

CNC

8037 ·TC·

Operating manual

Ref. 1310
Soft: V01.4x



FAGOR AUTOMATION



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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.

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ABOUT THE PRODUCT

BASIC CHARACTERISTICS

BASIC CHARACTERISTICS	
Monitor	7.5" Color LCD
Block processing time	7 ms
Look-ahead	75 blocks
RAM memory	1 Mb
Flash memory	128 MB
PLC cycle time	3 ms / 1000 instructions
Minimum position loop	4 ms
USB	Standard
RS-232 serial line	Standard
DNC (via RS232)	Standard
Ethernet	Option
5 V or 24 V probe inputs	2
Local digital inputs and outputs.	16 I / 8 O 40 I / 24 O 56 I / 32 O
Feedback inputs for the axes and spindle	4 TTL / 1Vpp inputs
Feedback inputs for handwheels	2 TTL inputs
Analog outputs	4 for axes and spindle
CAN servo drive system for Fagor servo drive connection.	Option
Remote CAN modules, for digital I/O expansion (RIO).	Option



Before start-up, verify that the machine that integrates this CNC meets the 89/392/CEE Directive.

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SOFTWARE OPTIONS

	Model		
	M	T	TC
Number of axes	3	2	2
Number of spindles	1	1	1
Electronic threading	Standard	Standard	Standard
Tool magazine management:	Standard	Standard	Standard
Machining canned cycles	Standard	Standard	Standard
Multiple machining	Standard	-----	-----
Rigid tapping	Standard	Standard	Standard
DNC	Standard	Standard	Standard
Tool radius compensation	Standard	Standard	Standard
Retracing	Standard	-----	-----
Jerk control	Standard	Standard	Standard
Feed forward	Standard	Standard	Standard
Oscilloscope function (Setup assistance)	Standard	Standard	Standard
Roundness test (Setup assistance)	Standard	Standard	Standard

About the product



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DECLARATION OF CONFORMITY

The manufacturer:

Fagor Automation S. Coop.

Barrio de San Andrés Nº 19, C.P. 20500, Mondragón -Guipúzcoa- (SPAIN).

Declares:

Under their responsibility that the product:

8037 CNC

Consisting of the following modules and accessories:

8037-M, 8037-T, 8037-TC

Remote modules RIO

ETHERNET, ETHERNET-CAN-CAN AXES, ETHERNET-CAN AXES

Note. Some additional characters may follow the references mentioned above. They all comply with the directives listed. However, check that that's the case by checking the label of the unit itself.

Referred to by this declaration with following directives:

Low voltage regulations.

EN 60204-1: 2006 Electrical equipment on machines — Part 1. General requirements.

Regulation on electromagnetic compatibility.

EN 61131-2: 2007 PLC — Part 2. Requirements and equipment tests.

As instructed by the European Community Directives 2006/95/EEC on Low Voltage and 2004/108/EC on Electromagnetic Compatibility and its updates.

In Mondragón, March 14th, 2012

Fagor Automation, S. Coop.


Director Gerente
Pedro Ruiz de Aguirre



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VERSION HISTORY

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

The version history uses the following abbreviations:

INST	Installation manual
PRG	Programming manual
OPT	Operating manual
OPT-TC	Operating manual for the TC option.

Software V01.42

March 2012

First version.



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Version history



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SAFETY CONDITIONS

Read the following safety measures in order to prevent harming people or damage to this product and those products connected to it.

This unit may only be repaired by authorized personnel at Fagor Automation.

Fagor Automation shall not be held responsible of any physical damage or defective unit resulting from not complying with these basic safety regulations.

PRECAUTIONS AGAINST PERSONAL DAMAGE

- Interconnection of modules.

Use the connection cables provided with the unit.

- Use proper Mains AC power cables

To avoid risks, use only the Mains AC cables recommended for this unit.

- Avoid electrical overloads.

In order to avoid electrical discharges and fire hazards, do not apply electrical voltage outside the range selected on the rear panel of the central unit.

- Ground connection.

In order to avoid electrical discharges, connect the ground terminals of all the modules to the main ground terminal. Before connecting the inputs and outputs of this unit, make sure that all the grounding connections are properly made.

- Before powering the unit up, make sure that it is connected to ground.

In order to avoid electrical discharges, make sure that all the grounding connections are properly made.

- Do not work in humid environments.

In order to avoid electrical discharges, always work under 90% of relative humidity (non-condensing) and 45 °C (113° F).

- Do not work in explosive environments.

In order to avoid risks or damages, do no work in explosive environments.



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PRECAUTIONS AGAINST PRODUCT DAMAGE

- Working environment.

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

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

- 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 complies with the European directives on electromagnetic compatibility. Nevertheless, it is recommended to keep it away from sources of electromagnetic disturbance such as:

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

- Enclosures.

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

- Avoid disturbances coming from the machine tool.

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

- DC relay coils. Diode type 1N4000.
- AC relay coils. RC connected as close to the coils as possible with approximate values of $R=220 \Omega / 1 W$ y $C=0,2 \mu F / 600 V$.
- AC motors. RC connected between phases, with values of $R=300 \Omega / 6 W$ y $C=0,47 \mu F / 600 V$.

- Use the proper power supply.

Use an external regulated 24 Vdc power supply for the inputs and outputs.

- Grounding of the power supply.

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

- Analog inputs and outputs connection.

It is recommended to connect them using shielded cables and connecting their shields (mesh) to the corresponding pin.

- Ambient conditions.

The working temperature must be between +5 °C and +40 °C (41°F and 104°F)

The storage temperature must be between -25 °C and +70 °C. (-13°F and 158°F)

- Central unit enclosure (8037 CNC).

Make sure that the needed gap is kept between the central unit and each wall of the enclosure. Use a DC fan to improve enclosure ventilation.

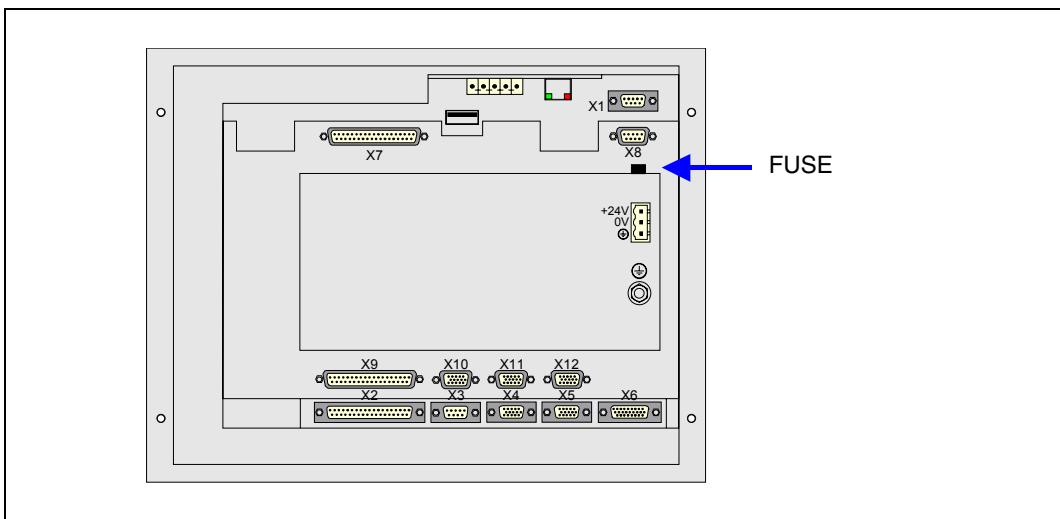
- Power switch.

This power switch must be mounted in such a way that it is easily accessed and at a distance between 0.7 meters (27.5 inches) and 1.7 meters (5.5ft) off the floor.

PROTECTIONS OF THE UNIT ITSELF (8037)

- Central Unit.

It has a 4 A 250V external fast fuse (F).



- Inputs-Outputs.

All the digital inputs and outputs have galvanic isolation via optocouplers between the CNC circuitry and the outside.

Safety conditions

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PRECAUTIONS DURING REPAIR



Do not get into the inside of the unit. Only personnel authorized by Fagor Automation may manipulate the inside of this unit.

Do not handle the connectors with the unit connected to main AC power. Before manipulating the connectors (inputs/outputs, feedback, etc.) make sure that the unit is not connected to AC power.

SAFETY SYMBOLS

- Symbols which may appear on the manual.



Symbol for danger or prohibition.

It indicates actions or operations that may cause damage to people or to units.



Warning symbol.

It indicates situations that may be caused by certain operations and the actions to be taken to prevent them.



Obligation symbol.

It indicates actions and operations that must be carried out.



Information symbol.

It indicates notes, warnings and advises.

WARRANTY TERMS

INITIAL WARRANTY

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

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

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

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

FAGOR has exclusive competence in deciding whether the repair enters within the term defined as the warranty period.

EXCLUDING CLAUSES

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

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

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



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WARRANTY ON REPAIRS

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

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

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

For sold parts the warranty is 12 months length.

MAINTENANCE CONTRACTS

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



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MATERIAL RETURNING TERMS

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

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



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Material returning terms



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ADDITIONAL REMARKS

Mount the CNC away from coolants, chemical products, blows, etc. which could damage it. Before turning the unit on, verify that the ground connections have been properly made.

In case of a malfunction or failure, disconnect it and call the technical service. Do not get into the inside of the unit.



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Additional remarks



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FAGOR DOCUMENTATION

OEM manual

It is directed to the machine builder or person in charge of installing and starting-up the CNC.

USER-M manual

Directed to the end user.

It describes how to operate and program in M mode.

USER-T manual

Directed to the end user.

It describes how to operate and program in T mode.

TC Manual

Directed to the end user.

It describes how to operate and program in TC mode.

It contains a self-teaching manual.



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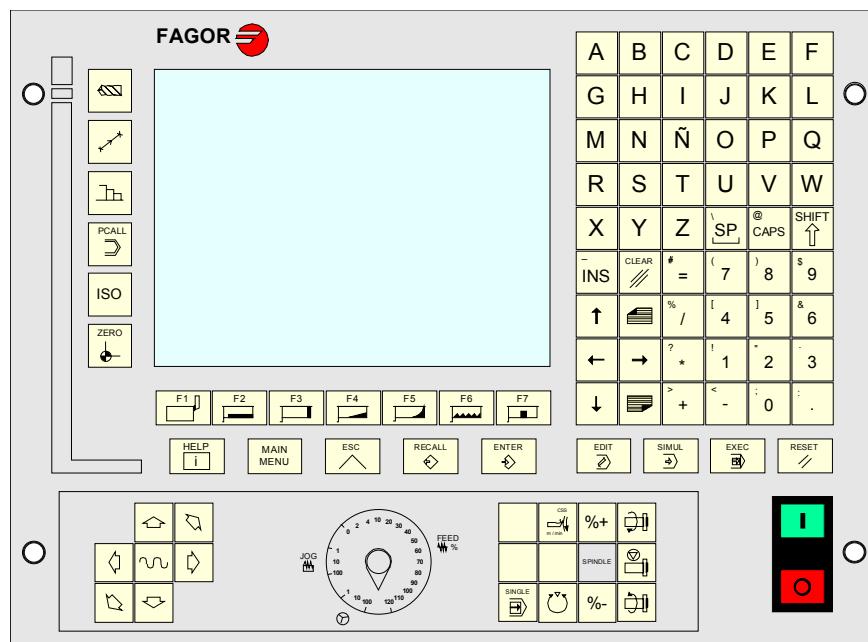


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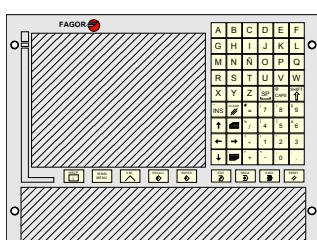
GENERAL CONCEPTS

1

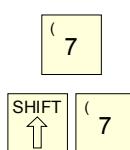
1.1 Keyboard



Alphanumeric keyboard and command keys



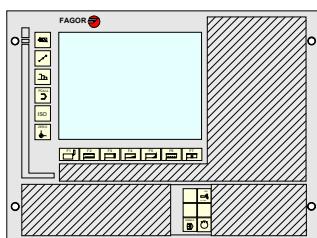
Select the 7 character.



Select the (character.



Specific keys of the TC model



These keys may be used for:

- Selecting and defining the machining operations.
- Selecting the spindle work mode.
- Selecting the single block or automatic execution Mode.

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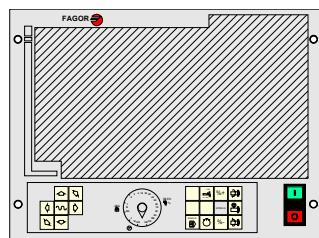
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JOG keys

1.

GENERAL CONCEPTS
Keyboard



These keys may be used for:

- Moving the axes of the machine.
- Governing the spindle.
- Modifying the feedrate of the axes and the spindle speed.
- Starting and stopping the execution.

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.TC. OPTION
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1.2 General concepts

It offers all the features of the T model plus those specific of the TC mode. For example, the CNC setup must be done in T mode.

In TC work mode, programs P900000 through P999999 are reserved for the CNC itself; in other words, the user cannot use them as part-programs.

On the other hand, in order to work in TC mode, the CNC must have programs P999997 and P999998 stored in its memory. Both programs are related to the software version and, consequently, are not supplied by Fagor Automation. Whenever the CNC detects a new software version, it updates these programs automatically and, for safety, it makes a copy of the old ones in the KeyCF.

Likewise, subroutines 0000 through 8999 are free to use and subroutines 9000 through 9999 are reserved for the CNC.



Programs P999997 and P999998 are associated with the software version. Fagor Automation shall not be held responsible of the CNC's performance if programs P999997 and P999998 have been deleted from memory or do not match the software version.

1.

Subroutines reserved for the CNC

Some of the subroutines reserved for the CNC have the following meaning:

9998 Subroutine that the CNC will execute at the beginning of each part-program.

9999 Subroutine that the CNC will execute at the end of each part-program.

Every time a new part-program is edited, the CNC inserts a call to the relevant subroutine at the beginning and at the end of the program.

Both subroutines must be defined by the machine manufacturer, even when no operation is to be carried out at the beginning and at the end of the part-program. If they are not defined, the CNC will issue an error message when trying to execute a part-program.

Example of how to define subroutine 9998.

```
( SUB 9998) ; Definition of subroutine 9998.  
... ; Program blocks defined by the OEM.  
(RET) ; End of subroutine.
```

OEM (manufacturer's) parameters

OEM parameters and subroutines with OEM parameters can only be used in OEM programs; those defined with the [O] attribute. Modifying one of these parameters in the tables requires an OEM password.

When using OEM parameters in the configuration programs, this program must have the [O] attribute; otherwise, the CNC will issue an error when editing the user cycles that refer to OEM parameters in write mode.



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.TC- OPTION
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Programs reserved for the CNC

Some of the programs reserved for the CNC have the following meaning:

P999998

It is a program of subroutines that the CNC uses to interpret the programs edited in TC format and execute them later on.



This program must not be modified. If this program is modified or deleted, Fagor Automation will not be held responsible of the CNC's performance.

If the manufacturer needs to create his own subroutines (for home search, tool change, etc.), as well as subroutines 9998 and 9999, they must be included in another program, for example P899999.

P999997

It is a text program that contains:

- The sentences and texts that will be displayed on the various screens of the TC mode.
- The help texts for the icons, in the work cycles, that are shown on the lower left side of the screen.
- The messages (MSG) and errors (ERR) that may come up at the TC model.

All the texts, messages and errors that may be translated into the desired language.



When modifying program 999997, it is recommended to make a backup copy of it because the CNC replaces that program when selecting another language or updating the software version.

Considerations about the texts

The format of a line is as follows:

;Text number - explanatory comment (not displayed) - \$Text to be displayed

All the program lines must begin with the ";" character and the text to be displayed must be preceded by the "\$" symbol. If a line begins with ";;", the CNC assumes that the whole line is a program comment.

