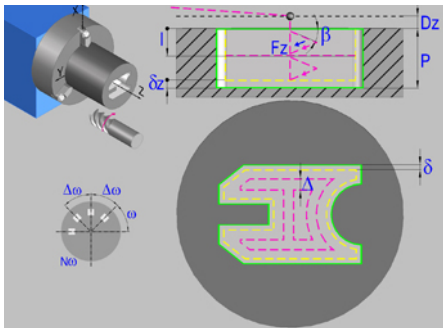
If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- 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.

6.12 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.

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω Total number of operations.

Define a profile.

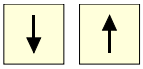
This icon lets define the profile using the profile editor; it may be an existing profile or a new one.

Profile definition.

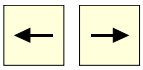
P Name of the program with the profile (between 0 and 999).

- 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.

Penetration feedrate.

Fz Penetration feedrate.

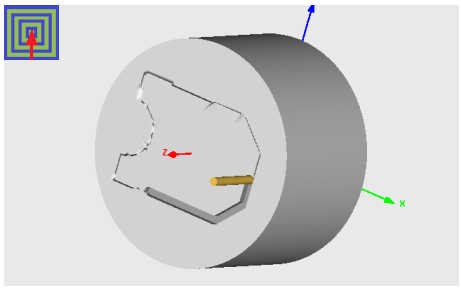
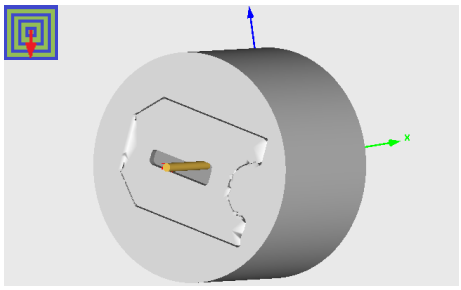
Machining strategy (icon).

Outward-inward concentric machining. The wall of the pocket is machined in the first pass. Depending on the geometry, this option may be faster, but the wear on the tool is greater.

- The starting point is near the initial point of the outer contour, and varies very little if the radius of the roughing tool is changed and does not vary at all if the roughing pass is changed.
- If a drilling operation is programmed before the roughing (or the bottom finishing if there is no roughing operation), the CNC chooses the optimal drilling point, as close as possible to the initial point of the outer contour.

Inward-outward concentric machining. The wall of the pocket is machined in the last pass. Depending on the geometry, this option maybe slower, but there is less wear on the tool. The CNC machines the pocket, attempting to minimize the number of sections where the tool machines over the largest surface, the number of penetrations in part sections and the machining time.

- The starting point does not depend on the initial point of the outer contour and can vary if either the tool radius or the machining pass is changed (milling width). In a pocket without islands, the initial point is found near the center of the pocket. In a pocket with islands, where different areas can be formed that require roughing, the initial point will be located in the center of the largest area.
- If a drilling operation is programmed before the roughing (or the bottom finishing, if there is no roughing operation), the CNC chooses the optimal drilling points (there may be more than one point).

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

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 (icon).

For pockets with islands, the machining of the outer profile is done in the direction indicated by the icon, where as the islands are machined in the opposite 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 (icon).

For pockets with islands, the machining of the outer profile is done in the direction indicated by the icon, where as the islands are machined in the opposite 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.

6.12.1 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 carried out under the conditions set for the roughing operation.
6. Finishing operation (if it has been programmed).

   If the finishing operation has been programmed with another tool, the CNC will change the tool and will move to the change position if so required by the machine.

   The cycle eliminates the finishing stocks. This operation is done with the conditions set for the finishing operation.

   The cycle offers two finishing stocks; at the bottom and on the walls. If both stocks are programmed ($\delta, \delta x$), the cycle executes the finishing operation at the bottom first and then on the walls. The finishing on the walls is carried out in as many penetration passes as indicated by parameter N. If there are no side finishing stocks, but parameter N is not 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.

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

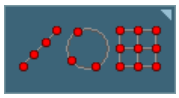
REF. 2109

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.

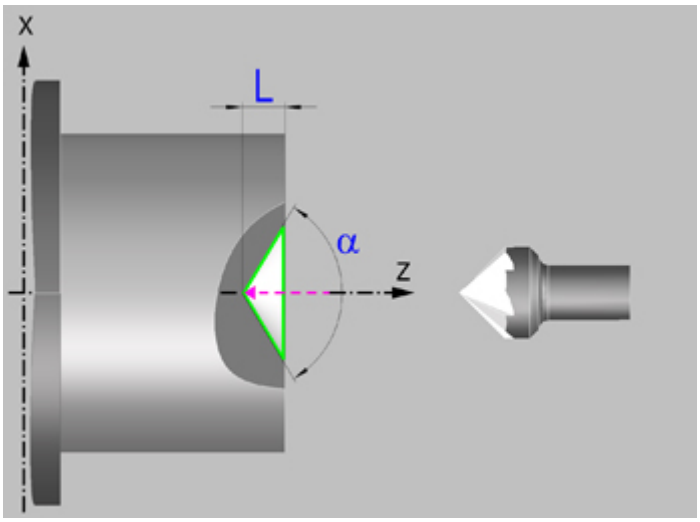
To associate multiple machining to a cycle, first select and define a machining cycle from those allowed. Then, without quitting the editing of the cycle, press the softkey associated with multiple machining and select one of them.

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.

7.1 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.

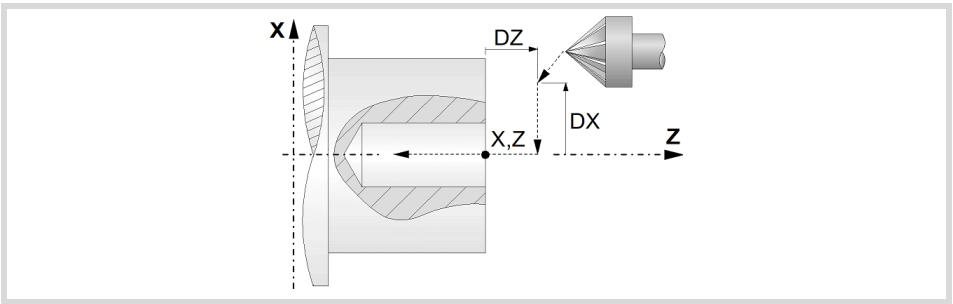
Although the center punching is normally done in the rotation center, the CNC enables X to be defined with a value other than 0 and make grooves in the front face of the part. For center punching, X must be 0.

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.

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.

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.

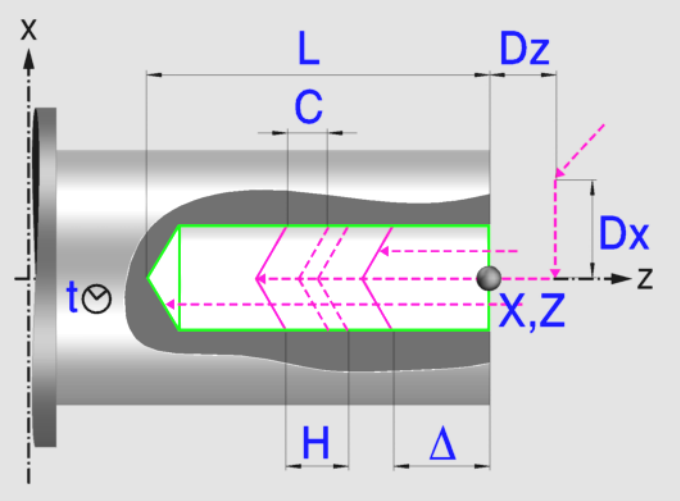
7.1.1 Basic operation.

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).

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

7.2 Drilling.



Geometric parameters.

Coordinates of the starting point

X X coordinate of the starting point; if not programmed, the cycle will assume a 0 value. The coordinate must be given in the active units, radius or diameter.

Z Z coordinate of the starting point.

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

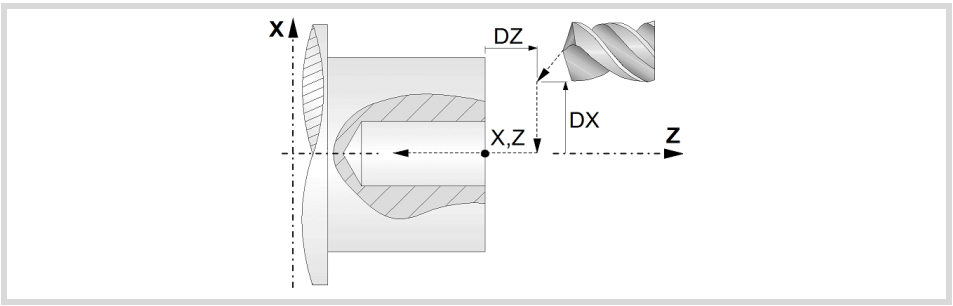
Although the drilling is normally done in the rotation center, the CNC enables X to be defined with a value other than 0 and make grooves in the front face of the part. For drilling, X must be 0.

Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dx, Dz Safety distance referred to the starting point.

The value of the safety distance on X is always defined in radius. The value of the safety distance in Z is always assumed with positive values.



Drilling depth.

L Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.

Spindle parameters.

Spindle turning direction

The cycle lets select the spindle turning direction.

Spindle clockwise.

Spindle counterclockwise.

Coolant status.

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

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

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

Maximum spindle speed.

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

Spindle range (gear).

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

Machining parameters.

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

Δ Maximum drilling peck (step).

KΔ Reduction factor of the drilling peck; if not programmed, the cycle will assume a factor of 1.

δ Minimum drilling step (peck); constant-peck or variable-peck drilling.

The first drilling peck will be "Δ", the second one "KΔ D", the third one "KΔ (KΔ D)" and so on until reaching the minimum peck; in other words, from the second peck on, the new peck will be the result of multiplying the previous peck by KΔ.

If the minimum drilling peck is greater than the maximum peck ($\delta > \Delta$), the cycle carries out a drilling at a constant peck equal to the minimum peck "δ". If the minimum drilling peck is smaller than the maximum peck ($\delta < \Delta$), the cycle carries out a drilling at a variable peck.

Withdrawal and approach distance in each penetration.