Examples:

;44 \$M/MIN	Is message 44 and displays the text "M/MIN"
;;General text	The CNC treats it as a comment
;;44 Feedrate \$M/MIN	The CNC treats it as a comment
;44 Feedrate \$M/MIN	Is message 44 whose hidden explanatory comment is "Feedrate" and displays the text "M/MIN"

Considerations about the messages

The format must be respected. Only the text after SAVEMSG may be translated:

Example:

Original message:	N9500(MSG"SAVEMSG: TURNING CYCLE")
Translated message:	N9500(MSG"SAVEMSG: ZILINDRAKETA ZIKLOA")

Considerations about the errors

The format must be respected. Only the text between quote marks ("text") may be translated.

Example:

Original text:	N9000(ERROR"Cycle without roughing")
Translated text:	N9000(ERROR"Arbastatu gabeko zikloa")

P998000 ... P998999

They are the profiles defined by the user with the profile editor. In the TC mode, the user defines them with 3 digits (from 0 to 999) and the CNC saves them internally as P998xxx.

1.2.1 P999997 text program management

On power up, the CNC copies the texts of program P999997 into the system memory.

- It checks if program P999997 is in user memory, if not, it looks in the KeyCF and if it is not there either, it assumes the default ones and copies them into program P999997 of the user memory.
- When selecting mainland Chinese, it ignores program P999997 and it always assumes the default ones.

If when switching from T mode to TC mode, it cannot find program P999997 because it has been deleted, it is initialized like on power-up.

When modifying the texts of program P999997, turn the CNC off and back on so it assumes the new texts.

The CNC carries out the following operations when changing the language, the software version and when adding TC conversational modes (new software features):

- It copies, for safety, the texts that were being used into KeyCF as program P999993.
- It deletes the program P999997 that may be in the KeyCF.
- It assumes the new texts that are provided by default and copies them into program P999997 of the user memory.

To change the texts, after modifying program P999997, turn the CNC off and back on so it assumes the new texts.

1.

GENERAL CONCEPTS

General concepts



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1.

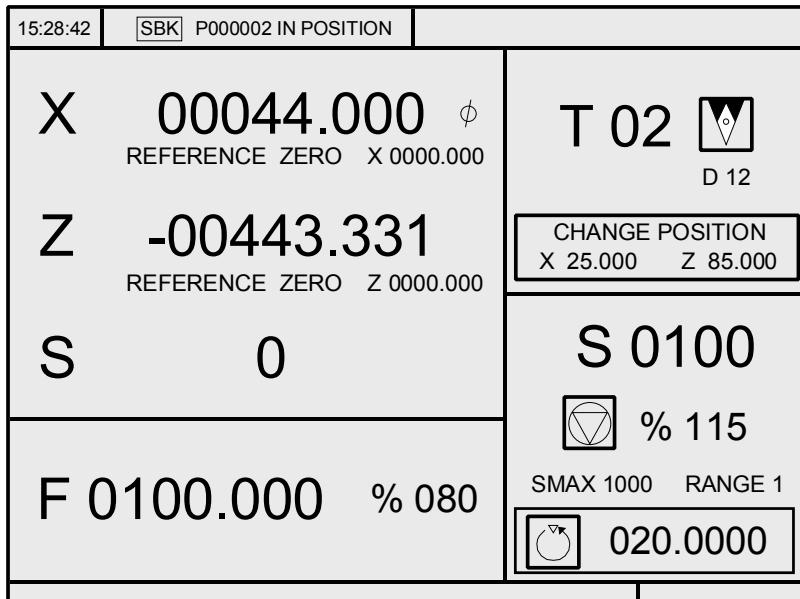
GENERAL CONCEPTS

Power-up

1.3 Power-up

On power-up and after the keystroke sequence [SHIFT] [RESET], the CNC shows "page 0" defined by the manufacturer; if there is no "page 0", it shows the standard screen of the work mode. Press any key to access the work mode.

The standard screen of the TC mode is the following:



There are two work modes; TC mode and T mode. Press the key sequence [SHIFT] [ESC] to toggle from one mode to the other.



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



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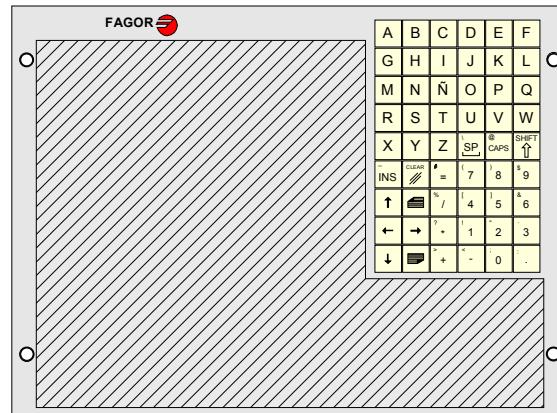
1.4 Operating in T mode



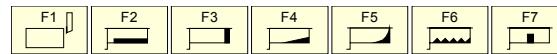
There are two work modes; TC mode and T mode. Press the key sequence [SHIFT] [ESC] to toggle from one mode to the other.

The keyboard is designed for also working in T mode. In T mode, use the alphanumeric keyboard and softkeys F1 through F7.

Alphanumeric keyboard:



Softkeys F1 to F7 are:



1.

GENERAL CONCEPTS
Operating in T mode



CNC 8037

.TC- OPTION
SOFT: V01.4x

1.5 Video off



The keystroke sequence [SHIFT] [CLEAR] clears the CRT screen (it goes blank). Press any key to restore the image.

Also, any message (PLC, program, etc.) restores the CNC image.

1.

GENERAL CONCEPTS
Video off

FAGOR 

CNC 8037

.TC. OPTION
SOFT: V01.4x

1.6 Managing the CYCLE START key

In order to avoid undesired executions when pressing key sequences that are not supported in TC mode, the CNC changes the "Start" icon at the top of the window from green to gray and shows a message indicating that it is an invalid action.

For example, if "M3 Start" is pressed (sequence not supported in TC mode) while a part-program is selected, the CNC issues a warning and prevents the selected part-program from running when detecting the "Start" key.

1.

GENERAL CONCEPTS
Managing the CYCLE START key



CNC 8037

·TC· OPTION
SOFT: V01.4x

1.

GENERAL CONCEPTS

Managing the CYCLE START key



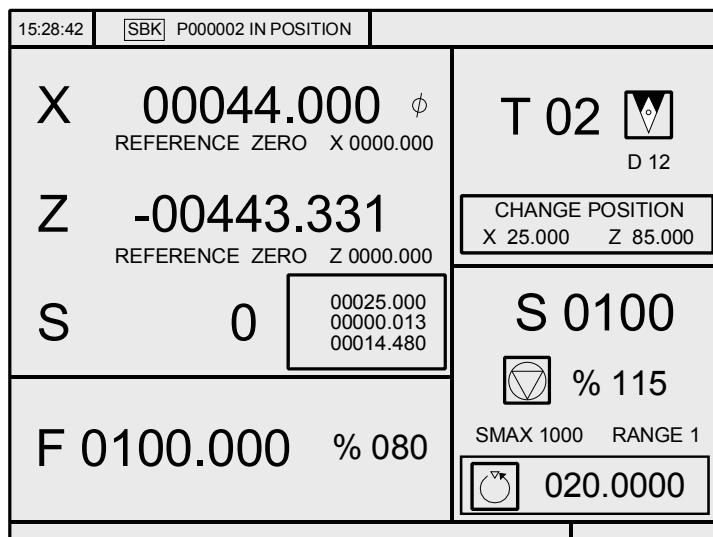
CNC 8037

.TC. OPTION
SOFT: V01.4x

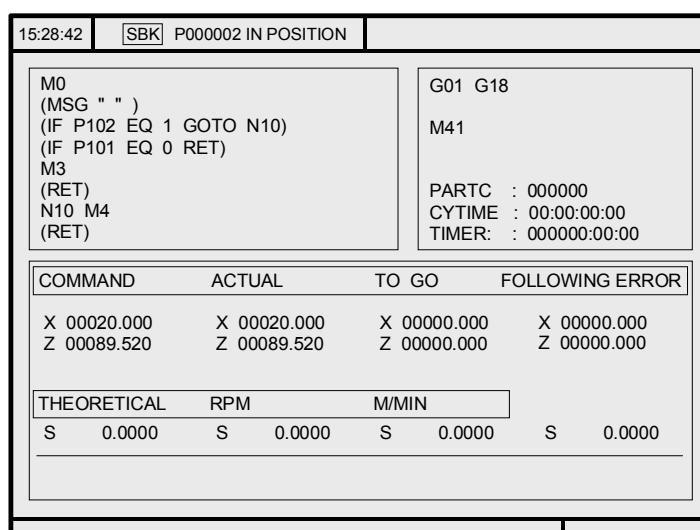
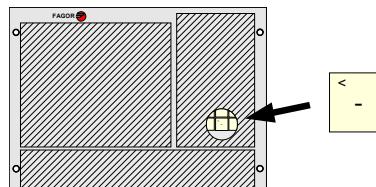
OPERATING IN JOG MODE

2

The standard screen of the TC mode is the following:



When pressing the [-] key, the CNC shows the special screen of the TC mode.



CNC 8037

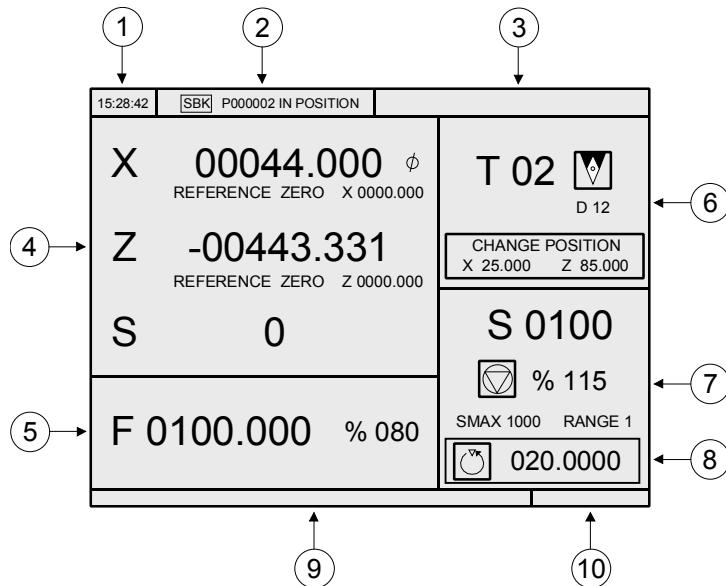
.TC- OPTION
SOFT: V01.4x

2.

2.1 Introduction

2.1.1 Standard screen of the TC mode

The standard screen of the TC mode offers the following data:



1. Clock.

2. This window may show the following data:

SBK When "Single Block" execution mode is selected.

DNC When the DNC mode is active.

P..... Number of the program currently selected.

Message "In position" - "Execution" - "Interrupted" - "RESET".

PLC messages.

3. This window shows the CNC messages.

4. This window may show the following data:

X, Z coordinates of the axes. The Ø symbol indicates that the axis is working in diameter.

In small characters, the axis coordinates referred to machine reference zero. These values are useful when letting the user define a tool change point (see zone 6) The CNC shows this data when text 33 of program 999997 has not been defined.

The real spindle rpm (S symbol).

5. The information shown in this window depends on the position of the switch.

In all cases, it shows the axis feedrate "F" currently selected and the % of F being applied.

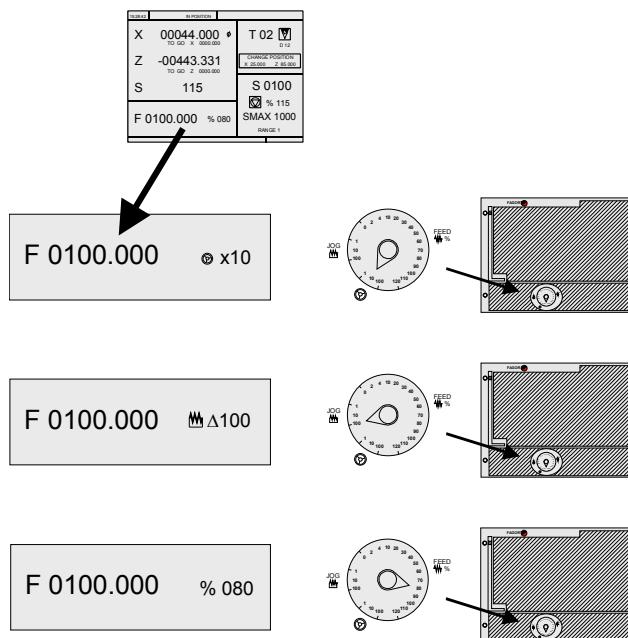
When feed-hold is active, the color of the feedrate value changes.



CNC 8037

.TC. OPTION
SOFT: V01.4x

Here are all the possible cases.



6. This window shows, in large characters, the selected tool number "T" and, in small characters, the "D" offset associated with the tool. If the tool number and the offset number are the same, the CNC will not show the "D" value. The window also shows a drawing of the location code (shape) associated with the tool.

This window also shows the coordinates of the tool change point referred to machine reference zero. The CNC does not show this window when text 47 of program 999997 has not been defined.

7. This window shows everything related to the spindle:

The theoretical turning speed that is selected; "S" value when constant turning speed and "CSS" value when working at constant surface speed.

The spindle status. It is represented with an icon and may be turning clockwise, counterclockwise or stopped.

The % of spindle speed being applied.

The maximum spindle rpm.

The active spindle speed gear (range). The CNC does not show this data when text 28 of program 999997 has not been defined.

8. Spindle angular increment when working in spindle orientation mode.

9. When accessing a work cycle, this window shows the help text associated with the selected icon.

That help text must be defined in program P999997 and edited in the desired language. See chapter "[1 General concepts](#)".

10. Reserved.

Displaying the active PLC messages

At the screen, press [+] of the alphanumeric keyboard, the CNC shows a window with all the active PLC messages. Besides, this window is also displayed whenever there is a program in execution.

The [**↑**] [**↓**] [PAGE UP] [PAGE DOWN] keys are used to move around the messages. The [ESC] key is used to close the window.

The window is only displayed when there are more than one active message.

Direct access to the oscilloscope

The oscilloscope may be accessed from the standard screen by pressing "7" and then "1" as long as no data is being written into any field.

2.

OPERATING IN JOG MODE
Introduction

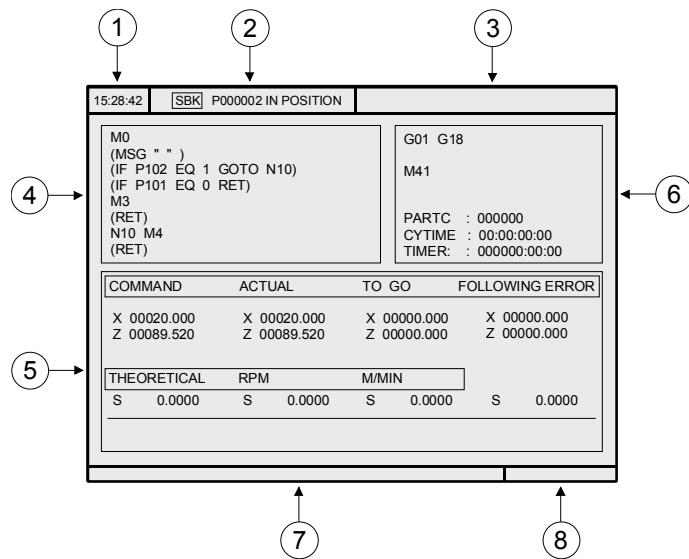
FAGOR

CNC 8037

.TC- OPTION
SOFT: V01.4x

2.1.2 Description of the special screen of the TC mode

The special screen of the TC mode offers the following data:



1. Clock.

2. This window may show the following data:

SBK When "Single Block" execution mode is selected.

DNC When the DNC mode is active.

P..... Number of the program currently selected.

Message "In position" - "Execution" - "Interrupted" - "RESET".

PLC messages.

3. This window shows the CNC messages.

4. This window shows the lines of the program currently selected.

5. The X and Z axes have the following fields:

COMMAND	It indicates the programmed coordinate or position that the axis must reach.
---------	--

ACTUAL	It indicates the actual (current) position of the axis.
--------	---

TO GO	It indicates the distance which is left to run to the programmed coordinate.
-------	--

FOLLOWING ERROR	Difference between the theoretical value and the real value of the position.
-----------------	--

The spindle (S) has the following fields:

THEORETICAL	Programmed theoretical S speed.
-------------	---------------------------------

RPM	Speed in rpm.
-----	---------------

M/MIN	Speed in meters per minute.
-------	-----------------------------

FOLLOWING ERROR	When working with spindle orientation (M19), it indicates the difference between the theoretical and the real speeds.
-----------------	---

2.

6. This window shows the status of the "G" functions and the auxiliary "M" functions that are active. Likewise, it shows the value of the variables.

PARTC It indicates the number of consecutive parts executed with the same part-program.

Every time a new program is selected, this variable is reset to "0".

CYTIME It indicates the time elapsed while executing the part. It is given in "hours : minutes: seconds: hundredths of a second".

Every time a part-program execution starts, even when repetitive, this variable is reset to "0".

TIMER It indicates the count of the timer enabled by PLC. It is given in "hours : minutes: seconds".

7. Reserved.

8. Reserved.

Displaying the active PLC messages

At the screen, press [+] of the alphanumeric keyboard, the CNC shows a window with all the active PLC messages. Besides, this window is also displayed whenever there is a program in execution.

The [\uparrow] [\downarrow] [PAGE UP] [PAGE DOWN] keys are used to move around the messages. The [ESC] key is used to close the window.

The window is only displayed when there are more than one active message.

Direct access to the oscilloscope

The oscilloscope may be accessed from the auxiliary screen by pressing "7" and then "1" as long as no data is being written into any field.

2

OPERATING IN JOG MODE

Introduction

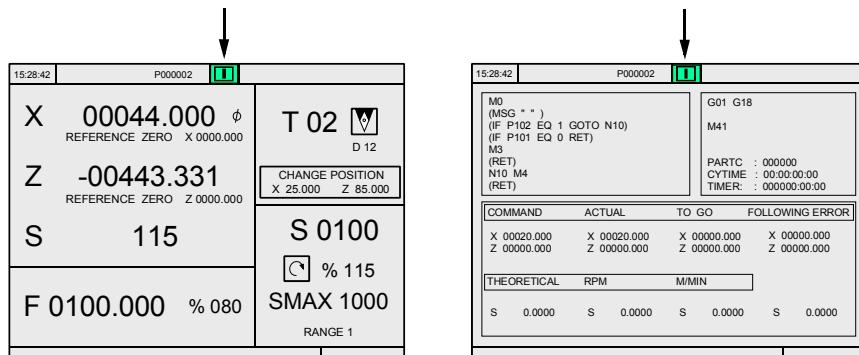


CNC 8037

.TC- OPTION
SOFT: V01.4x

2.1.3 Selecting a program for simulation or execution

When selecting a part-program or operation saved as part of a part-program for simulation or execution, the CNC selects that part-program and shows it highlighted next to the green "start" symbol in the top center window.



When the top center window shows the part-program selected next to the green "start" symbol, the CNC acts as follows:

- If [START] is pressed, the CNC executes the part-program that is selected.
- If [CLEAR] is pressed, the CNC de-selects the part-program and removes it from the top center window.

2.

OPERATING IN JOG MODE
Introduction

FAGOR

CNC 8037

.TC. OPTION
SOFT: V01.4x

2.2 Axis control

2.2.1 Work units

When accessing the TC mode, the CNC assumes the work units «mm or inches», «mm/min. or mm/rev», «radius or diameter» etc. selected by machine parameter.

To modify those values, access the T mode and change the corresponding machine parameter.

2.

OPERATING IN JOG MODE
Axis control



CNC 8037

·TC· OPTION
SOFT: V01.4x

2.2.2 Coordinate preset

The coordinates must be preset on one axis at a time proceeding as follows:

1. Press the key of the desired axis, [X] or [Z].
The CNC will highlight the coordinate of that axis indicating that it is selected.
2. Key in the value to preset the axis.
To quit the preset mode, press [ESC].
3. Press [ENTER] for the CNC to assume that value as the new value for the point.
The CNC requests confirmation of the command. Press the [ENTER] to confirm it or [ESC] to quit the preset mode.

2.

OPERATING IN JOG MODE
Axis control

FAGOR 

CNC 8037

.TC. OPTION
SOFT: V01.4x

2.2.3 Managing the axis feedrate (F)

To set a particular axis feedrate value, proceed as follows:

1. Press the [F] key.

The CNC will highlight the current value that it is selected.

2. Key in the desired new feedrate value.

To quit the preset selection mode, press [ESC].

3. Press [START] for the CNC to assume that value as the new value for axis feedrate.

2.

OPERATING IN JOG MODE
Axis control



CNC 8037

.TC- OPTION
SOFT: V01.4x

2.

OPERATING IN JOG MODE
Home search

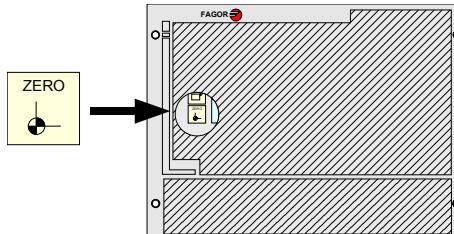
2.3 Home search

Home search may be done in 2 ways:

- Homing all the axes.
- Homing a single axis.

Homing all the axes

To home all the axes, press [ZERO].



The CNC requests confirmation of the command (text 48 of program 999997). Press [START], the CNC will execute the home search subroutine defined by the OEM in general machine parameters P34 (REFPSUB).

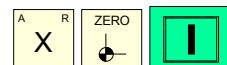


After searching home this way, the CNC will maintain the part zero or zero offset active at the time. A home search subroutine (general machine parameter P34 other than 0) must be defined when using this method. Otherwise, the CNC will display the corresponding error.

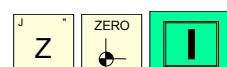
Homing a single axis

To home a single axis, press the key of the desired axis and the key for home search.

In either case, the CNC requests confirmation of the command (text 48 of program 999997).



It homes the X axis.



It homes the Z axis.



After searching home this way, the CNC will not maintain the part zero or zero offset active at the time and assumes the machine reference zero as the new part zero.



CNC 8037

.TC. OPTION
SOFT: V01.4x

2.4 Zero offset table

It is possible to manage the zero offset table (G54G59) from the conversational mode. G159N20). This table contains the same values as that of the conversational mode.

Press the [ZERO] key to access the zero offset table as well as to get out of it. The zero offset table may be accessed in the following ways.

- From the standard screen, as long as no axis is selected. The CNC will request confirmation of the command.
- From ISO mode, when the "zero offsets and presets" cycle is selected.

The zero offset table looks like this. It shows all the offsets, PLC offset included, and their value in each axis.

	X	Z
PLC	0.0000	0.0000
G54	0.0000	0.0000
G55	0.0000	0.0000
G56	0.0000	0.0000
G57	0.0000	0.0000
ΔG58	0.0000	0.0000
ΔG59	0.0000	0.0000

When scrolling the focus through the table, the elements appear in different colors as follows.

Color	Meaning
Green background. Text in white.	The real value of the table and the value shown on the screen are the same.
Red background. White text.	The real value of the table and the value shown on the screen are NOT the same. The value on the table has been changed, but it has not been validated. Press [ENTER] to validate the change.
Blue background.	The zero offset is active. There may be two active zero offsets at the same time, one absolute (G54 ... G57, G159N7 ... G159N20) and another one incremental (G58G59).

How to edit the table data

The following operations are possible in the zero offset table. Press [ENTER] to validate any changes.

- Editing a zero offset.

It is edited one axis at a time. Select a data with the focus and edit its value. If the focus is placed on a zero offset (G54 ... G59, G159N7 ... G159N20), the editing begins on the first axis of that zero offset.

- Load the active zero offset into the table.

Place the focus on the zero offset to be defined (G54 ... G59, G159N7 ... G159N20) and press the [RECALL] key. The active preset is saved in the selected zero offset.

If instead of placing the focus on a zero offset, it is placed on one of the axes, only that axis will be affected.

- Deleting a zero offset.

Place the focus on the zero offset to be deleted (G54 ... G59, G159N7 ... G159N20) and press the [CLEAR] key. All the axes of that zero offset are reset to zero.

If instead of placing the focus on a zero offset, it is placed on one of the axes, only that axis will be affected.

2

OPERATING IN JOG MODE
Zero offset table



CNC 8037

.TC- OPTION
SOFT: V01.4x

2.5 Jog movement

When making a move in manual, both in jog and with handwheels, the moving axis appears in reverse video.

- With gantry axes, only the master axis is highlighted.
- With a path handwheel, no axis is highlighted; but it is in path-jog.

2.

OPERATING IN JOG MODE
Jog movement



CNC 8037

.TC. OPTION
SOFT: V01.4x

2.5.1 Moving an axis to a particular position (coordinate)

The movements of axes to a particular coordinate are made one at a time as follows

[X] Target coordinate [START]

[Z] Target coordinate [START]

2.

OPERATING IN JOG MODE
Jog movement

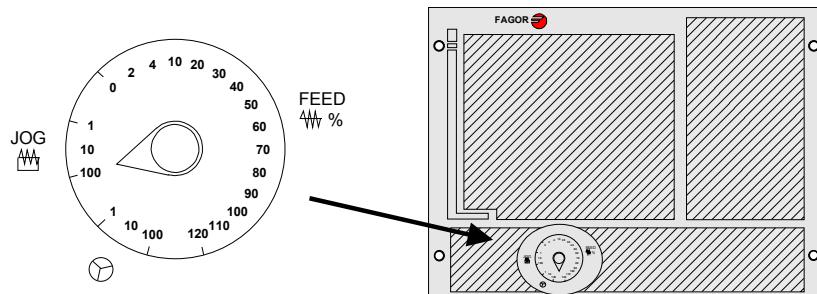


CNC 8037

.TC- OPTION
SOFT: V01.4x

2.5.2 Incremental movement

Turn the JOG switch to one of the JOG positions.



The incremental movement must be made one axis at a time. To do that, press the JOG keys for the direction of the axis to be jogged.

Every time a key is pressed, the corresponding axis moves the amount set by the switch. This movement is made at the selected feedrate (F).

Switch position	Distance
1	0.001 mm or 0.0001 inches
10	0.010 mm or 0.0010 inches
100	0.100 mm or 0.0100 inches

OPERATING IN JOG MODE
Jog movement

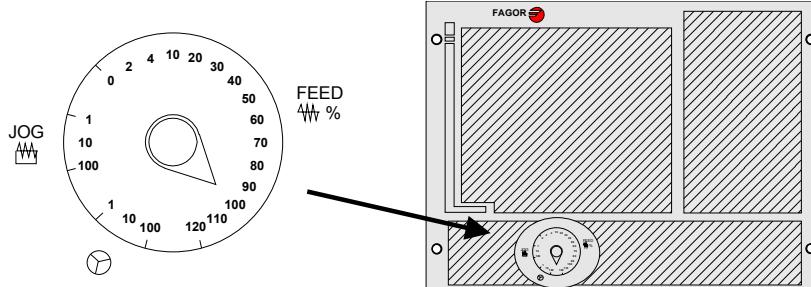
FAGOR

CNC 8037

.TC. OPTION
SOFT: V01.4x

2.5.3 Continuous movement

Place the movement selector in the continuous-jog position and select at the feedrate override switch (FEED) the percentage (0% to 120%) of the feedrate to be applied.



The continuous jog must be made one axis at a time. To do that, press the JOG keys for the direction of the axis to be jogged.

The axis moves at a feedrate equal to the selected percentage (0% to 120%) of feedrate "F".



If while jogging an axis, the rapid key is pressed, the axis will move at the maximum feedrate possible, set by axis machine parameter "G00FEED". This feedrate will be applied while that key is kept pressed and the previous feedrate will be restored when that key is released.

Depending on the status of the general logic input "LATCHMAN", the movement will be carried out as follows:

- If the PLC sets this mark low, the axis will be jogged while pressing the corresponding Jog key.
- If the PLC sets this mark high, the axes will start moving from the moment the JOG key is pressed until the same is pressed again, or another JOG key is pressed. In this case, the movement will be transferred to that indicated by the new key.

The following cases are possible when working with "F" in mm/rev:

- The spindle is running.
- The spindle is stopped, but a spindle speed S has been selected.
- The spindle is stopped and no spindle speed S has been selected.

2.

OPERATING IN JOG MODE

Jog movement

FAGOR

CNC 8037

.TC- OPTION
SOFT: V01.4x

The spindle is running:

The CNC moves the axes at the programmed F.

The spindle is stopped, but a spindle speed S has been selected:

S 0500

The CNC calculates the feedrate F in mm/min for the theoretical S and moves the axis.



% 115

For example if "F 2.000" and "S 500":

$$F \text{ (mm/min)} = F \text{ (mm/rev)} \times S \text{ (rpm)} = 2 \times 500 = 1000 \text{ mm/min.}$$

The axis moves at a feedrate of 1000 mm/min.

The spindle is stopped and no spindle speed S has been selected:

S 0500



% 115

If F = 0, the CNC moves the axes in rapid.

If F is other than 0, the axes can only be moved by pressing the rapid key and an axis key. The CNC moves the axis in rapid.

If the axis to be jogged does not belong to the active plane, the movement is carried out in mm/minute; thus, it is not necessary to program an S at the spindle.

Also, if any axis of the plane is the Y axis, it is not necessary either to program the S for jog movements in any axis, regardless of whether it belongs to the plane or not.

This is especially interesting for auxiliary axes, center rests and tailstocks, because, in those cases, the S has no effect.

2.5.4 Path-jog

The "path jog" mode acts when the switch is in one of the continuous or incremental jog positions. This feature may be used to act upon the jog keys of an axis to move both axes of the plane at the same time for chamfering (straight sections) and rounding (curved sections). The CNC assumes as "Path jog" the keys associated with the X axis.



This feature must be managed from the PLC. This feature is usually activated and deactivated by means of an external push-button or a key configured for that purpose, as well as the selection of the type of path.

2.

OPERATING IN JOG MODE
Jog movement

The next example uses the [O2] key to activate and deactivate the path-jog mode and the [O3] key to indicate the type of movement.

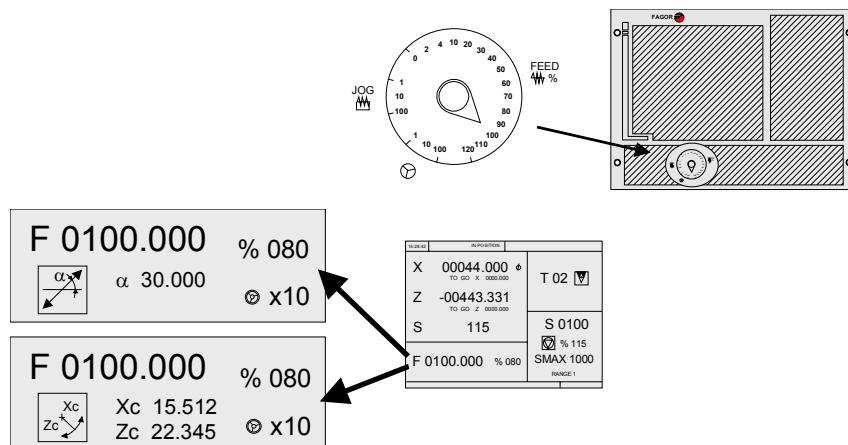
Activate / deactivate the path-jog mode.

DFU B29 R561 = CPL M5054

It selects the type of movement, straight section or arc section.

DFU B31 R561 = CPL M5053

While in jog mode and having selected path-jog, the CNC shows the following information:



For a linear movement (top figure), the path angle must be defined and for an arc (bottom figure), the center coordinates must be indicated. To define these variables, press the [F] key and then one of these keys: [\leftarrow] [\rightarrow] [\uparrow] [\downarrow].



CNC 8037

.TC- OPTION
SOFT: V01.4x

Operation in path-jog mode

The "path jog" mode is only available with the X axis keys. When pressing one of the keys associated with the X axis, the CNC behaves as follows:

Switch position	Path-jog	Type of movement
Continuous jog	OFF	Only the axis and in the indicated direction
	ON	Both axes in the indicated direction and along the indicated path
Incremental jog	OFF	Only the axis, the selected distance and in the indicated direction
	ON	Both axes, the selected distance and in the indicated direction, but along the indicated path
Handwheel		It ignores the keys.