H Withdrawal distance after each peck; if not programmed, the cycle will assume a value of 1 mm. When programming H=0, it will withdraw up to the Z coordinate of the safety point.

C Approach distance in each penetration. If parameter C is not assigned a value or 0 is assigned, it assumes 1 mm by default.

Dwell at the bottom.

t The waiting time in seconds, after drilling, until the withdrawal movement starts.

Penetration feedrate.

F Penetration feedrate.

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.

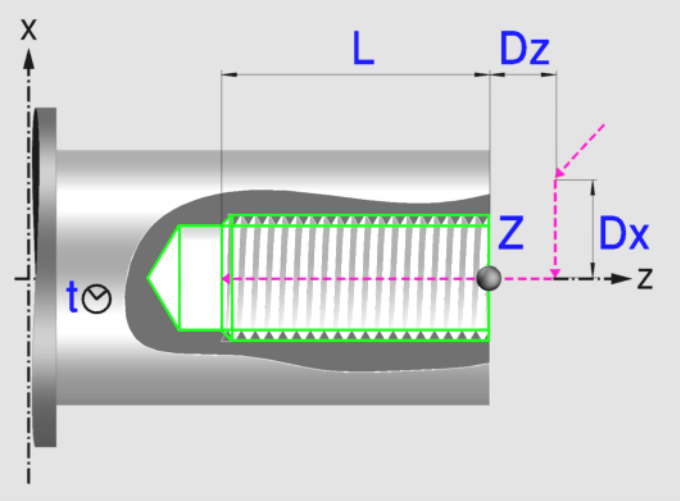
   If the minimum drilling peck is greater than the maximum peck ($\delta > \Delta$), the cycle carries out a drilling at a constant peck equal to the minimum peck "δ". If the minimum drilling peck is smaller than the maximum peck ($\delta < \Delta$), the cycle carries out a drilling at a variable peck.

   In a drilling operation at variable peck, parameter Δdefines the drilling peck and KΔ the reduction factor for this peck. 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.

   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. Than, 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).

7.3 Tapping.



Geometric parameters.

Coordinates of the starting point

Z Z coordinate of the starting point.

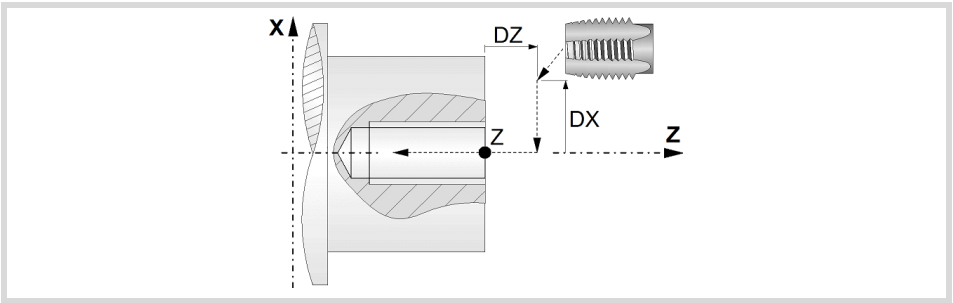
To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dx, Dz Safety distance referred to the starting point.

The value of the safety distance on X is always defined in radius. The value of the safety distance in Z is always assumed with positive values.



Tapping depth.

L Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.

Tapping type.

Tapping with a clutch.

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

Spindle parameters.

Spindle turning direction

The cycle lets select the spindle turning direction.

Spindle clockwise.

Spindle counterclockwise.

Coolant status.

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

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

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

Spindle range (gear).

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

Machining parameters.

Define the thread.

Define the thread with the feedrate and the turning direction.

Define the thread with the pitch and the turning direction.

F Penetration feedrate.

P Thread pitch.

Dwell at the bottom.

t The waiting time in seconds, after tapping, until the withdrawal movement starts.

Spindle speed.

S Spindle speed.

Tool data.

T Tool.

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

Machining M functions.

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

Deactivating the execution of M functions before machining.

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

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 taping 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.

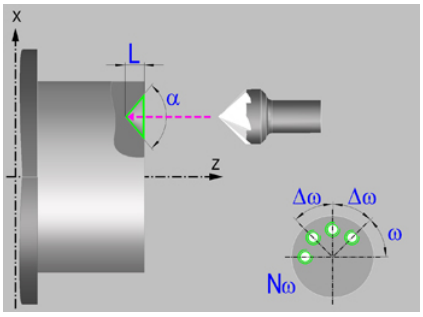
   If a dwell time at the bottom has been defined, the spindle stops and, after this time, the spindle starts in the other direction and withdraws at working feedrate at the approach point.
7. Once the operation or cycle has ended, the tool will return to the safety point.
8. If it is rigid tapping, the CNC will stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

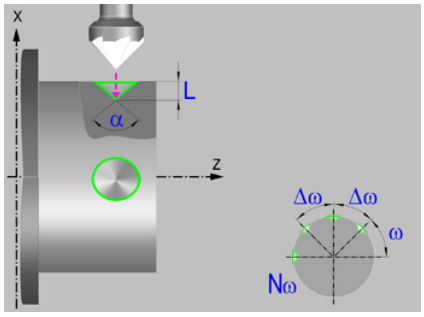
   When tapping with a clutch, the CNC will not stop the spindle.

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.

7.4 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.

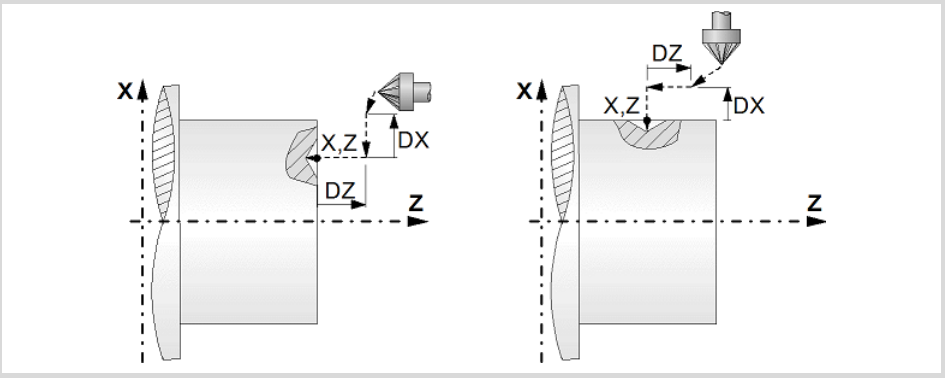
C — C coordinate of the first positioning movement.

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.



- 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.

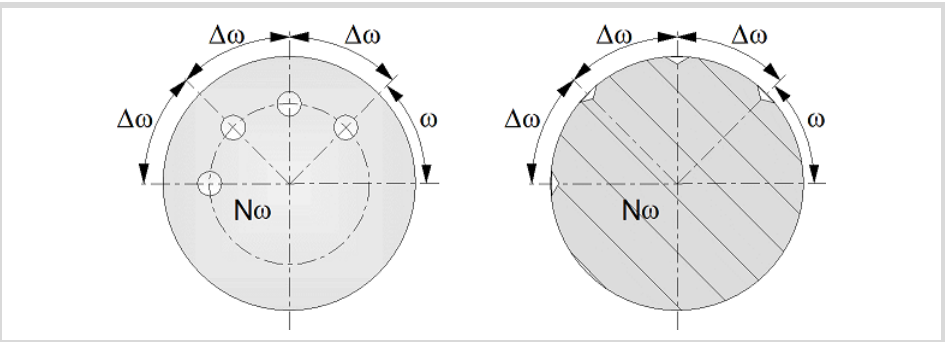
ϕ — Center-punching diameter.

Angular position of the machining operations.

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

Δω — Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω — Total number of operations.



Live-tool parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

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.

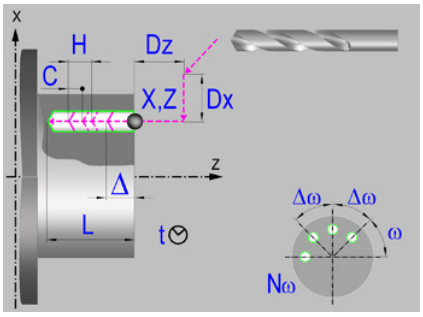
7.4.1 Basic operation.

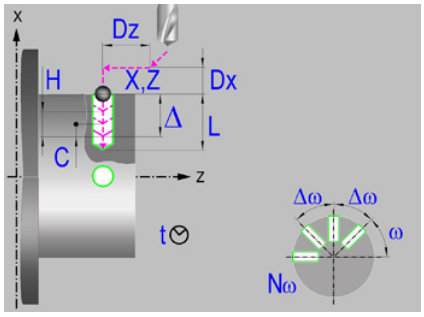
The machining steps in this cycle are as follows:

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.

7.5 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.

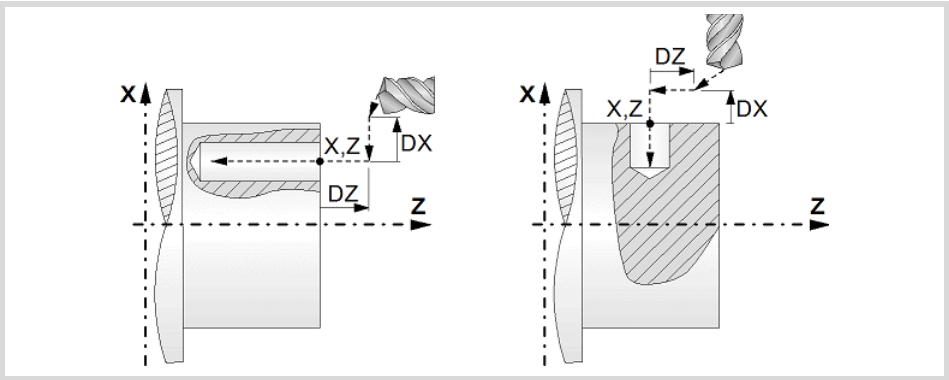
To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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.

- 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.

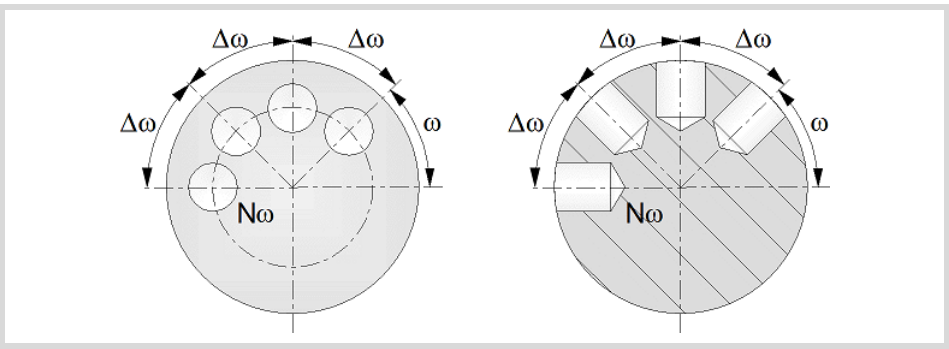
L Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.

Angular position of the machining operations.

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

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω Total number of operations.



Live-tool parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

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).

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 (δ > Δ), 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 (δ < Δ), 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.

Machining M functions.

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

Deactivating the execution of M functions before machining.

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

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

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.

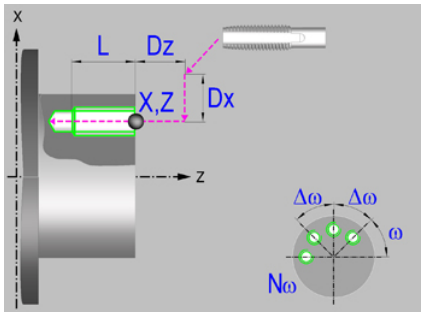
   In a drilling operation at variable peck, parameter Δdefines the drilling peck and KΔ the reduction factor for this peck. The first drilling peck will be "Δ", the second one "KΔ D", the third one "KΔ (KΔ D)" and so on until reaching the minimum peck; in other words, from the second peck on, the new peck will be the result of multiplying the previous peck by KΔ.

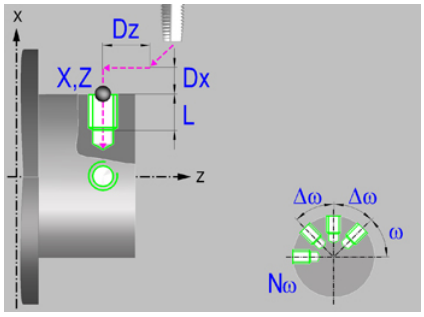
   In either case, the cycle repeats the following steps until reaching the programmed depth.

   First, rapid approach to the distance indicated by parameter C with respect to the previous penetration. Than, 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.

7.6 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.

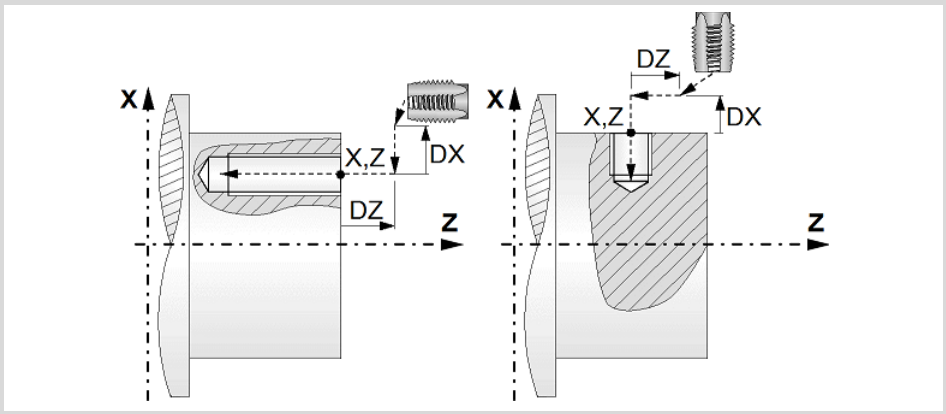
To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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.

- Machining on the face of the part. The value of the safety distance on X is always defined in radius and it may be either positive or negative. The value of the safety distance in Z is always assumed with positive values.
- Machining on the side of the part. The value of the safety distance on X is always defined in radius and it is always assumed with positive values. The value of the safety distance in Z may be either negative or positive.



Tapping type.

Tapping with a clutch.

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

Programming of the tapping operation.

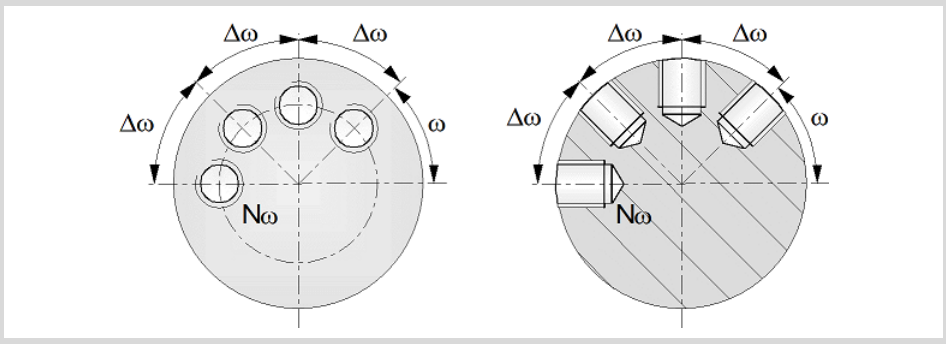
L — Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.

Angular position of the machining operations.

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

Δω — Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω — Total number of operations.



Live-tool parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

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.

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.

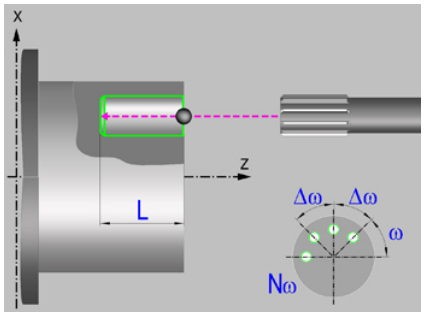
7.6.1 Basic operation.

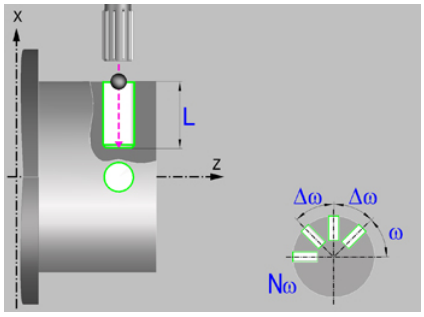
The machining steps in this cycle are as follows:

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 (Io).
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.

7.7 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.

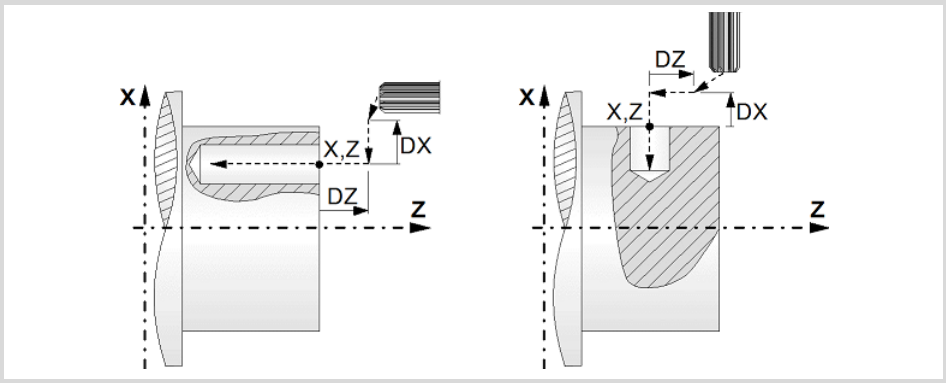
To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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.

- 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.

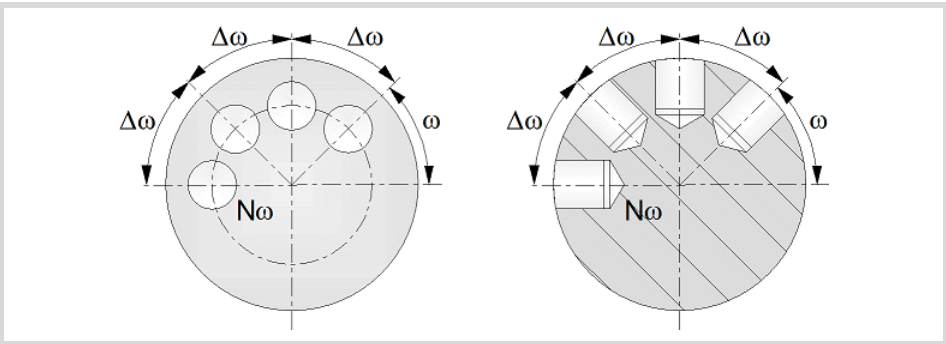
L Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.

Angular position of the machining operations.

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

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω Total number of operations.



Live-tool parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

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 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.

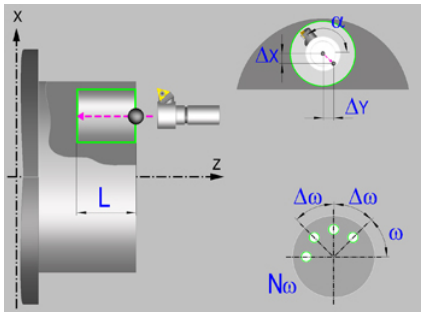
7.7.1 Basic operation.

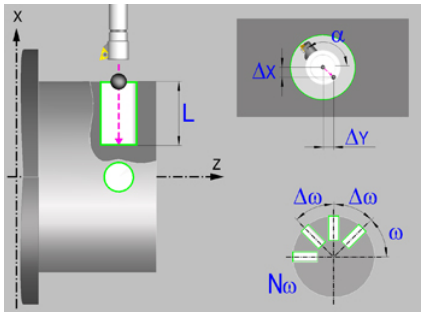
The machining steps in this cycle are as follows:

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.

7.8 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.