The rest of the jog keys always work in the same way, whether "path jog" is on or off. The rest of the keys move only the axis and in the indicated direction.

The movements in path-jog may be aborted by pressing the [STOP] key or setting the jog switch to one of the handwheel positions.

Considerations about the jog movements

This mode assumes as axis feedrate the one selected in jog mode and it will also be affected by the feedrate override switch. If F0 is selected, it assumes the one indicated by machine parameter "JOGFEED (P43)". This mode ignores the rapid jog key.

Path-jog movements respect the travel limits and the work zones.

2.

OPERATING IN JOG MODE
Jog movement

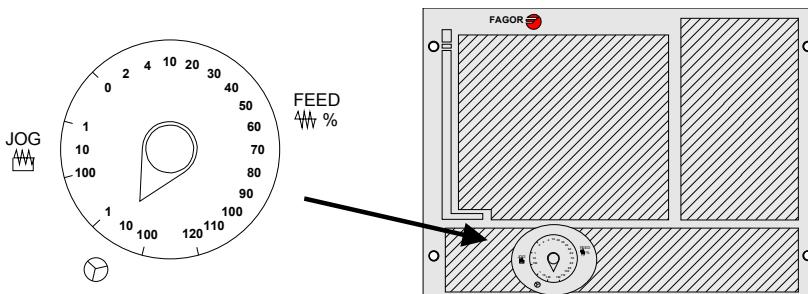


CNC 8037

.TC. OPTION
SOFT: V01.4X

2.5.5 Movement with an electronic handwheel

This option may be used to govern the movements of the machine using an electronic handwheel. To do that, turn the left switch to any of the handwheel positions.



The positions available are 1, 10 and 100; they indicate the multiplying factor being applied besides the internal x4 to the feedback pulses supplied by the electronic handwheel.

Switch position	Distance per turn
1	0.100 mm or 0.0100 inches
10	1.000 mm or 0.1000 inches
100	10.000 mm or 1.0000 inches



It may happen that depending on the turning speed and the selector switch position, the CNC be demanded a faster feedrate than the maximum allowed (axis machine parameter "G00FEED"). The CNC will move the axis the indicated distance but at the maximum feedrate allowed.

The machine has an electronic handwheel

Once the desired switch position has been selected, press one of the JOG keys for the axis to be jogged. The bottom of the screen shows the selected axis in small characters and next to the handwheel symbol.

When using a FAGOR handwheel with an axis selector button, the axis may be selected as follows:

- Push the button on the back of the handwheel. The CNC select the first axis and it highlights it.
- When pressing the button again, the CNC selects the next axis and so on in a rotary fashion.
- To deselect the axis, hold the button pressed for more than 2 seconds.

Once the axis has been selected, it will move as the handwheel is being turned and in the direction indicated by it.

The machine has two or three electronic handwheels

Each axis will move as the corresponding handwheel is being turned according to the switch position and in the direction indicated by it.

When the machine has a general handwheel and individual handwheels (associated with each axis of the machine), the individual handwheels have the highest priority; i.e. when moving an individual handwheel, the CNC will ignore the general handwheel.

2.

OPERATING IN JOG MODE

Jog movement

FAGOR

CNC 8037

-TC- OPTION
SOFT: V01.4x

2.5.6 Feed handwheel

Usually, when making a part for the first time, the machine feedrate is controlled by means of the feedrate override switch.

From this version on, it is also possible to use the machine handwheels to control that feedrate. This way, the machining feedrate will depend on how fast the handwheel is turned.



This feature must be managed from the PLC. This feature is usually activated and deactivated by means of an external push-button or a key configured for that purpose.

The following CNC variables return the number of pulses the handwheel has turned.

- | | |
|--------|--|
| HANPF | Shows the number of pulses of the 1st handwheel. |
| HANPS | Shows the number of pulses of the 2nd handwheel. |
| HANPT | Shows the number of pulses of the 3rd handwheel. |
| HANPFO | Shows the number of pulses of the 4th handwheel. |

2.5.7 Path-handwheel

The "path handwheel" mode acts when the switch is in one of the handwheel positions. With this feature, it is possible to jog two axes of the plane at the same time along a linear path (chamfer) or circular path (rounding) with a single handwheel. The CNC assumes as the path handwheel the general handwheel or, when this one is missing, the one associated with the X axis.



This feature must be managed from the PLC. This feature is usually activated and deactivated by means of an external push-button or a key configured for that purpose, as well as the selection of the type of path.

The next example uses the [O2] key to activate and deactivate the path-handwheel mode and the [O3] key to indicate the type of movement.

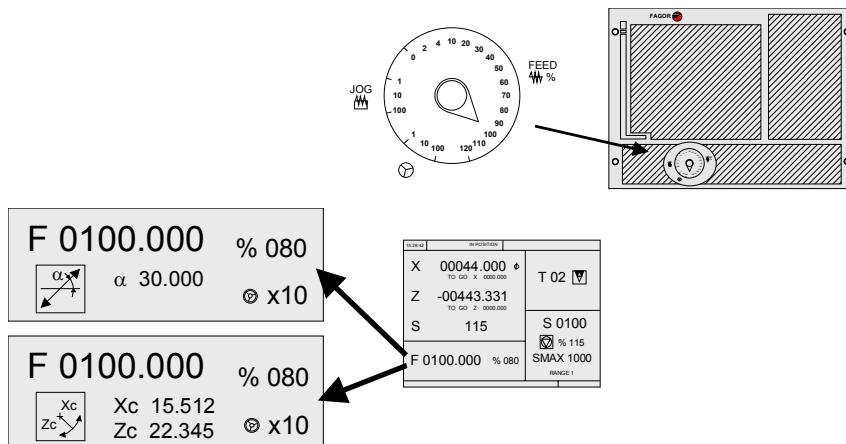
Activate / deactivate the path-handwheel mode.

DFU B29 R561 = CPL M5054

It selects the type of movement, straight section or arc section.

DFU B31 R561 = CPL M5053

While in handwheel mode and having selected path-handwheel, the CNC shows the following information:



For a linear movement (top figure), the path angle must be defined and for an arc (bottom figure), the center coordinates must be indicated. To define these variables, press the [F] key and then one of these keys: [\leftarrow] [\rightarrow] [\uparrow] [\downarrow].

Operation in path-handwheel mode

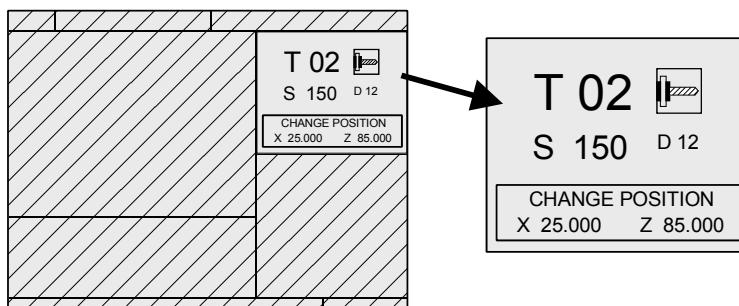
When selecting the path handwheel mode, the CNC behaves as follows.

- If there is a general handwheel, it will be the one working in path handwheel mode. The individual handwheels, if any, will remain associated with the corresponding axes.
- If there is no general handwheel, the individual handwheel associated with the X axis then works in path-handwheel mode.

The movements in path-handwheel may be aborted by pressing the [STOP] key or setting the jog switch to one of the continuous or incremental positions.

2.6 Tool control

The standard screen of the TC mode offers the following tool data.



This window displays the following information:

- In large characters, the tool "T" number currently selected and a graphic representation of its tip.
- The "D" offset number associated with the tool.
- The "S" rpm that are selected for the live tool. This value is only shown when a live tool has been selected.
- The position values of the tool change point. The CNC does not show this window when text 47 of program 999997 has not been defined.

To select another tool, follow these steps:

1. Press the [T] key.
The CNC highlights the tool number.
 2. Key in the number of the tool to be selected.
To quit the preset selection mode, press [ESC].
 3. Press [START] for the CNC to select the new tool.
The CNC will manage the tool change. Once the new tool has been selected, the CNC refreshes the graphic representation for the location code (shape) associated with the new tool.
- Another offset may be assigned to the tool temporarily without modifying the one associated with it.
1. To access the "D" field, press [T] and [$\uparrow\downarrow$].
 2. Key in the number of the desired tool offset and press [START].
The CNC temporarily assumes the new offset for the current tool. The internal table is not modified, the tool's associated offset is still the one assigned to it when it was calibrated.

2.

OPERATING IN JOG MODE
Tool control

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.TC. OPTION
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2.6.1 Tool change

Depending on the type of tool changer, the following options are possible:

- Machine with automatic tool changer.
- Machine with manual tool changer.

In either case, the CNC acts as follows:

- The CNC executes the subroutine associated with the tool change (general machine parameter P60 "TOOLSUB").
- The CNC sends to the PLC all the necessary information for it to manage the tool change.
- The CNC assumes the new tool values (offsets, geometry, etc.).

Example of how to manage a manual tool changer.

- Subroutine 55 is defined as the subroutine associated with the tools.
General machine parameter P60 "TOOLSUB" = 55.

The subroutine associated with the tools may contain the following information:

```
( SUB 55)
(P100 = NBTOOL)
; Assigns the requested tool number to P100.
(P101 = MS3)
; If spindle counterclockwise P102=1.
G0 G53... XP?? ZP??
; Movement to the tool change point.
M5
; Spindle stop.
(MSG "SELECT T?P100 AND PRESS START")
; Message to select the tool change.
M0
; Stop the program stop and wait for START to be pressed.
(MSG "" "")
; Deletes previous message.
(IF P102 EQ 1 GOTO N10)
; Restores the spindle turning direction.
(IF P101 EQ 0 RET)
M3
(RET)
N10 M4
(RET)
```

- The tool is selected after executing the subroutine.

General machine parameter P71 "TAFTERS" = YES.

- The movement to the change point only takes place when executing an operation or cycle of the TC mode.

When a cycle has been selected (CYCEXE other than 0)

The program is being executed (OPMODA bit 0 = 1).

- Once the subroutine is completed, the CNC executes function T??, sends to the PLC all the necessary information for it to manage the tool change and assumes the new tool values (offsets, geometry, etc.).

2.

OPERATING IN JOG MODE

Tool control



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2.6.2 Variable tool change point

If the manufacturer so wishes, he can let the user define the tool change point every time. Obviously, this feature depends on the type of machine and type of tool changer.

This feature may be used to change the tool next to the part, thus avoiding movements to a tool change point located far away from it.

To do this:

- Define the text 47 of program 999997 so the CNC requests the X, Z coordinates of the tool change point.

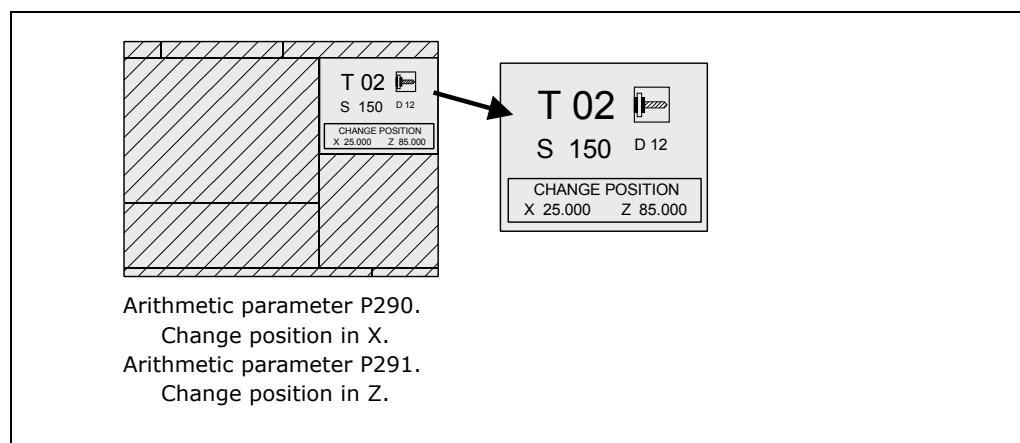
For example: ;47 \$CHANGE POSITION

These coordinates must be referred to machine zero point, so the zero offsets do not affect the tool change point. Therefore, the CNC can show, next to the X, Z coordinates and in small characters, the coordinates of the axes referred to machine reference zero.

- Text 33 of program 999997 must be defined so the CNC shows the coordinates of the axes referred to machine reference zero.

For example: ;33 \$MACHINE ZERO

Since the operator can change the tool change point at any time, the subroutine associated with the tools must consider those values. Arithmetic parameters P290 and P291 contain the values set by the operator as tool change position in X and Z respectively.



In subroutine 55 of the previous section, the line setting the movement to the tool change point must be modified:

Where it says:

G0 G53 XP??? ZP??? ; Movement to the tool change point.

It must say:

G0 G53 XP290 ZP291 ;User-defined movement to the change point.

Define the coordinates of the tool change point (X, Z)

1. Press the [T] key to select the «T» field.
2. Then press the [X], [Y] or [Z] key of the desired axis or the [\leftarrow] [\rightarrow] [\uparrow] [\downarrow] keys.
3. After placing the cursor on the coordinates of the axis to be defined, define the desired values.

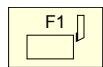
After placing the cursor on the coordinates of the axes to be defined, the value is entered in one of the following ways.

- Entering the value manually. Key in the desired value and press [ENTER].
- Assign the current machine position.

Jog the axis with the handwheel or the JOG keys up to the desired point. Press [RECALL] so the selected data assumes the value shown in the top right window and press [ENTER].

The top right window shows the tool position at all times.

2.7 Tool calibration



This mode may be used to define the tools and calibrate them without a probe.

This mode is also available while executing a program and during tool inspection.

What can be done in tool calibration mode

2

OPERATING IN JOG MODE
Tool calibration

The data that may be modified from the calibration cycles depend on when this mode is accessed. The following limitations must be borne in mind when accessing the tool calibration mode with a program in execution or from tool inspection.

Without a program in execution nor in tool inspection.

When editing the active tool, it is possible:

- Modify all the data.
- Change the active tool ($T ?? + [START]$).

When NOT editing the active tool, it is possible:

- Modify all the data except the part dimensions.
- Change the active tool ($T ?? + [START]$).

Program in execution or interrupted.

When editing the active tool, it is possible:

- To modify the I and K data.
- Select another tool ($T?? + [RECALL]$) and modify the I and K data.

When NOT editing the active tool, it is possible:

- To modify the I, K and D data.
- Select another tool ($T?? + [RECALL]$) and modify the I, K and D data.

Program in tool inspection.

When editing the active tool, it is possible:

- To modify the I and K data.
- Select another tool ($T?? + [RECALL]$) and modify the I and K data.
- Change the active tool ($T ?? + [START]$).

When NOT editing the active tool, it is possible:

- To modify the I, K and D data.
- Select another tool ($T?? + [RECALL]$) and modify the I, K and D data.
- Change the active tool ($T ?? + [START]$).



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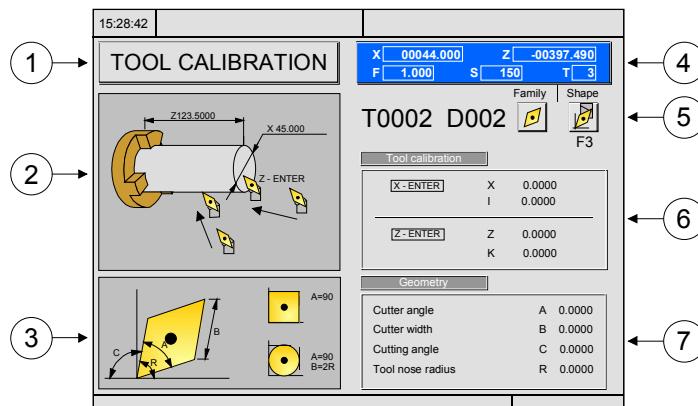
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2.

OPERATING IN JOG MODE
Tool calibration

2.7.1 Define the tool in the tool table

When accessing this level, the CNC shows the following screen.



1. Indicating the selected work mode: "Tool calibration".
 2. Help graphics for calibrating the tool.
 3. Help graphics for defining tool geometry.
 4. Current machine status.
 5. Tool number, tool offset number, location code (shape) and tool family.
 6. Length values defined for this tool.
 7. Values for the geometry of the tool.
- Real X Z coordinates, real axis feedrate F, real spindle speed S and currently selected tool T.