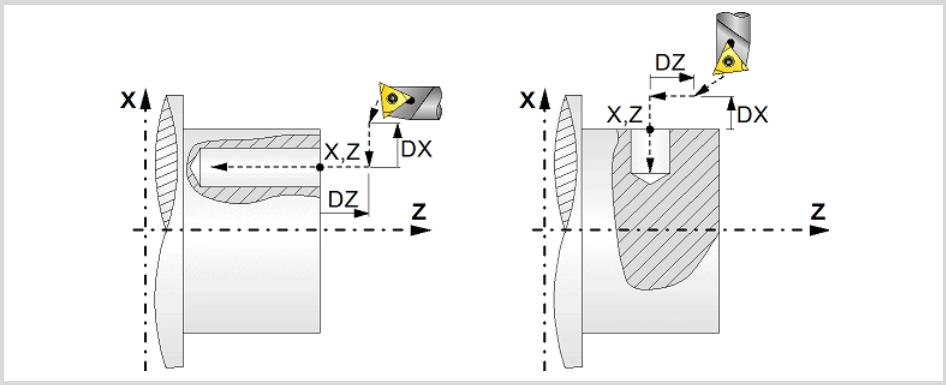
To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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.

- 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.

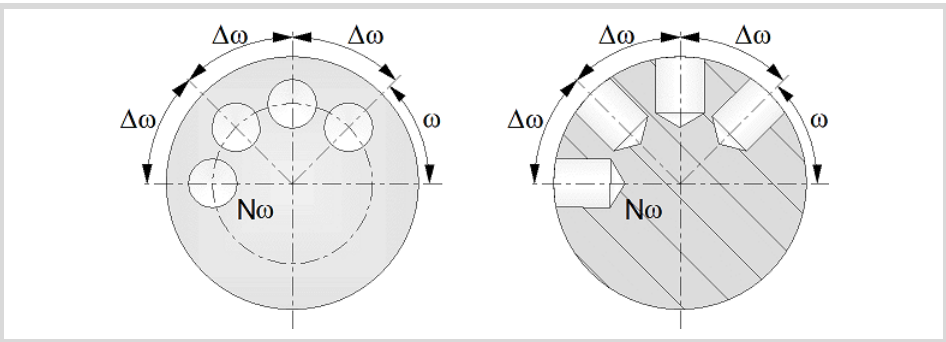
L Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.

Angular position of the machining operations.

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

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω Total number of operations.



Live-tool parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

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.

Type of withdrawal.

Withdrawal at work feedrate while the spindle is turning.

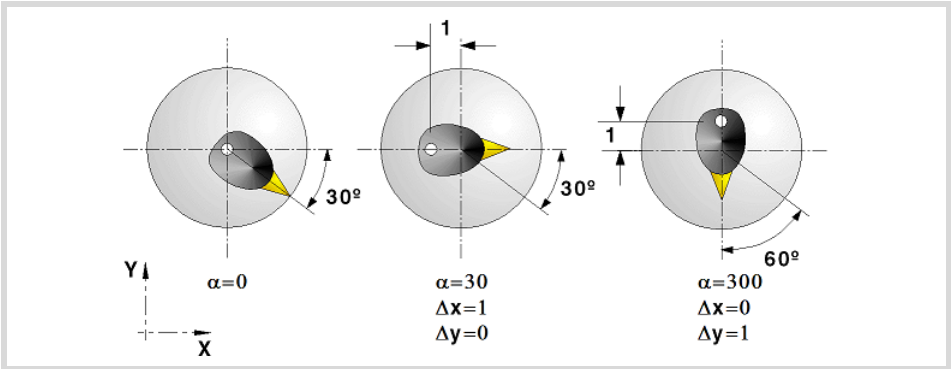
Rapid withdrawal while the spindle is stopped and the tool is touching the surface.

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.

7.8.1 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.

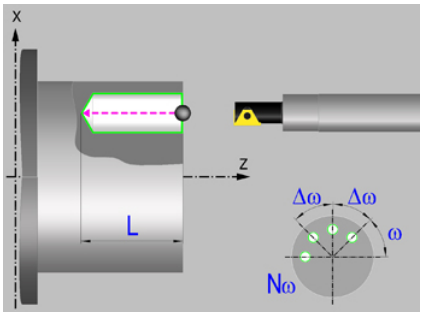
   Withdrawal at work feedrate while the spindle is turning.

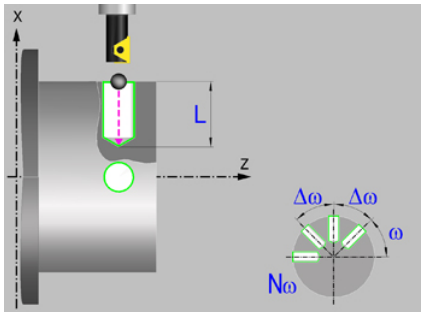
   Rapid withdrawal while the spindle is stopped and the tool is touching the surface.

   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.

7.9 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.

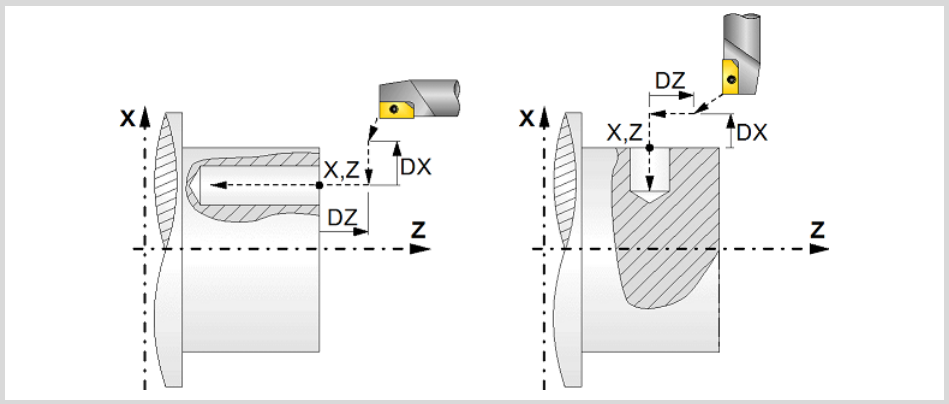
To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

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.



Programming the machining operation.

L Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.

Ø Hole diameter.

Øk Pre-hole diameter.

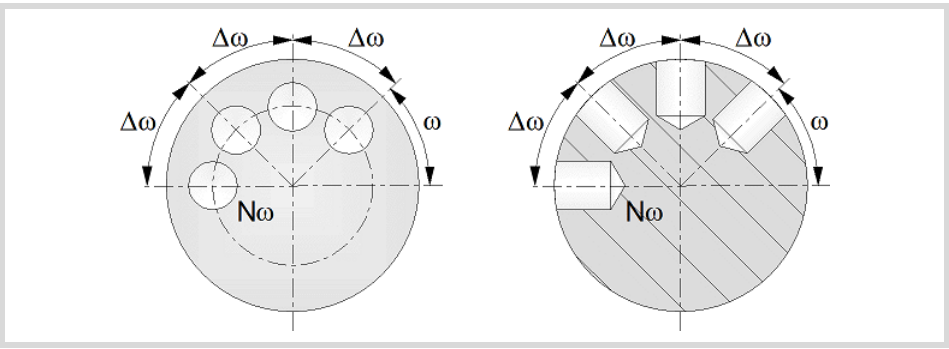
When starting from a hole that was machined earlier, parameter Øk defines the diameter of that hole; if not programmed or programmed with a 0 value, the cycle assumes that there is no previous hole.

Angular position of the machining operations.

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

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω Total number of operations.



Live-tool parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

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.

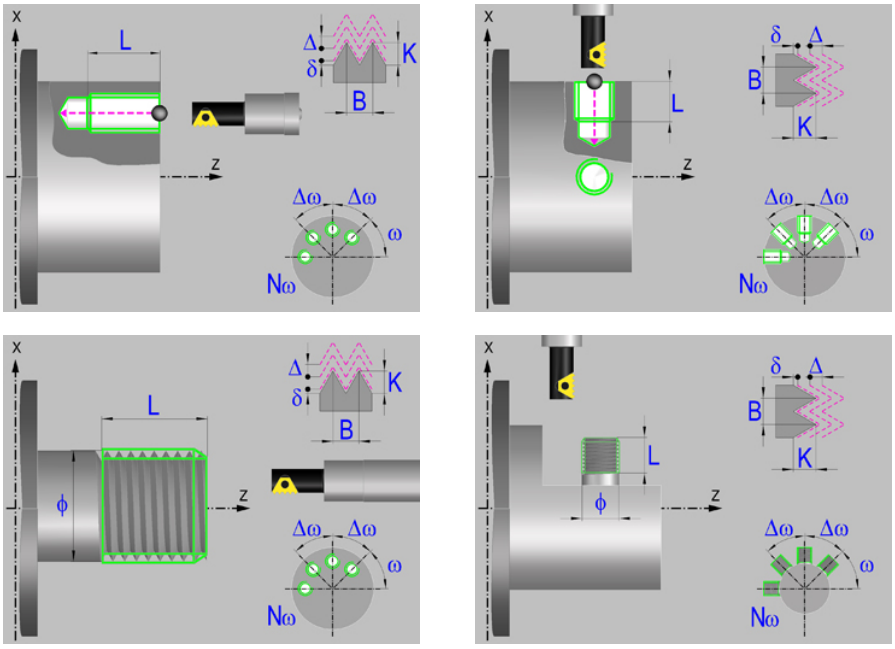
7.9.1 Basic operation.

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.

7.10 Multiple thread milling.

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



Geometric parameters.

Machining on the face or side of the part.

Machining on the face of the part.

Machining on the side of the part.

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

Inside or outside machining.

Inside machining.

Outside machining.

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

Coordinates of the profile's starting and final points.

X — X coordinate of the center of the hole or of the boss. The coordinate must be given in the active units, radius or diameter.

Z — Z coordinate of the center of the hole or of the boss.

Y — Y coordinate of the first positioning movement.

Yf — Y coordinate for the withdrawal movement after the cycle ends; if not programmed, there is no Y axis withdrawal. If the cycle is machining in multiple mode, the Y withdrawal point is only applied to the last cycle.

This parameter lets withdraw the tool to the right position in order to be able to continue with the turning operations that do not use the Y axis. If the next cycle also uses the Y axis, there is no need to withdraw (retract) the axis.

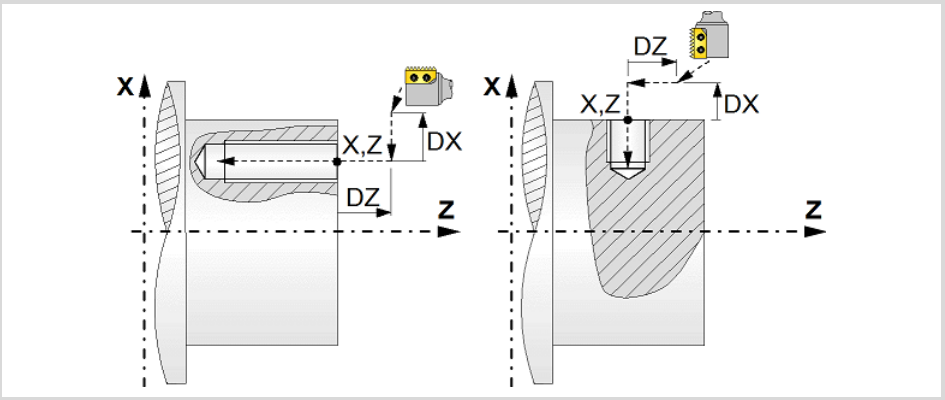
To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dx, Dz Safety distance referred to the starting point.

- Machining on the face of the part. The value of the safety distance on X is always defined in radius and it may be either positive or negative. The value of the safety distance in Z is always assumed with positive values.
- Machining on the side of the part. The value of the safety distance on X is always defined in radius and it is always assumed with positive values. The value of the safety distance in Z may be either negative or positive.



Programming the machining operation.

L Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.

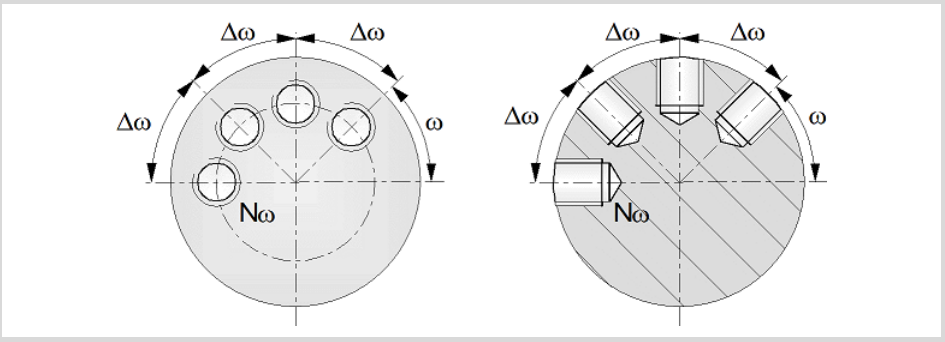
Ø Nominal thread diameter.

Angular position of the machining operations.

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

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω Total number of operations.



Live-tool parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

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.

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.

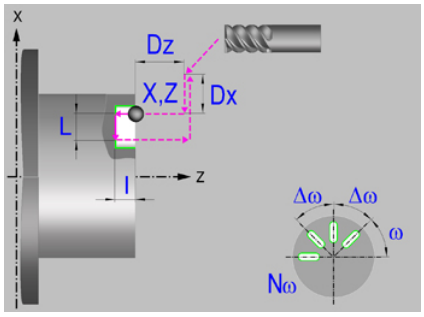
  Rapid movement to the thread entry point of the next threading path.

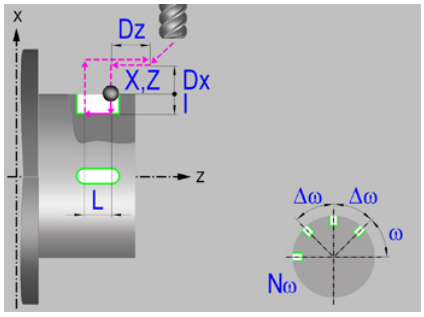
  Repetition of the previous 2 steps until reaching the bottom of the thread. It must be borne in mind that in the last helical exit, the exit point will exceed the coordinate of the bottom of the thread along the longitudinal axis.

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.

7.11 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.

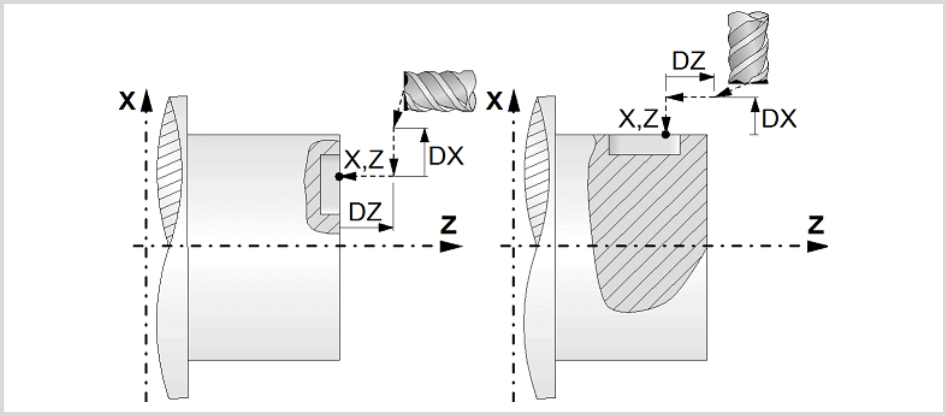
Safety distance.

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.

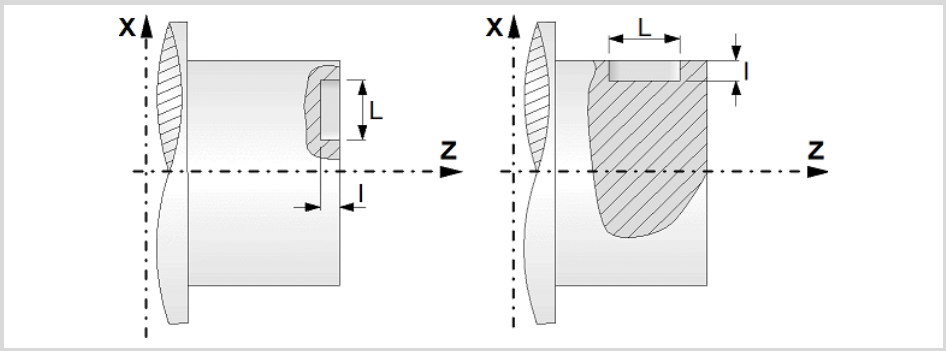
- Machining on the side of the part. The value of the safety distance on X is always defined in radius and it is always assumed with positive values. The value of the safety distance in Z may be either negative or positive.



Programming of the drilling operation.

I Depth of the slot.

L Length of the slot.

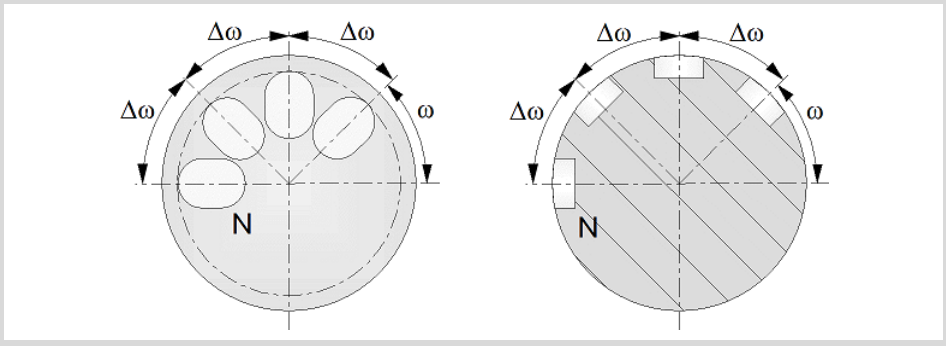


Angular position of the machining operations.

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

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω Total number of operations.



Live-tool parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

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.

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.

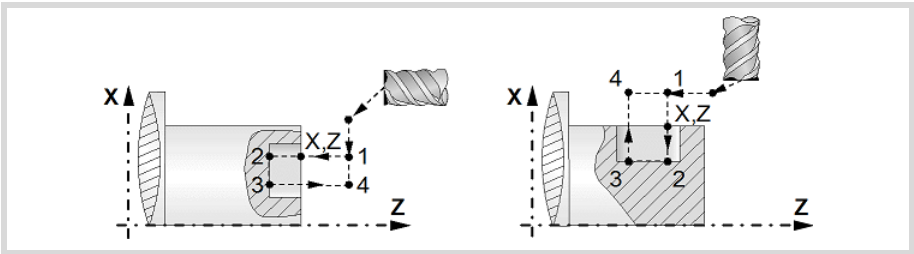
7.11.1 Basic operation.

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.

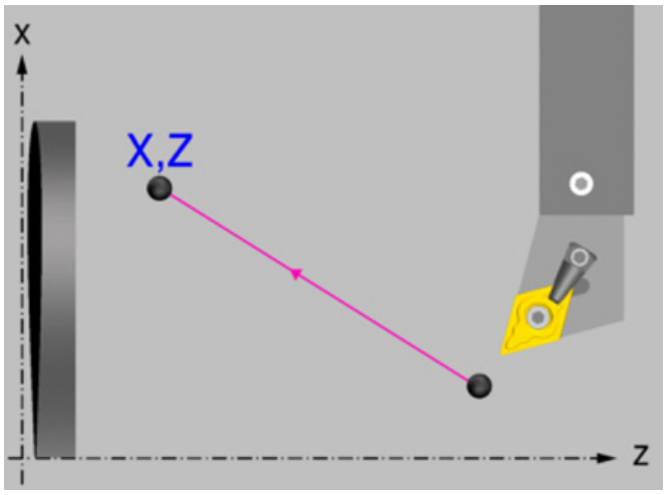
CANNED CYCLES OF THE EDITOR. POSITIONING.

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.



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.

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.