Define the tool data

Proceed as follows to define a tool in the tool table:

Select the number of the tool to be defined.

1. Press the [T] key to select the "T" field.
2. Key in the desired tool number and press [RECALL].

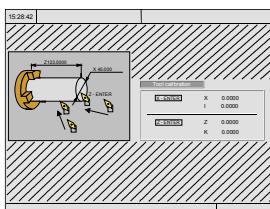
If the tool is defined, the CNC will show the values stored in the table. If the tool is not defined, the CNC assigns an offset with the same number to it and all the data is reset to 0.

Select the number of the offset tool to be associated with this tool.

1. The "D" field must be selected. If it is not, use the [**↓**] key.
2. Key in the desired offset number to be associated with the tool and press [ENTER].

Define the tool dimensions.

The data for the tool is the following.

	<table border="0"> <tr> <td>X</td><td>Tool's X dimension (in radius).</td></tr> <tr> <td>Z</td><td>Tool's Z dimension.</td></tr> <tr> <td>I</td><td>X wear offset (in diameter).</td></tr> <tr> <td>K</td><td>Z wear offset.</td></tr> </table>	X	Tool's X dimension (in radius).	Z	Tool's Z dimension.	I	X wear offset (in diameter).	K	Z wear offset.
X	Tool's X dimension (in radius).								
Z	Tool's Z dimension.								
I	X wear offset (in diameter).								
K	Z wear offset.								

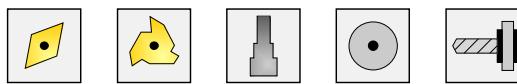
Even if the tool dimensions are known, it is recommended to measure it. See "[2.7.2 Manual tool calibration](#)" on page 61.

Once it is measured, the CNC updates the X, Z fields and sets the I and K data to 0.

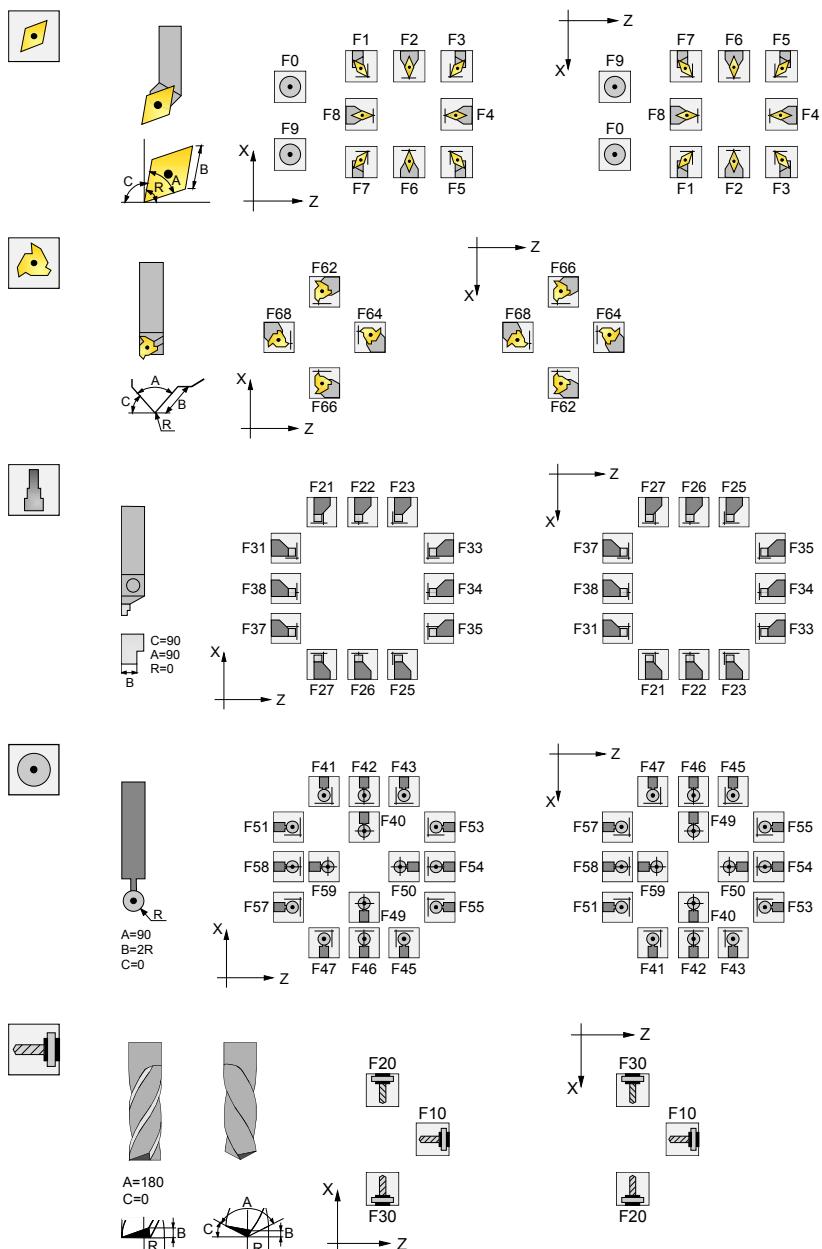
To define these values, select the corresponding field with the [**←**][**→**][**↑**][**↓**] keys, key in the desired value and press [ENTER].

Defining the tool type.

Place the cursor over the icon for tool type and press the [-] key. The available tool types are:

**Define the location code of the tool.**

Place the cursor over the icon for tool type and press the [-] key. The available tool types are:



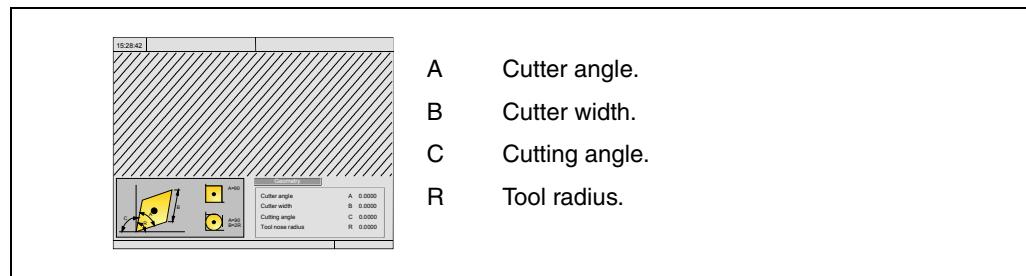
2

OPERATING IN JOG MODE

Tool calibration

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Define the rest of the data related to the tool.



The right window shows the tool geometry values and the left window shows help graphics. To define one of these values, select the corresponding field, key in the desired value and press [ENTER].

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Tool calibration

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2.7.2 Manual tool calibration

Before measuring the tool, it must be defined in the tool table. See "[2.7.1 Define the tool in the tool table](#)" on page 58.

There are 2 ways to calibrate a tool.

- When having a tool setting table.

Use the window that shows the tool dimensions to define that data. Define the X Z dimensions and the I, K wear.

- When not having any measuring device.

The measurements will be taken with the CNC. Use the window for tool calibration.



Tool calibration using a master part of known dimensions.

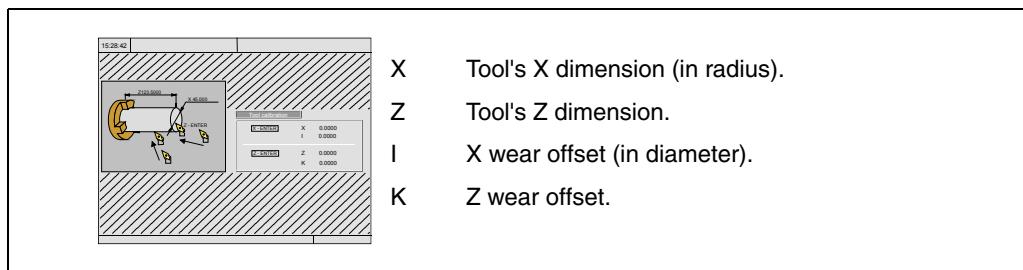
2

OPERATING IN JOG MODE

Tool calibration

Define the tool length or modify the length offsets

This window shows the dimensions assigned to the selected tool.



The X and Z data indicate the tool dimensions. I and K indicate the offset the CNC must apply to compensate for tool wear.

The CNC adds the value of the "I" offset to X length and the value of the "K" offset to the Z length to calculate the real dimensions ($R+I$, $L+K$) that must be used.

- Every time the X length or the Z length value is defined, the CNC sets the "I" and "K" fields to 0 respectively.
- The "I" and "K" data are accumulative. In other words, if the "I" has a value of 0.20 and the value of 0.05 is entered, the CNC assigns the value of 0.25 (0.20+0.05) to the "I" field.
- If one sets $I=0$ or $K=0$, they are both reset to 0.

To change one of these values, select the corresponding field, key in the desired value and press [ENTER].

Tool calibration

Place a part of known dimensions in the spindle and define its dimensions in the left window.

To measure the tool, the tool must be selected on the machine. If it is not, press the [T] key, key in the desired number of the tool to be calibrated and press [START].

Measuring the tool.

1. Approach the tool to the part, touch it with it along the X axis and press [X] + [ENTER].
2. Approach the tool to the part, touch it with it along the Z axis and press [Z] + [ENTER].

The tool has been calibrated. The CNC updates the X, Z data and sets the I and K to 0. The actual tool length is ($X+I$) and ($Z+K$); the "I" must be given in diameter.



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Modifying the tool data while executing a program

It is possible to modifying the tool values (dimensions and geometry) without interrupting the execution of a program.



To do that, press the tool calibration key. The CNC will show the tool calibration screen with all the data for the active tool and it will allow modifying its data or of any other tool.

To exit this screen, press [ESC].

2.

OPERATING IN JOGG MODE
Tool calibration

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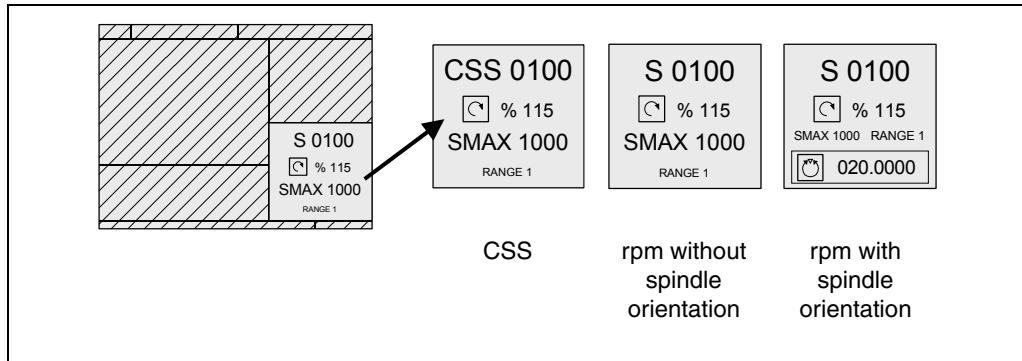
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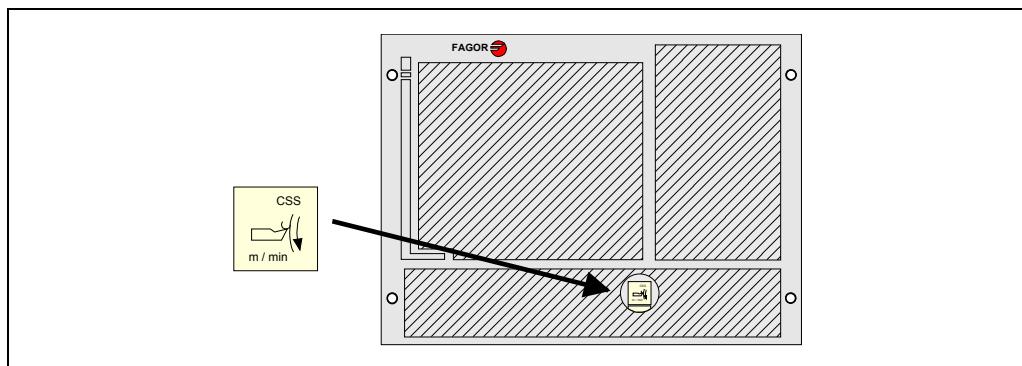
2.8 Spindle control

The standard screen of the TC mode has a window that shows the following spindle related data.

Since it is possible to work with the spindle in rpm, at CSS or in orientation mode, the information shown by that window will be different in each case.



To switch from one mode to the other, press the key:



On CNC power-up and after the keystroke sequence [SHIFT] [RESET], the CNC selects the rpm mode. When working at constant surface speed (CSS), the light of the [CSS] key is on.

2.

OPERATING IN JOG MODE
Spindle control

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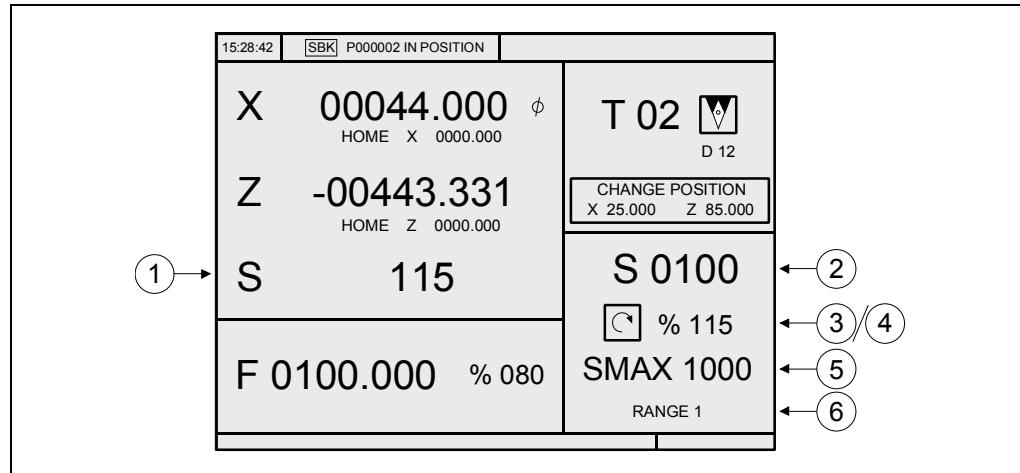
.TC- OPTION
SOFT: V01.4x

2.

OPERATING IN JOG MODE
Spindle control

2.8.1 Spindle in rpm

The CNC displays the following information.

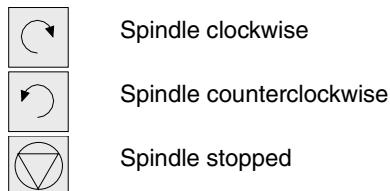


1. Real spindle speed in rpm.
2. Theoretical spindle speed in rpm.

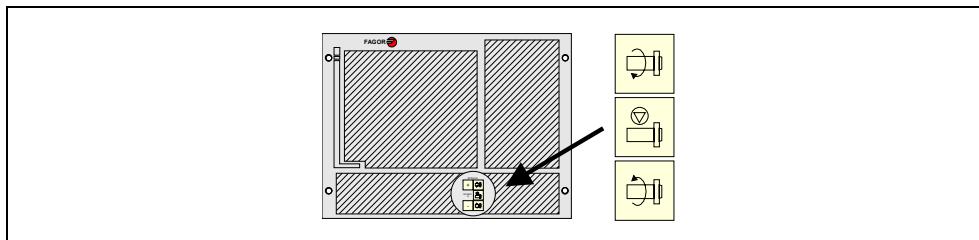
To select another speed, press the [S] key. The CNC highlights the current value.

Enter the new value and press [START]. The CNC assumes that value and refreshes the real spindle speed.

3. Spindle status.

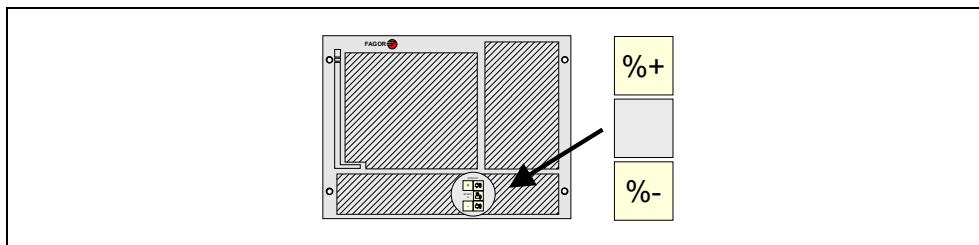


To modify the spindle status, press the following keys:



4. Percentage of the theoretical spindle speed being applied.

To modify the percentage (%), press the following keys.