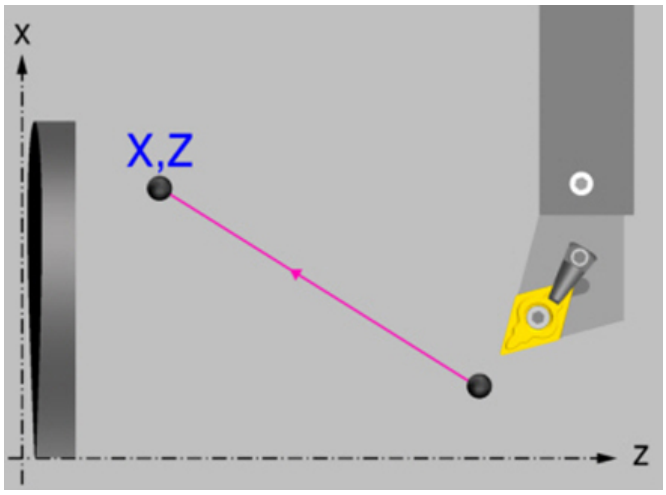
S Spindle turning speed (rpm) or cutting speed.

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.

8.2 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.

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.

FAGOR

FAGOR AUTOMATION

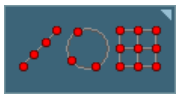
CNCelite
8058 8060
8065

REF. 2109

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.

9

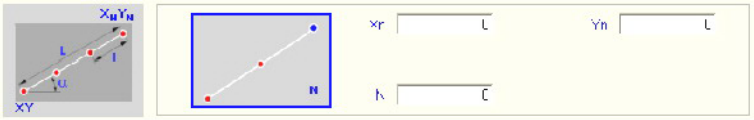
When pressing the softkey, the editor shows the cycle used last. When pressing the same softkey again, the menu shows all the cycles of the group.



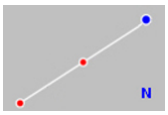
- Multiple machining in straight line
- Multiple machining in arc pattern
- Multiple machining in rectangular pattern
- Multiple machining in grid pattern
- Random multiple machining (several points defined by the user).
- Without multiple machining.

9.1 Multiple machining in straight line.

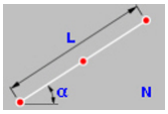
There are 5 different ways to define the machining operation. Use the icon to select the desired option.



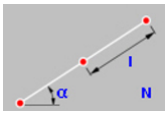
Positioning options.



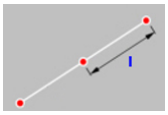
End point and number of machining operations.



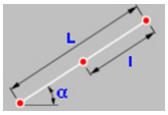
Length, angle and number of machining operations.



Distance between machining operations, angle and number of machining operations.



End point and distance between two machining operations.

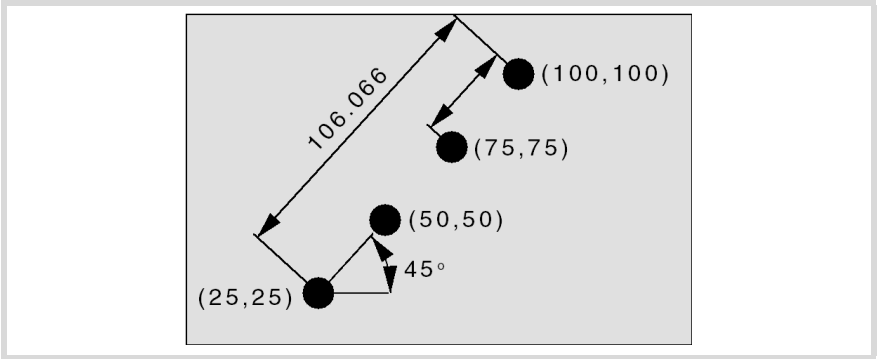


Length, angle and distance between two machining operations.

Xn, Yn	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.

9.1.1 Programming example.

The canned cycle defined at point X25, Y25 is to be repeated at the rest of the points.



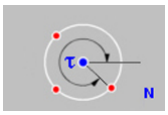
We now show the 5 possible ways to define it.

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

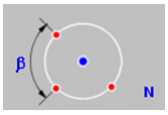
9.2 Multiple machining in arc pattern.

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

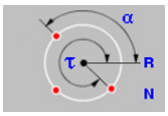
Positioning options.



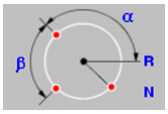
Arc center coordinates, number of machining operations and final angle of the arc.



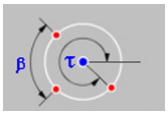
Arc center coordinates, number of machining operations and angular distance between two machining operations.



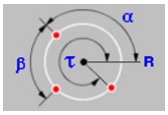
Arc radius, number of machining operations and starting and final angles of the arc.



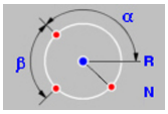
Arc radius, number of machining operations, starting angle of the arc and angular distance between two machining operations.



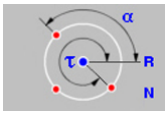
Arc center coordinates, final angle of the arc and angular distance between two machining operations.



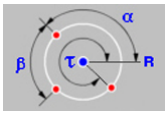
Arc radius, angular distance between two machining operations and starting and final angles of the arc.



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.

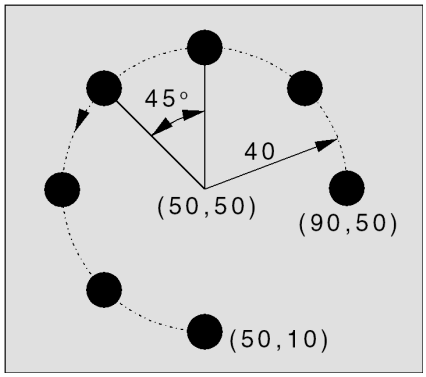


Arc center coordinates, arc radius, angular distance between two machining operations and starting and final angles of the arc.

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.

9.2.1 Programming example.

The canned cycle defined at point X90, Y50 is to be repeated at the rest of the points.

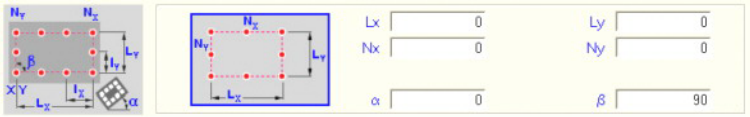


We now show the 9 possible ways to define it.

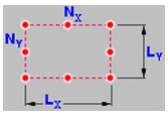
1)	Arc center coordinates.	Xa 50, Ya 50
	Total number of machining operations.	N 7
	Angle of the end point.	τ 270
2)	Arc center coordinates.	Xa 50, Ya 50
	Total number of machining operations.	N 7
	Angular distance between machining operations.	β 45
3)	Radius of the arc.	R 40
	Total number of machining 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

9.3 Multiple machining in rectangular pattern.

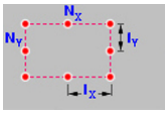
There are 3 ways to define the machining operation. Use the icon to select the desired option.



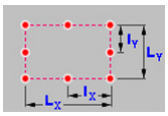
Positioning options.



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.

Lx, Ly — Length of the rectangle, on each axis.

Nx, Ny — Number of machining operations (including that of the cycle definition point) on each axis.

Ix, Iy — Distance between two machining operations, on each axis.

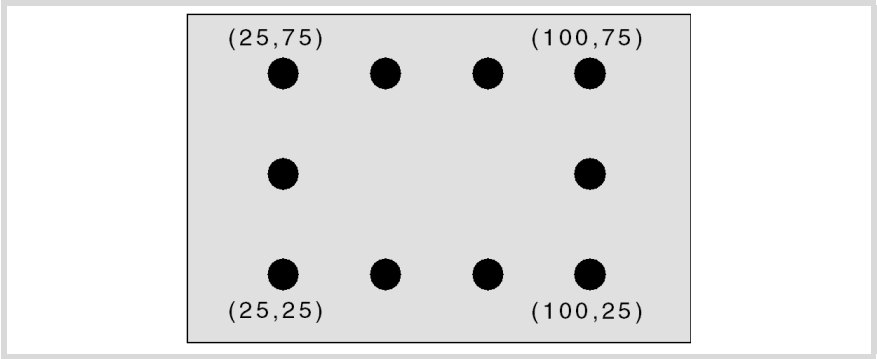
α — Angle of the rectangle with the abscissa axis.

β — Angle between both sides.

The cycle assumes as starting point the lower left one; if it is not, define the machining length (Lx, Ly) with the proper sign or the distance between machining operations (Ix, Iy).

9.3.1 Programming example.

The canned cycle defined at point X25, Y25 is to be repeated at the rest of the points.

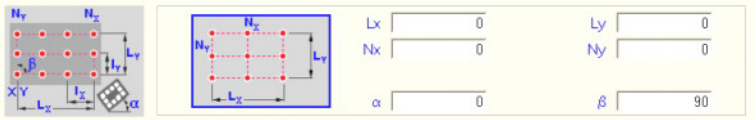


We now show the 3 possible ways to define it.

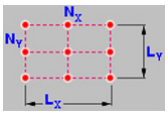
1)	Length of each side of the rectangle.	Lx 75, Ly 50
	Number of machining operations.	Nx 4, Ny 3
	Angle with respect to the abscissa axis.	α 0
	Angle between paths.	β 90
2)	Number of machining operations.	Nx 4, Ny 3
	Distance between machining operations.	Ix 25, Iy 25
	Angle with respect to the abscissa axis.	α 0
	Angle between paths.	β 90
3)	Length of each side of the rectangle.	Lx 75, Ly 50
	Distance between machining operations.	Ix 25, Iy 25
	Angle with respect to the abscissa axis.	α 0
	Angle between paths.	β 90

9.4 Multiple machining in grid pattern.

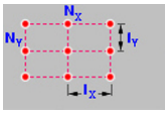
There are 3 ways to define the machining operation. Use the icon to select the desired option.



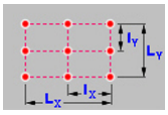
Positioning options.



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.

Lx, Ly — Length of the grid, on each axis.

Nx, Ny — Number of machining operations (including that of the cycle definition point) on each axis.

Ix, Iy — Distance between two machining operations, on each axis.