5. Maximum spindle speed in rpm.

To select another speed, press the [S] key twice. The CNC highlights the current value. Enter the new value and press [ENTER]. The CNC assumes this value and does not allow the spindle to exceed these rpm.

The maximum spindle speed is saved in the MDISL variable. This variable is updated (refreshed) changing the SMAX value and when programming function "G92 S" via ISO.

6. Spindle gear currently selected.

This value cannot be changed when using an automatic gear change.

When not using an automatic tool changer, press the [S] key and then use the [**▼**] key to highlight the current value. Enter the gear number to be selected and press [ENTER] or [START].



When the machine does not use spindle gears, this message makes no sense. That is why the CNC does not show this message when text 28 of program 999997 has not been defined.

2.

OPERATING IN JOG MODE
Spindle control

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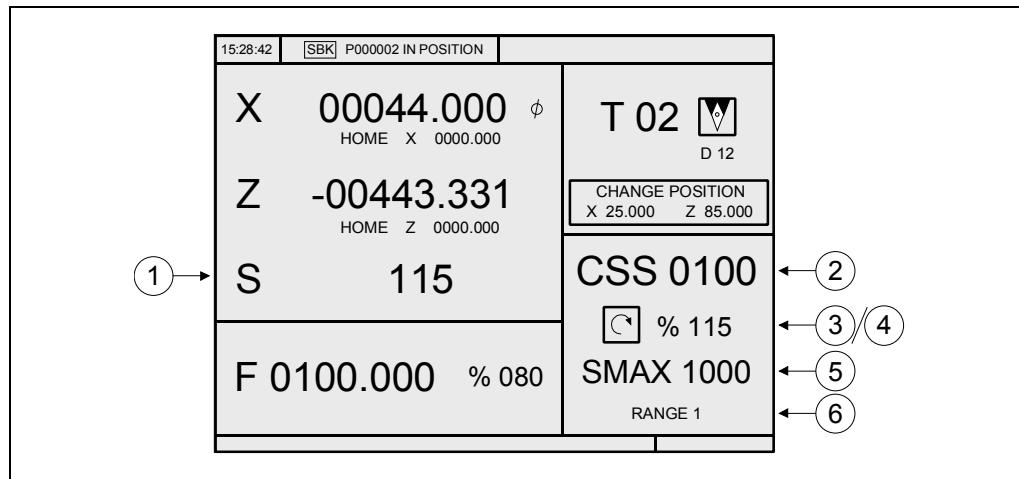
2.

OPERATING IN JOG MODE
Spindle control

2.8.2 Spindle in constant surface speed mode

In constant surface speed mode, the user sets the tangential speed that must always be kept between the tool tip and the part. Therefore, the spindle rpm depend on the position of the tool tip with respect to the rotation axis. Thus, if the tool tip gets away from the rotation axis, the spindle slows down and if it gets closer, it speeds up.

The CNC displays the following information.



1. Real spindle speed in rpm.

2. Theoretical constant surface speed. This speed is defined in meters/min or feet/min.

To select another speed, press the [S] key. The CNC highlights the current value.

Enter the new value and press [START]. The CNC assumes this value and if the spindle is turning, it updates the real spindle speed.

3. Spindle status.

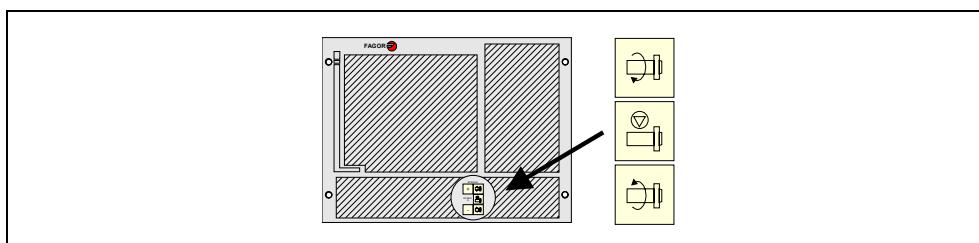


Spindle clockwise

Spindle counterclockwise

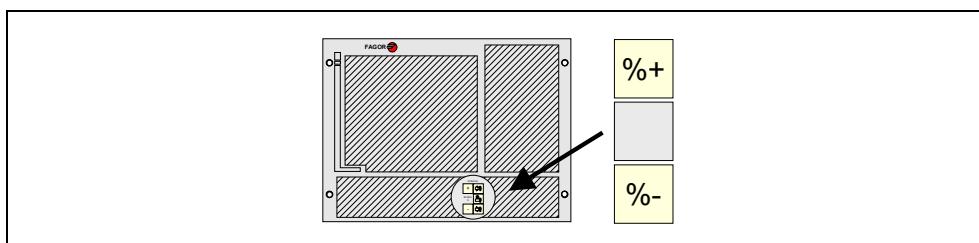
Spindle stopped

To modify the spindle status, press the following keys:



4. Percentage of the theoretical spindle speed being applied.

To modify the percentage (%), press the following keys.



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5. Maximum spindle speed in rpm.

To select another speed, press the [S] key twice. The CNC highlights the current value. Enter the new value and press [ENTER]. The CNC assumes this value and does not allow the spindle to exceed these rpm.

The maximum spindle speed is saved in the MDISL variable. This variable is updated (refreshed) changing the SMAX value and when programming function "G92 S" via ISO.

6. Spindle gear currently selected.

This value cannot be changed when using an automatic gear change.

When not using an automatic tool changer, press the [S] key and then use the [\downarrow] key to highlight the current value. Enter the gear number to be selected and press [ENTER] or [START].



When the machine does not use spindle gears, this message makes no sense. That is why the CNC does not show this message when text 28 of program 999997 has not been defined.

2

OPERATING IN JOG MODE

Spindle control

Working at constant surface speed mode

When selecting the constant surface speed mode, the CNC assumes the spindle gear (range) that is currently selected. The following cases may occur in this mode, when selecting a new constant surface speed value:

- The spindle is stopped.

The CNC selects the new speed, but it does not apply it until the starts turning.

- The spindle is running.

The CNC, depending on the position of the axis, calculates and turns the spindle at the corresponding rpm so the constant surface speed is the one defined.

The following cases may occur when moving the axes when working at constant surface speed:

- The spindle is running.

The CNC moves the axes at the programmed F.

As the X axis moves, the CNC adapts the spindle speed (rpm) to maintain the selected constant speed. Thus, if the tool tip gets away from the rotation axis, the spindle slows down and if it gets closer, it speeds up.

The CNC limits the spindle rpm to the maximum speed set by "SMAX".

- The spindle is stopped, but a spindle speed S has been selected.

The CNC calculates the feedrate F in mm/min for the last programmed S and moves the axis.

For example if «F 2.000» and «S 500»:

$$\text{Feedrate} = F \text{ (mm/rev)} \times S \text{ (rev/min)} = 2 \times 500 = 1000 \text{ mm/min.}$$

The axis moves at a feedrate of 1000 mm/min.

- The spindle is stopped and no spindle speed S has been selected.

If F = 0, the CNC moves the axes in rapid.



If F is other than 0, the axes can only be moved by pressing the rapid key and an axis key. The CNC moves the axis in rapid.

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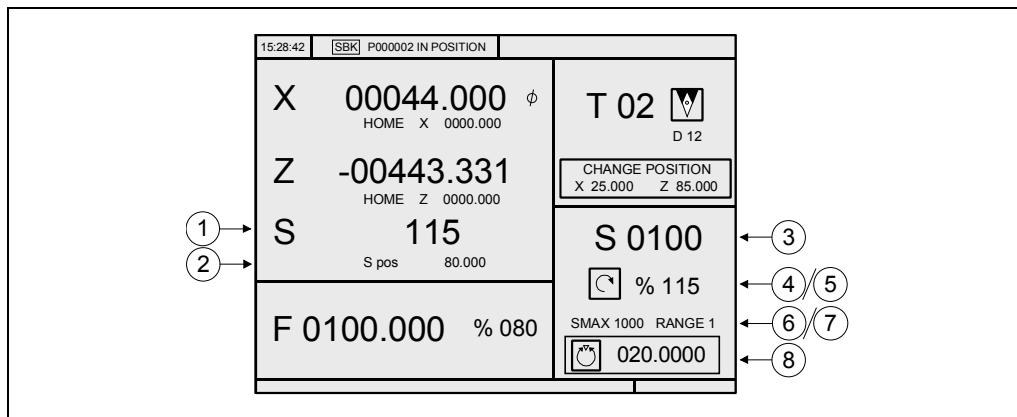
.TC- OPTION
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2.

OPERATING IN JOG MODE
Spindle control

2.8.3 Spindle orientation

When using spindle orientation (general machine parameter REFEED1 (P34) other than 0), the CNC shows the following information.



1. Real spindle speed in rpm.
2. Spindle angular position in degrees.

This information is shown when working in spindle orientation mode. When switching to RPM mode, it only shows the actual (real) spindle speed.

3. Theoretical spindle speed in rpm.

To select another speed, press the [S] key. The CNC highlights the current value.

Enter the new value and press [START]. The CNC assumes that value and refreshes the real spindle speed.

4. Spindle status.



Spindle clockwise

Spindle counterclockwise

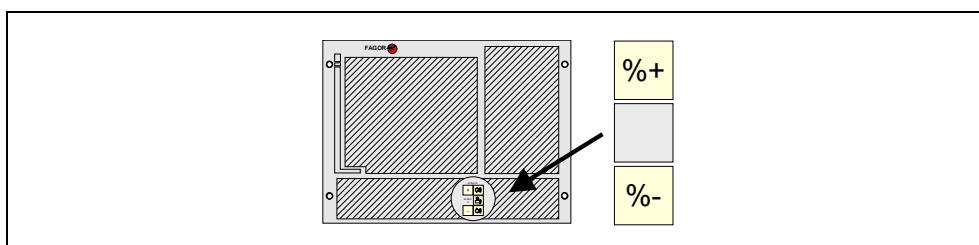
Spindle stopped

When working in spindle orientation mode, it always shows the "spindle stopped" symbol.

5. Percentage of the theoretical spindle speed being applied.

The CNC does not apply this factor when working in spindle orientation mode. It only applies it when working in RPM mode.

To modify the percentage (%), press the following keys.



6. Maximum spindle speed in rpm.

To select another speed, press the [S] key twice. The CNC highlights the current value. Enter the new value and press [ENTER]. The CNC assumes this value and does not allow the spindle to exceed these rpm.

The maximum spindle speed is saved in the MDISL variable. This variable is updated (refreshed) changing the SMAX value and when programming function "G92 S" via ISO.

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.TC. OPTION
SOFT: V01.4x

7. Spindle gear currently selected.

To select another gear, when not using an automatic tool changer, press the [S] key and then use the [**↓**] key until the current value is highlighted.

Enter the gear number to be selected and press [ENTER] or [START].



When the machine does not use spindle gears, this message makes no sense. That is why the CNC does not show this message when text 28 of program 999997 has not been defined.

8. Spindle angular increment when working in spindle orientation mode.

To select another value, press the [S] key three times. The CNC highlights the current value. Enter the new value and press [ENTER].

2

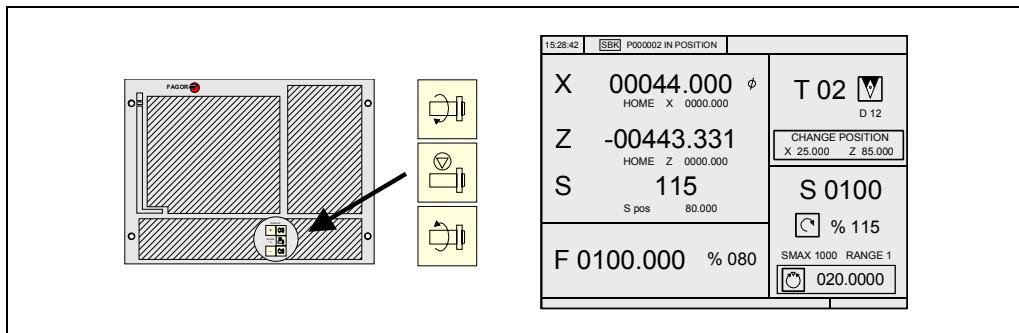
OPERATING IN JOG MODE
Spindle control

Working with spindle orientation

When having spindle orientation, the CNC uses the same screen when working in RPM mode and when working in spindle orientation mode.

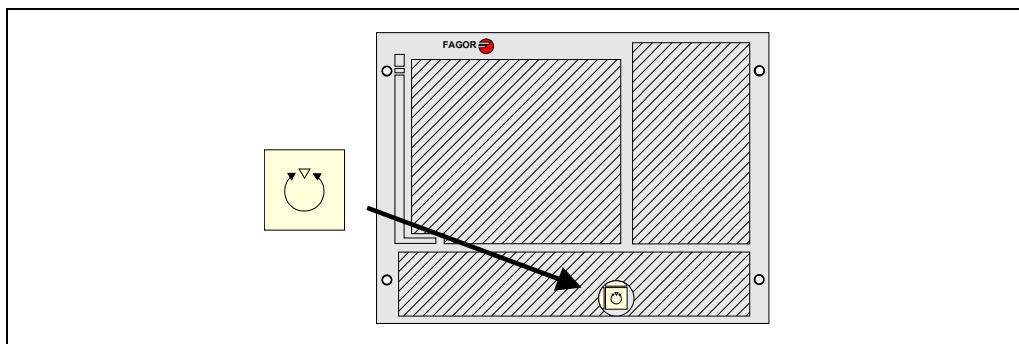
RPM mode.

Press one of these three keys to select this mode. The screen will not show the angular position of the spindle.



Spindle orientation mode

Press the key for spindle orientation to select this mode:



The spindle will stop (if it was turning), it then searches home and finally positions at the angular position indicated on the lower right side of the screen (at 20° in the upper figure).

Every time the spindle orientation key is pressed, the spindle position increments in that value (20° in the upper figure).

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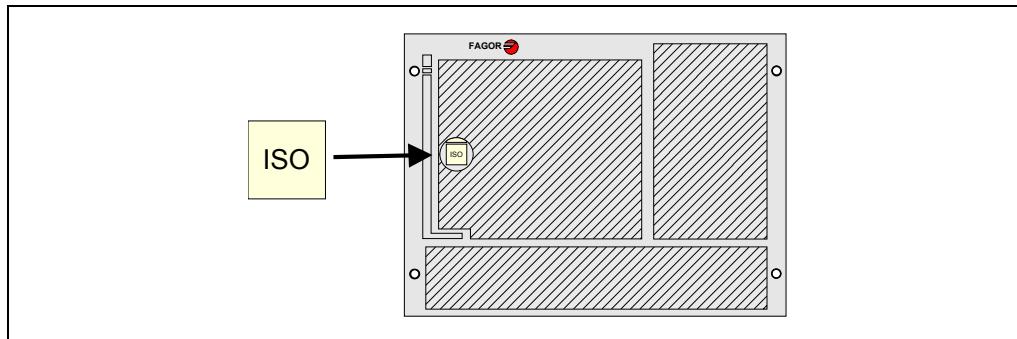
2.9 ISO management

Access to the MDI mode or the ISO mode.

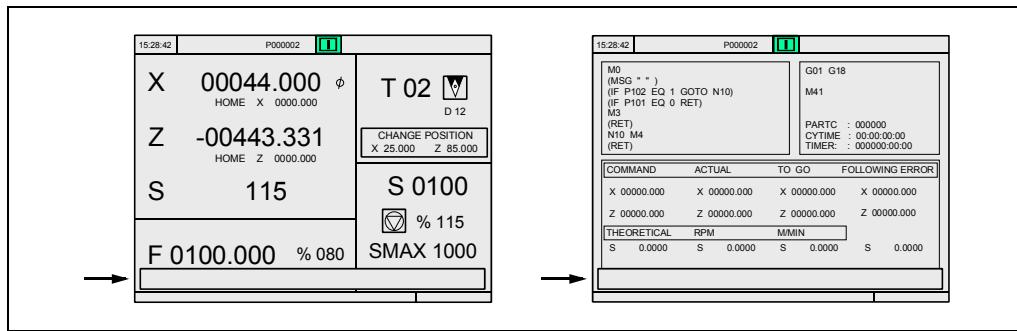
The ISO key may be used to access the MDI mode or the ISO mode.

2.

OPERATING IN JOG MODE
ISO management



To access the MDI mode, the CNC must be in jog mode and the ISO key must be pressed. The CNC will show a window at the bottom of the standard (or special) screen.



In this window, it is possible to edit a block in ISO code and then execute it, like in MDI in T mode.

Displaying the last 10 MDI instructions.

From the MDI mode, pressing the [UP ARROW] or [DOWN ARROW] key opens a window that shows the last 10 instructions that have been executed. This window resizes itself to fit the number of instructions that have been saved.

To execute or modify an MDI line that has been executed earlier, proceed as follows:

- Go into MDI mode.
- Press the [UP ARROW] or [DOWN ARROW] key to open the window that shows the last MDI instructions (up to 10).
- Use the [UP ARROW] or [DOWN ARROW] key to select the desired instruction.
 - Press [START] to execute the selected instruction.
 - Press [ENTER] to modify the selected instruction. Once the instruction has been modified, press [START] to execute it.

Some points to consider:

- An MDI instruction is saved only if it is correct and if it is not the same as the previous one on the list.
- The instructions are kept saved even after turning the unit off.

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.TC. OPTION
SOFT: V01.4x

Generating an ISO-coded program

In the conversational mode of the CNC, it is possible to generate an ISO-coded program from an operation (cycle) or on a part-program. See "[6.5 Graphic representation](#)" on page 184.

2.

OPERATING IN JOG MODE
ISO management



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2.

OPERATING IN JOG MODE
ISO management



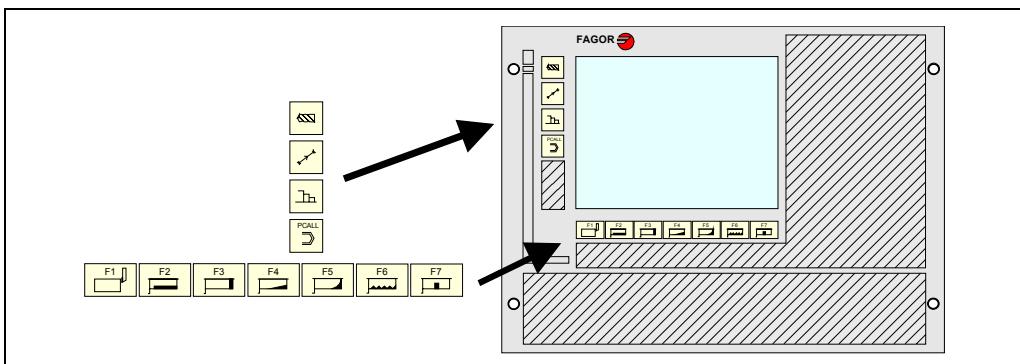
CNC 8037

.TC. OPTION
SOFT: V01.4x

WORKING WITH OPERATIONS OR CYCLES

3

Use the following CNC keys to select the different machining operations or cycles.



User cycles



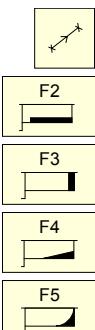
When pressing [PCALL], the CNC shows all the user cycles defined by the machine manufacturer with the WGDRAW application.

The user cycle is edited like any other standard cycle of the TC mode. Once all the required data has been defined, the user can simulate or execute the cycle like any other standard cycle of the TC mode.

Cycles or operations of the CNC

When using any other key, the CNC selects the corresponding standard machining cycle and changes the display on the screen.

Standard machining operations or cycles may be selected with each one of the following keys:



Positioning cycle.

Turning cycle.

Facing cycle.

Taper turning cycle.

Rounding cycle.



Threading cycle.



Grooving cycle.



Drilling and tapping cycle.



Profiling cycle.



When the machining operation or cycle has several levels, either the [PAGE UP] or the [PAGE DOWN] keys must be pressed to select the desired cycle level.

It is possible to combine ISO-coded blocks with standard and/or user cycles to create part-programs. Chapter "[5 Saving programs](#)" describes in detail how to do it and how to use those programs.

To de-select a cycle and return to the standard screen, press the key for the selected cycle or the [ESC] key.



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When operating in conversational mode, do not use global parameters 150 through 299 (both included), because the operations or cycles can modify these parameters and cause the machine to malfunction.

3.

WORKING WITH OPERATIONS OR CYCLES

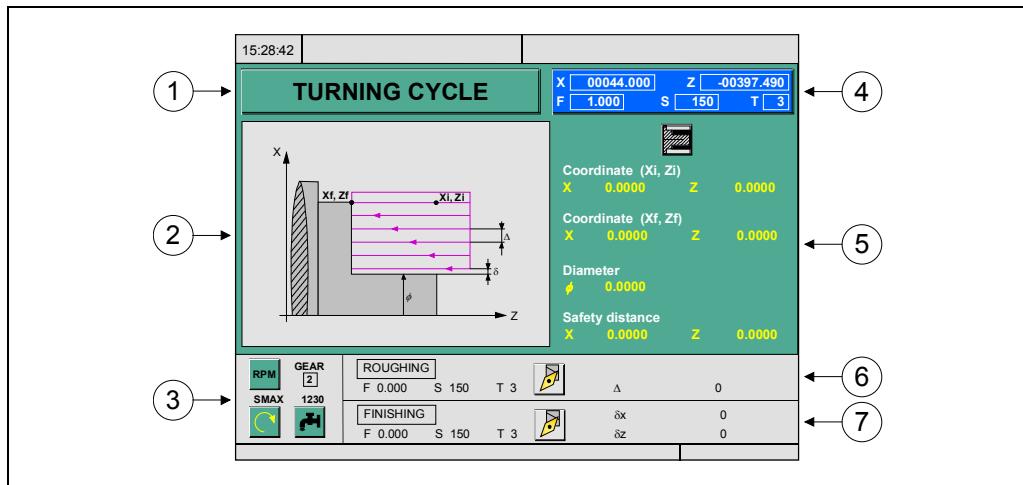


CNC 8037

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3.1 Operation editing mode

Once the operation has been selected, the CNC shows a screen like the following:

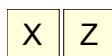


1. Name of the selected operation or work cycle.
2. Help graphics.
3. Spindle conditions to execute the cycle.
4. Current machine status. Coordinates and machining conditions.
5. Data defining the machining geometry.
6. Machining conditions for the roughing operation.
7. Machining conditions for the finishing operation.

The CNC will highlight an icon, a coordinate or one of the data defining the operation or cycle indicating that it has been selected. Use the following keys to select another icon, data or coordinate.



The CNC selects the previous one or the next one.



The CNC selects the first coordinate for that axis. Pressing that key again selects the next coordinate for that axis.



The CNC selects the corresponding roughing data. Pressing that key again selects the corresponding finishing data.



The CNC selects the "S" roughing data. Pressing that key again selects the finishing "S" data and pressing it again selects the data for the spindle SMAX.

The coordinates for the X axis are defined in the work units, radius or diameter. Later on, each operation or cycle indicates the units used to define the data associated with the X axis (safety distance, pass, excess stock, etc.).

3.

WORKING WITH OPERATIONS OR CYCLES

Operation editing mode

FAGOR

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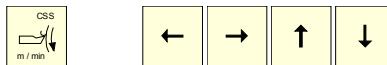
.TC- OPTION
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3.

3.1.1 Definition of spindle conditions

Work mode (RPM) or (CSS)

Place the cursor on the "RPM" or "CSS" icon. To do this, use the [CSS] key or the [\uparrow][\downarrow][\leftarrow][\rightarrow] keys.



Once the data has been selected, press the [CSS] key or the [-] key to change the work mode.

Spindle range

Place the cursor on this data, key in the desired value and press [ENTER].

Maximum spindle speed (S) in rpm

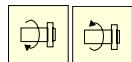
Place the cursor on this data, key in the desired value and press [ENTER].

Spindle turning direction

There are two ways to select the spindle turning direction.



Place the cursor on this data and press the [-] key to change the icon.



Start the spindle in the desired direction with the JOG keys. The CNC starts the spindle and assumes that turning direction as spindle turning data for the cycle.

Coolant

There are two ways to turn the coolant on or off.



Place the cursor on this data and press the [-] key to change the icon.



Turns the coolant on. The CNC sends the M8 function to the PLC.



Turns the coolant off. The CNC sends the M9 function to the PLC.

Once the operation or cycle has ended, or the part-program it belongs to, the CNC sends the M9 function to the PLC.

3.1.2 Definition of machining conditions

Some operations keep the machining conditions throughout the execution (positioning cycles, drilling cycle, etc.). Other operations use some machining conditions for roughing and others for finishing (turning cycle, rounding cycle, etc.).

This section describes how to define all this data.

Selecting the roughing operation.

Place the cursor on the roughing checkbox, select or de-select the roughing operation pressing the [-] key and press [ENTER]. When de-selecting the roughing, all its data will stay in gray.

The data "side finishing stock" of the finishing portion is turned on/off using the roughing checkbox

Selecting the finishing operation.

Place the cursor on the finishing checkbox, select or de-select the finishing operation pressing the [-] key and press [ENTER]. When de-selecting the finishing, all its data will stay in gray.

Axis feedrate (F).



Place the cursor on this data, key in the desired value and press [ENTER].

Spindle speed (S).



Place the cursor on this data, key in the desired value and press [ENTER].

Machining tool (T).

Place the cursor on this data, key in the desired value and press [ENTER].

The CNC updates the associated offset (D) and refreshes the associated icon showing the graphics for the location code (shape) of the new tool.



It is also possible to access the tool calibration mode to check or modify the data for the selected tool. To do that, place the cursor on the "T" and press the key associated with tool calibration.

Press [ESC] to quit the tool calibration mode and return to the cycle

Tool offset number (D).

Place the cursor on this data, key in the desired value and press [ENTER].



Pocket machining direction.

Icon to set the machining direction of the roughing tool.

3.

WORKING WITH OPERATIONS OR CYCLES

Operation editing mode

FAGOR 

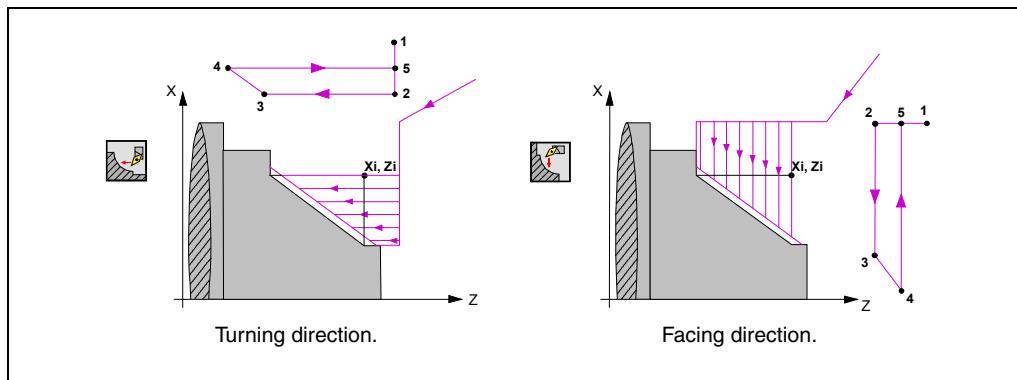
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SOFT: V01.4x

3.

Machining direction.

It is possible to select the machining direction (turning direction or facing direction) in certain cycles.



Place the cursor on this icon and press the [-] key. The icon changes and the help graphics are refreshed.

Roughing pass (Δ).

Place the cursor on this data, key in the desired value and press [ENTER]. The roughing pass is always defined in radius.

Finishing stock (δ).

The roughing pass is always defined in radius. Place the cursor on this data, key in the desired value and press [ENTER].

Sideways penetration angle (β, θ).

Sideways penetration angle. If programmed with a value smaller than or equal to 0°, or greater than 90°, it issues the corresponding error message. If not programmed, it assumes 90°.

Auxiliary (miscellaneous) "M" functions.

There is a window for setting up to 4 auxiliary M functions in the roughing and finishing operations. The functions will be executed in the same order they are inserted on the list.

Select the corresponding window with the [\leftarrow][\rightarrow] keys Use the [\uparrow][\downarrow] keys to move around the window.

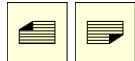
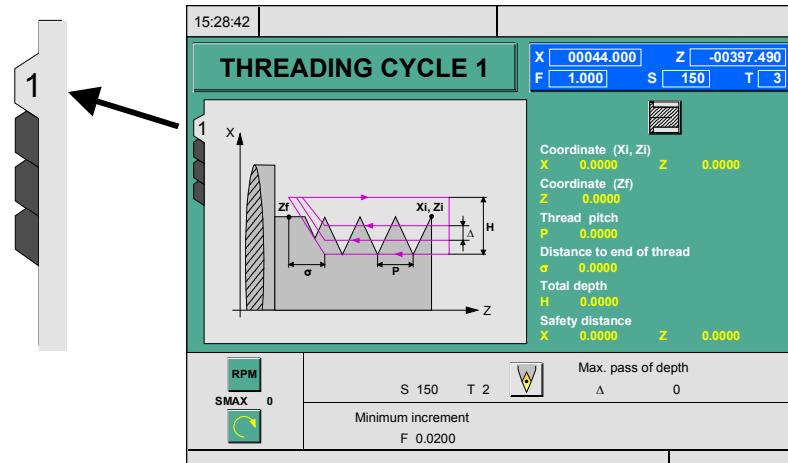
To delete a function, select it and press [CLEAR]



The availability of "M" functions in the cycles is determined with g.m.p. "CODISET (P147)".

3.1.3 Cycle level

All the cycles have several editing levels. Each level has its own screen and the main window of the cycle indicates, with tabs, the available levels and which one is selected.



To change levels, use the [PAGE UP] and [PAGE DOWN] keys to scroll up and down through the various levels.

3.

WORKING WITH OPERATIONS OR CYCLES

Operation editing mode

FAGOR

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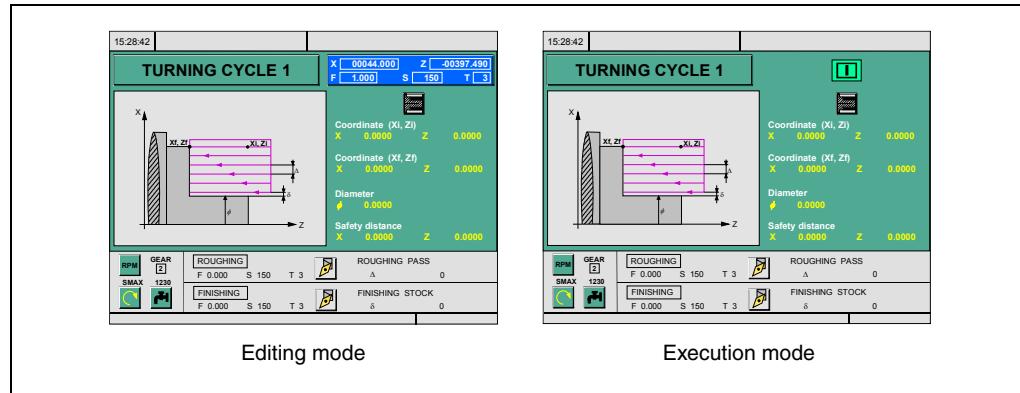
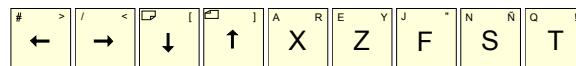
3.

WORKING WITH OPERATIONS OR CYCLES
Simulating and executing the operation

3.2 Simulating and executing the operation

All the operations or cycles have 2 work modes; execution and editing.

- Press [ESC] to switch from editing mode to execution mode.
- Press [ESC] to switch from executing mode to editing mode.



The operation or cycle may be simulated in either mode. To do that, press the [SIMUL] key.