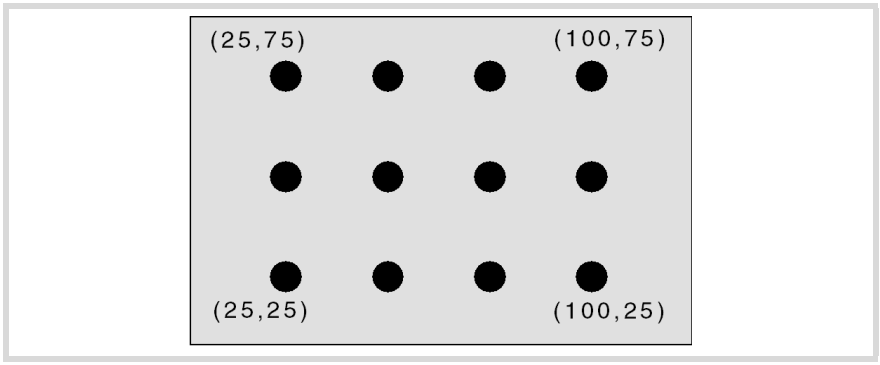
α — Angle of the grid with the abscissa axis.

β — Angle between both sides.

The cycle assumes as starting point the lower left one; if it is not, define the machining length (Lx, Ly) with the proper sign or the distance between machining operations (Ix, Iy).

9.4.1 Programming example.

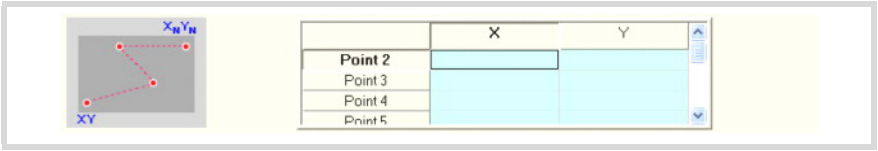
The canned cycle defined at point X25, Y25 is to be repeated at the rest of the points.



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.	Ix 25, Iy 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.	Ix 25, Iy 25
	Angle with respect to the abscissa axis.	α 0
	Angle between paths.	β 90

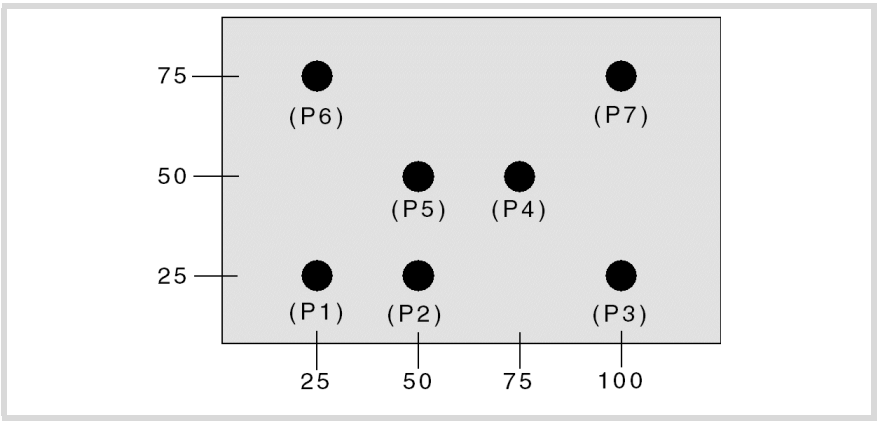
9.5 Random multiple machining.



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.



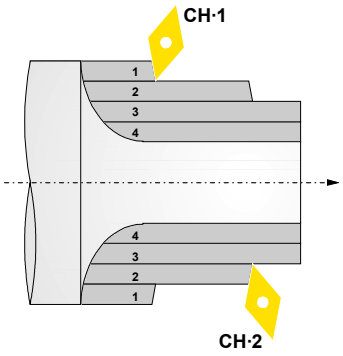
The canned cycle is defined at point (P1) X25, Y25. The rest of the points (P2) to (P7) must be defined in the area for multiple machining.

(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

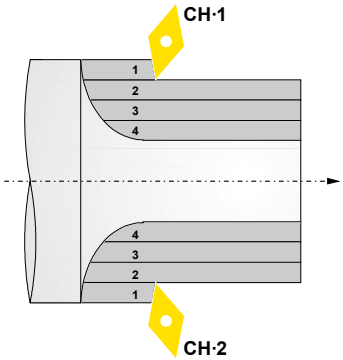
This feature can only be used on machines whose configuration allows it, e.g. TT lathes. The machine must have at least two channels and two turrets. If this feature is used with more channels, each one must have access to a different turret.

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 180º 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 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 programed.

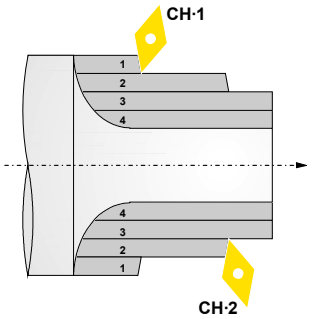
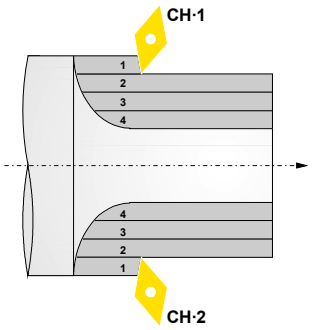
#DINDIST ON [{type},{channels}<,N{turns}><,D{distance}>]

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 

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
```

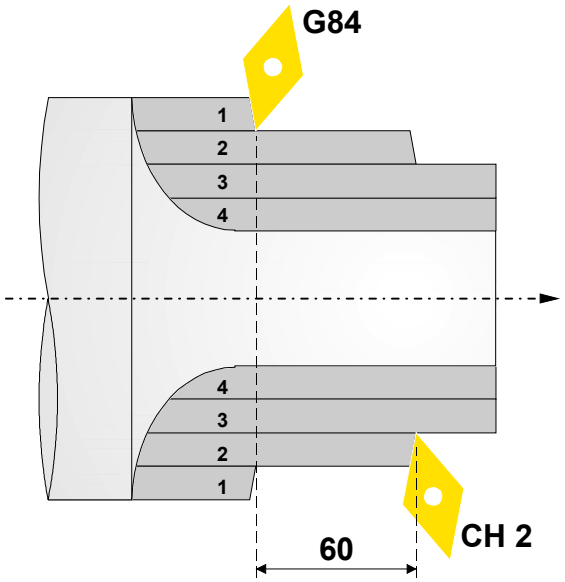

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

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
```



10.1.2 Synchronized identical passes

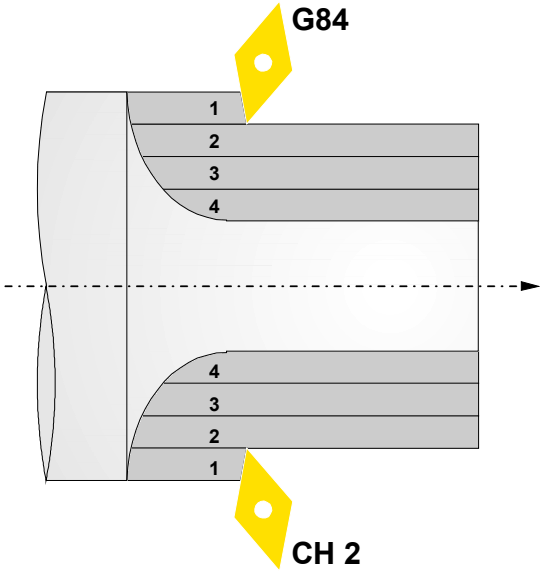
In this type of machining, the master channel executes the canned cycle and the CNC synchronizes the axes of the plane of the slave axes with the axes of the plane of the master channel so all the channels are executed in the same pass. The CNC only synchronizes the passes of the roughing operation; 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
```



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.

The available thread types are:

M (S.I.)	Metric thread with normal pitch (international system).
M (S.I.F.)	Metric thread with fine pitch (international system).
B.S.W. (W)	Whitworth thread with normal pitch.
B.S.F.	Whitworth thread with fine pitch.
U.N.C.	Unified American thread with normal pitch.
U.N.F.	Unified American thread with fine pitch.
B.S.P.	Whitworth gas thread.
A.P.I.	Threads for pipes of the oil sector.

11.1 Regular pitch metric thread — M (S.I.)

Diameter		Pitch		Depth (mm)	
(mm)	(inches)	(mm)	(inches)	Internal	External
0,3000	0,0118	0,0750	0,0030	0,0406	0,0460
0,4000	0,0157	0,1000	0,0039	0,0541	0,0613
0,5000	0,0197	0,1250	0,0049	0,0677	0,0767
0,6000	0,0236	0,1500	0,0059	0,0812	0,0920
0,8000	0,0315	0,2000	0,0079	0,1083	0,1227
1,0000	0,0394	0,2500	0,0098	0,1353	0,1534
1,2000	0,0472	0,2500	0,0098	0,1353	0,1534
1,4000	0,0551	0,3000	0,0118	0,1624	0,1840
1,6000	0,0630	0,3500	0,0138	0,1895	0,2147
1,7000	0,0669	0,3500	0,0138	0,1895	0,2147
1,8000	0,0709	0,3500	0,0138	0,1895	0,2147
2,0000	0,0787	0,4000	0,0157	0,2165	0,2454
2,2000	0,0866	0,4500	0,0177	0,2436	0,2760
2,3000	0,0906	0,4000	0,0157	0,2165	0,2454
2,5000	0,0984	0,4500	0,0177	0,2436	0,2760
2,6000	0,1024	0,4500	0,0177	0,2436	0,2760
3,0000	0,1181	0,5000	0,0197	0,2707	0,3067
3,5000	0,1378	0,6000	0,0236	0,3248	0,3680
4,0000	0,1575	0,7000	0,0276	0,3789	0,4294
4,5000	0,1772	0,7500	0,0295	0,4060	0,4601
5,0000	0,1969	0,8000	0,0315	0,4330	0,4907
5,5000	0,2165	0,9000	0,0354	0,4872	0,5521
6,0000	0,2362	1,0000	0,0394	0,5413	0,6134
7,0000	0,2756	1,0000	0,0394	0,5413	0,6134
8,0000	0,3150	1,2500	0,0492	0,6766	0,7668
9,0000	0,3543	1,2500	0,0492	0,6766	0,7668
10,0000	0,3937	1,5000	0,0591	0,8120	0,9201
11,0000	0,4331	1,5000	0,0591	0,8120	0,9201
12,0000	0,4724	1,7500	0,0689	0,9473	1,0735
14,0000	0,5512	2,0000	0,0787	1,0826	1,2268
16,0000	0,6299	2,0000	0,0787	1,0826	1,2268
18,0000	0,7087	2,5000	0,0984	1,3533	1,5335
20,0000	0,7874	2,5000	0,0984	1,3533	1,5335
22,0000	0,8661	2,5000	0,0984	1,3533	1,5335
24,0000	0,9449	3,0000	0,1181	1,6239	1,8402
27,0000	1,0630	3,0000	0,1181	1,6239	1,8402
30,0000	1,1811	3,5000	0,1378	1,8946	2,1469
33,0000	1,2992	3,5000	0,1378	1,8946	2,1469
36,0000	1,4173	4,0000	0,1575	2,1652	2,4536
39,0000	1,5354	4,0000	0,1575	2,1652	2,4536
42,0000	1,6535	4,5000	0,1772	2,4359	2,7603
45,0000	1,7717	4,5000	0,1772	2,4359	2,7603
48,0000	1,8898	5,0000	0,1969	2,7065	3,0670
52,0000	2,0472	5,0000	0,1969	2,7065	3,0670
56,0000	2,2047	5,5000	0,2165	2,9772	3,3737
60,0000	2,3622	5,5000	0,2165	2,9772	3,3737
64,0000	2,5197	6,0000	0,2362	3,2478	3,6804
68,0000	2,6772	6,0000	0,2362	3,2478	3,6804
72,0000	2,8346	6,0000	0,2362	3,2478	3,6804
76,0000	2,9921	6,0000	0,2362	3,2478	3,6804
80,0000	3,1496	6,0000	0,2362	3,2478	3,6804

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

The CNC calculates the depth with the formulae:

- Depth in internal threads = 0.5413 x Pitch
- Depth in external threads = 0.6134 x Pitch

Diameter		Pitch		Depth (mm)	
(mm)	(inches)	(mm)	(inches)	Internal	External
1,0000	0,0394	0,2000	0,0079	0,1083	0,1227
1,2000	0,0472	0,2000	0,0079	0,1083	0,1227
1,4000	0,0551	0,2000	0,0079	0,1083	0,1227
1,7000	0,0669	0,2000	0,0079	0,1083	0,1227
2,0000	0,0787	0,2500	0,0098	0,1353	0,1534
2,3000	0,0906	0,2500	0,0098	0,1353	0,1534
2,5000	0,0984	0,3500	0,0138	0,1895	0,2147
2,6000	0,1024	0,3500	0,0138	0,1895	0,2147
3,0000	0,1181	0,3500	0,0138	0,1895	0,2147
3,5000	0,1378	0,3500	0,0138	0,1895	0,2147
4,0000	0,1575	0,5000	0,0197	0,2707	0,3067
4,5000	0,1772	0,5000	0,0197	0,2707	0,3067
5,0000	0,1969	0,5000	0,0197	0,2707	0,3067
6,0000	0,2362	0,7500	0,0295	0,4060	0,4601
7,0000	0,2756	0,7500	0,0295	0,4060	0,4601
8,0000	0,3150	1,0000	0,0394	0,5413	0,6134
9,0000	0,3543	1,0000	0,0394	0,5413	0,6134
10,0000	0,3937	1,0000	0,0394	0,5413	0,6134
12,0000	0,4724	1,2500	0,0492	0,6766	0,7668
13,0000	0,5118	1,5000	0,0591	0,8120	0,9201
14,0000	0,5512	1,5000	0,0591	0,8120	0,9201
16,0000	0,6299	1,5000	0,0591	0,8120	0,9201
18,0000	0,7087	1,5000	0,0591	0,8120	0,9201
20,0000	0,7874	1,5000	0,0591	0,8120	0,9201
22,0000	0,8661	1,5000	0,0591	0,8120	0,9201
24,0000	0,9449	2,0000	0,0787	1,0826	1,2268
27,0000	1,0630	2,0000	0,0787	1,0826	1,2268
30,0000	1,1811	2,0000	0,0787	1,0826	1,2268
33,0000	1,2992	2,0000	0,0787	1,0826	1,2268
36,0000	1,4173	3,0000	0,1181	1,6239	1,8402
39,0000	1,5354	3,0000	0,1181	1,6239	1,8402
42,0000	1,6535	3,0000	0,1181	1,6239	1,8402
45,0000	1,7717	3,0000	0,1181	1,6239	1,8402
48,0000	1,8898	3,0000	0,1181	1,6239	1,8402
52,0000	2,0472	3,0000	0,1181	1,6239	1,8402
56,0000	2,2047	4,0000	0,1575	2,1652	2,4536
60,0000	2,3622	4,0000	0,1575	2,1652	2,4536
64,0000	2,5197	4,0000	0,1575	2,1652	2,4536
68,0000	2,6772	4,0000	0,1575	2,1652	2,4536
72,0000	2,8346	4,0000	0,1575	2,1652	2,4536
76,0000	2,9921	4,0000	0,1575	2,1652	2,4536
80,0000	3,1496	4,0000	0,1575	2,1652	2,4536

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			Depth (mm)	
	(mm)	(inches)	Edges	(mm)	(inches)	Internal	External
1/16	1,5875	0,0625	60	0,4233	0,0167	0,2710	0,2710
3/32	2,3812	0,0937	48	0,5292	0,0208	0,3388	0,3388
1/8	3,1750	0,1250	40	0,6350	0,0250	0,4066	0,4066
5/32	3,9687	0,1562	32	0,7938	0,0313	0,5083	0,5083
3/16	4,7625	0,1875	24	1,0583	0,0417	0,6776	0,6776
7/32	5,5562	0,2187	24	1,0583	0,0417	0,6776	0,6776
1/4	6,3500	0,2500	20	1,2700	0,0500	0,8132	0,8132
5/16	7,9375	0,3125	18	1,4111	0,0556	0,9035	0,9035
3/8	9,5250	0,3750	16	1,5875	0,0625	1,0165	1,0165
7/16	11,1125	0,4375	14	1,8143	0,0714	1,1617	1,1617
1/2	12,7000	0,5000	12	2,1167	0,0833	1,3553	1,3553
9/16	14,2875	0,5625	12	2,1167	0,0833	1,3553	1,3553
5/8	15,8750	0,6250	11	2,3091	0,0909	1,4785	1,4785
3/4	19,0500	0,7500	10	2,5400	0,1000	1,6264	1,6264
7/8	22,2250	0,8750	9	2,8222	0,1111	1,8071	1,8071
1	25,4000	1,0000	8	3,1750	0,1250	2,0330	2,0330
1 1/8	28,5750	1,1250	7	3,6286	0,1429	2,3234	2,3234
1 1/4	31,7500	1,2500	7	3,6286	0,1429	2,3234	2,3234
1 3/8	34,9250	1,3750	6	4,2333	0,1667	2,7106	2,7106
1 1/2	38,1000	1,5000	6	4,2333	0,1667	2,7106	2,7106
1 5/8	41,2750	1,6250	5	5,0800	0,2000	3,2527	3,2527
1 3/4	44,4500	1,7500	5	5,0800	0,2000	3,2527	3,2527
1 7/8	47,6250	1,8750	5	5,6444	0,2222	3,6141	3,6141
2	50,8000	2,0000	5	5,6444	0,2222	3,6141	3,6141
2 1/8	53,9750	2,1250	5	5,6444	0,2222	3,6141	3,6141
2 1/4	57,1500	2,2500	4	6,3500	0,2500	4,0659	4,0659
2 3/8	60,3250	2,3750	4	6,3500	0,2500	4,0659	4,0659
2 1/2	63,5000	2,5000	4	6,3500	0,2500	4,0659	4,0659
2 5/8	66,6750	2,6250	4	6,3500	0,2500	4,0659	4,0659
2 3/4	69,8500	2,7500	4	7,2571	0,2857	4,6467	4,6467
2 7/8	73,0250	2,8750	4	7,2571	0,2857	4,6467	4,6467
3	76,2000	3,0000	4	7,2571	0,2857	4,6467	4,6467
3 1/4	82,5500	3,2500	3	7,8154	0,3077	5,0042	5,0042
3 1/2	88,9000	3,5000	3	7,8154	0,3077	5,0042	5,0042
3 3/4	95,2500	3,7500	3	8,4667	0,3333	5,4212	5,4212
4	101,6000	4,0000	3	8,4667	0,3333	5,4212	5,4212
4 1/4	107,9500	4,2500	3	8,8348	0,3478	5,6569	5,6569
4 1/2	114,3000	4,5000	3	8,8348	0,3478	5,6569	5,6569
4 3/4	120,6500	4,7500	3	9,2364	0,3636	5,9141	5,9141
5	127,0000	5,0000	3	9,2364	0,3636	5,9141	5,9141

11.4 Fin pitch Whitworth thread — BSF

The threads must be defined in millimeters or inches. For example, to define a Whitworth thread with a 3/16 pitch, enter 4.7625 mm or 0.1875 inches.

The CNC calculates the pitch and the depth with the formulae:

- Pitch in millimeters = 25.4 / number of edges
- Pitch in inches = 1 / number of edges
- Depth in internal threads = 0,6403 x Pitch
- Depth in external threads = 0,6403 x Pitch

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

11.5 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			Depth (mm)	
	(mm)	(inches)	Edges	(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

11.6 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			Depth (mm)	
	(mm)	(inches)	Edges	(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

11.7 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			Depth (mm)	
	(mm)	(inches)	Edges	(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

11.8 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.

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

USER CYCLES.

12

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.

12.

USER CYCLES.

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

User notes:

User notes:

User notes:

FAGOR

FAGOR AUTOMATION

Fagor Automation S. Coop.

Bº San Andrés, 19 - Apdo. 144
E-20500 Arrasate-Mondragón, Spain
Tel: +34 943 039 800
Fax: +34 943 791 712
E-mail: info@fagorautomation.es
www.fagorautomation.com

AENOR
ER
Empresa Registrada
ER-0731/1994

CERTIFIED
IQNet
MANAGEMENT SYSTEM