To execute the operation or cycle, select the execution mode and press [START].



For further information on simulating and executing cycles, see the chapter "[6 Execution and simulation](#)".



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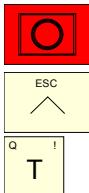
.TC. OPTION
SOFT: V01.4x

3.2.1 Background cycle editing

It is possible to edit an operation or cycle while executing a program or part (background editing). The new operation edited may be saved as part of a part-program other than the one being executed.

The operation being edited in background cannot be executed or simulated, and the current position of the machine cannot be assigned to a coordinate.

Use the following keys to inspect or change a tool while editing in background.



Interrupts the execution and goes on editing in background.

To quit background editing.

To access tool inspection.

Pressing the [T] key without quitting background editing selects the T field of the operation or of the canned cycle being edited.



Background editing is not possible while executing an independent operation or cycle. It can only be done while executing a program or part.

3.

WORKING WITH OPERATIONS OR CYCLES

Simulating and executing the operation

FAGOR

CNC 8037

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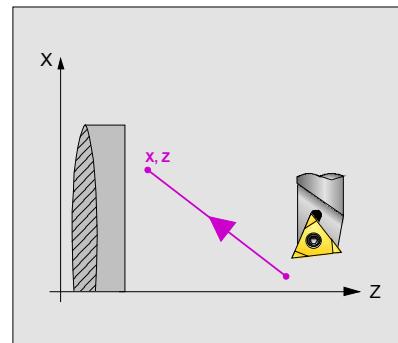
3.3 Positioning cycle

3.



This key accesses the positioning operation.

This cycle may be defined in two ways:



Level 1.

The following data must be defined:

- Coordinates of the target point.
- How the movement is to be carried out.
- The type of feed; in rapid or at the programming feedrate.

Level 2.

The following data must be defined:

- Coordinates of the target point.
- How the movement is to be carried out.
- The type of feed; in rapid or at the programming feedrate.
- The auxiliary functions that will be executed before and after the movement.

3.3.1 Definition of data

Order in which the axes move.



To select the moving order, place the cursor over this icon and press the [-] key.



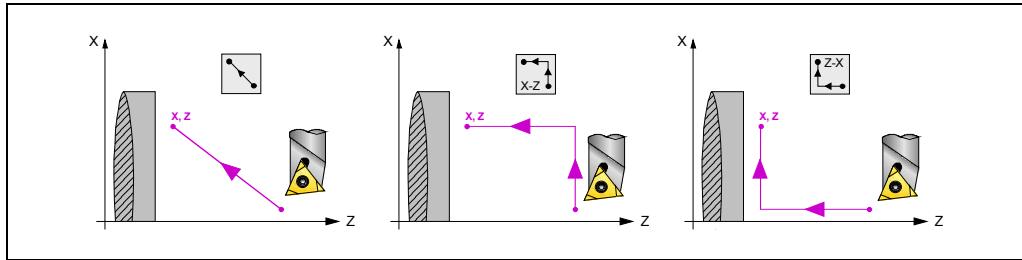
All two axes at the same time.



First X and then Z.



First Z and then X.



Type of feedrate.



To select the type of feedrate, place the cursor over this icon and press the [-] key.



Programmed feedrate.



Rapid traverse.

Coordinates of the target point (X, Z).

The coordinates are defined one by one. After placing the cursor on the coordinates of the axes to be defined, the value is entered in one of the following ways.

- Entering the value manually. Key in the desired value and press [ENTER].
- Assign the current machine position.

Jog the axis with the handwheel or the JOG keys up to the desired point. Press [RECALL] so the selected data assumes the value shown in the top right window and press [ENTER].

The top right window shows the tool position at all times.

Auxiliary (miscellaneous) "M" functions.

Auxiliary "M" functions are those set by the manufacturer for governing various devices of the machine. Some auxiliary "M" functions are used to interrupt the program, to select the spindle turning direction, to control the coolant, the spindle gear box, etc.

The programming manual describes how to program these functions and the installation manual describes how to set up the system to use them.

To define the auxiliary functions, select the corresponding window with the [\leftarrow][\rightarrow] keys. Use the [\uparrow][\downarrow] keys to move around the window. To delete a function, select it and press [CLEAR]

The functions will be executed in the same order they are inserted on the list.

3.

WORKING WITH OPERATIONS OR CYCLES
Positioning cycle

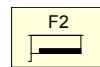
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CNC 8037

.TC- OPTION
SOFT: V01.4x

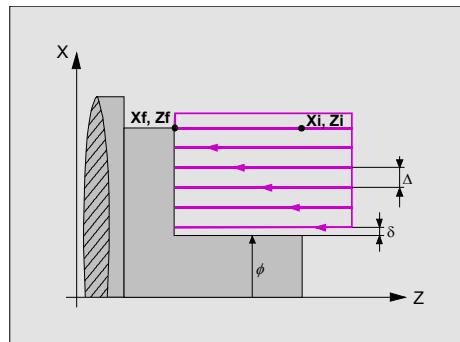
3.4 Turning cycle

3.



This key accesses the turning cycle.

This cycle may be defined in several ways:



Level 1.

The following data must be defined:

- Coordinates of the starting point.
- Coordinates of the last point.
- The final diameter.
- The safety distance.

Level 2.

The following data must be defined:

- Coordinates of the starting point.
- Coordinates of the last point.
- The final diameter.
- The type of machining at each corner.
- The safety distance.

3.4.1 Definition of data

Type of turning operation.



To select the type of turning operation, place the cursor over this icon and press the two-color key.



Inside turning.



Outside turning.

When changing the type of turning operation, the CNC changes the icon and shows the corresponding help screen.

3.

WORKING WITH OPERATIONS OR CYCLES

Turning cycle

Coordinates of the first point (X_i, Z_i) and of the last point (X_f, Z_f).

The coordinates are defined one by one. After placing the cursor on the coordinates of the axes to be defined, the value is entered in one of the following ways.

- Entering the value manually. Key in the desired value and press [ENTER].
- Assign the current machine position.

Jog the axis with the handwheel or the JOG keys up to the desired point. Press [RECALL] so the selected data assumes the value shown in the top right window and press [ENTER].

The top right window shows the tool position at all times.

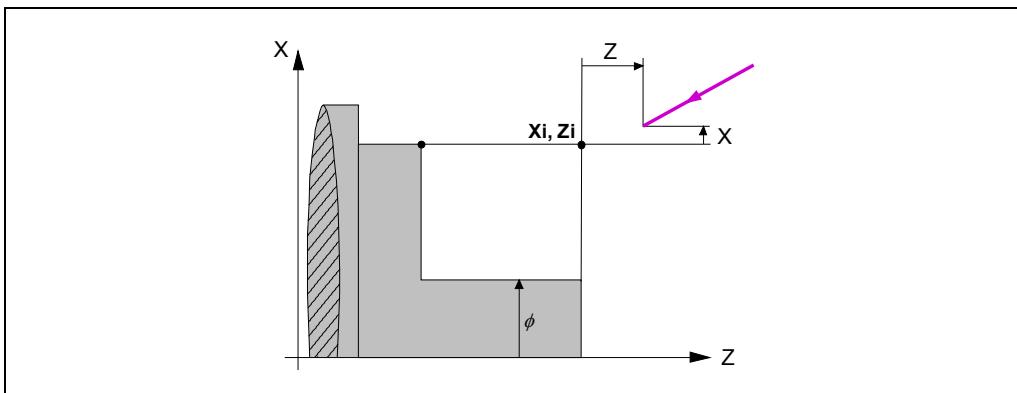
Final diameter (ϕ).

Place the cursor on this data, key in the desired value and press [ENTER].

Safety distance.

A part approach point may be set to prevent the tool from colliding with the part. The safety distance indicates the position of the approach point referred to the starting point (X_i, Z_i).

To change one of these values, place the cursor on the corresponding data, key in the desired value and press [ENTER].



The safety distance in X is always defined in radius.

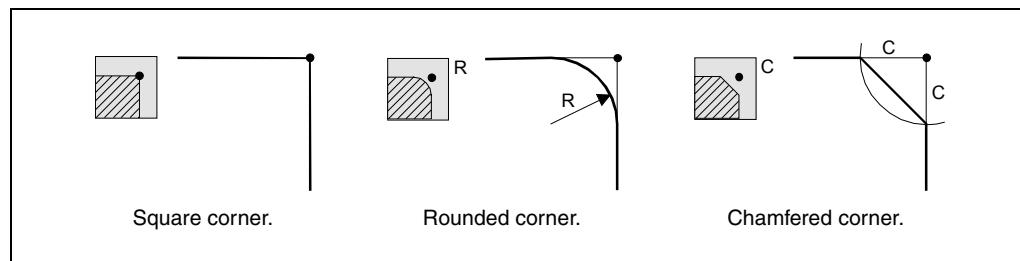
FAGOR

CNC 8037

-TC- OPTION
SOFT: V01.4x

Type of machining for each corner.

To select the type of corner, place the cursor over this icon and press the [-] key.



For a rounded corner, define the rounding radius (R); for a chamfer, define the distance from the theoretical corner to the chamfer point (C).

Finishing stocks in X-Z.

2 different residual stocks may be defined, one for each axis (X, Z). To define these residual stocks, place the cursor on the corresponding data, key in the desired value and press [ENTER].

3.

WORKING WITH OPERATIONS OR CYCLES

Turning cycle

FAGOR

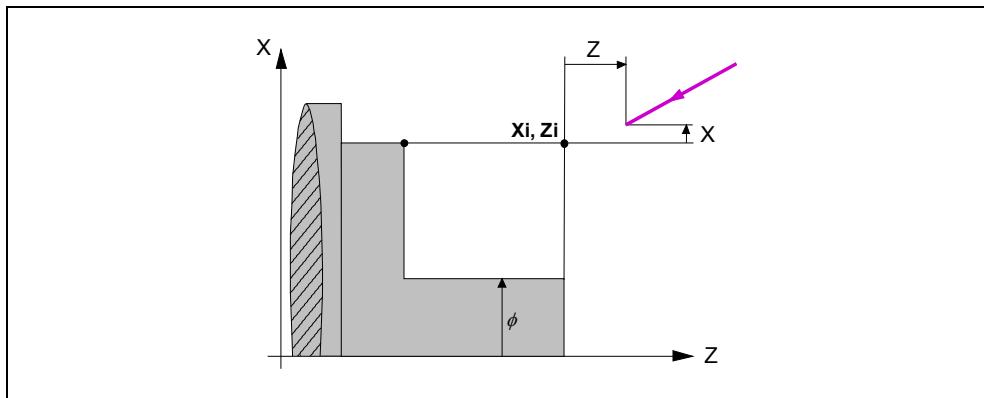
CNC 8037

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3.4.2 Basic operation

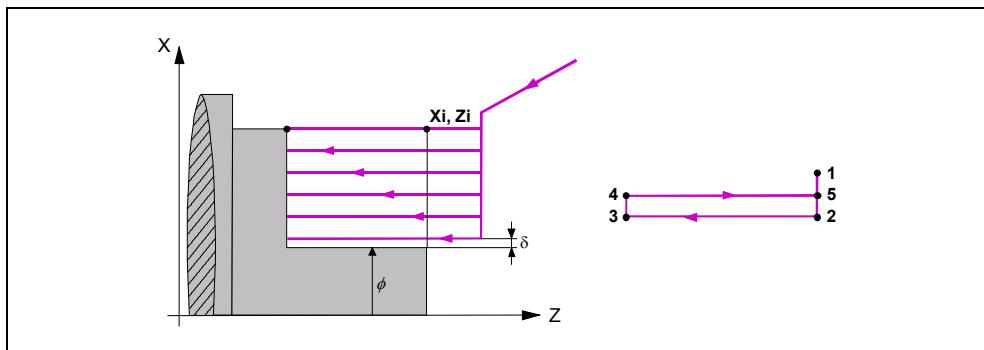
The machining steps for this cycle are:

1. If the roughing 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.
2. The spindle starts at the selected speed and in the indicated direction.
3. The tool approaches in rapid to the starting point (X_i, Z_i) keeping the selected safety distance in X and Z.



4. Roughing operation, with successive turning passes, up to a distance from the selected final diameter equal to the finishing stock.

This operation is carried out with the conditions set for the roughing operations; however, the CNC calculates the actual (real) pass so all the turning passes are identical. This pass will be equal to or under the defined value Δ .



Each turning 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 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 finishing of the part is carried out with the machining conditions set for finishing; feedrate of the axes (F), spindle speed (S) and turning direction.

6. Once the operation or cycle has ended, the tool will return to the position it occupied when calling the cycle; i.e. to the point where [START] was pressed.

When executing an entire part, combination of operations or cycles, the tool does not return to that point after executing each cycle.

7. The CNC stops the spindle, but it keeps the machining conditions set for finishing; tool (T), feedrate of the axes (F) and spindle speed (S).

3.

WORKING WITH OPERATIONS OR CYCLES
Turning cycle

FAGOR

CNC 8037

Considerations

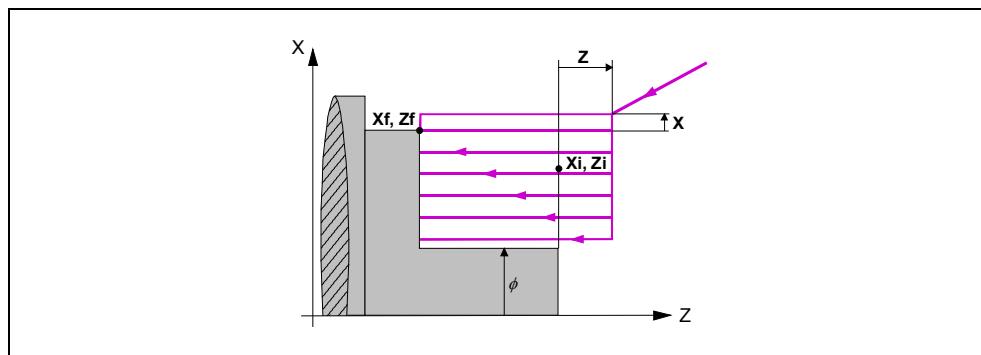
How to leave out the roughing or finishing operations.

By selecting T0 as the roughing tool, the cycle does not execute the roughing operation. In other words, after the approach movement, it will carry out the finishing operation.

By selecting T0 as the finishing tool, the cycle does not execute the finishing operation. In other words, after the roughing operation, the tool will move to the approach point maintaining the safety distance from the starting point (X_i, Z_i).

Different X_i and X_f coordinates.

When the surface to be machined is not totally cylindrical (different X_i and X_f coordinates), the CNC analyzes both coordinates and takes as the X coordinate of the starting point the one farthest away from the final diameter.

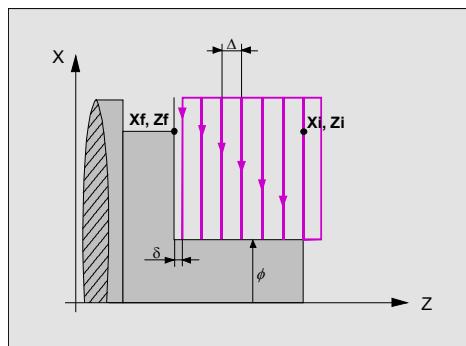


3.5 Facing cycle



This key accesses the facing cycle.

This cycle may be defined in several ways:



3.

WORKING WITH OPERATIONS OR CYCLES
Facing cycle

Level 1.

The following data must be defined:

- Coordinates of the starting point.
- Coordinates of the last point.
- The final diameter.
- The safety distance.

Level 2.

The following data must be defined:

- Coordinates of the starting point.
- Coordinates of the last point.
- The final diameter.
- The type of machining at each corner.
- The safety distance.

FAGOR

CNC 8037

-TC- OPTION
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3.

3.5.1 Definition of data

Coordinates of the first point (X_i, Z_i) and of the last point (X_f, Z_f).

The coordinates are defined one by one. After placing the cursor on the coordinates of the axes to be defined, the value is entered in one of the following ways.

- Entering the value manually. Key in the desired value and press [ENTER].
- Assign the current machine position.

Jog the axis with the handwheel or the JOG keys up to the desired point. Press [RECALL] so the selected data assumes the value shown in the top right window and press [ENTER].

The top right window shows the tool position at all times.

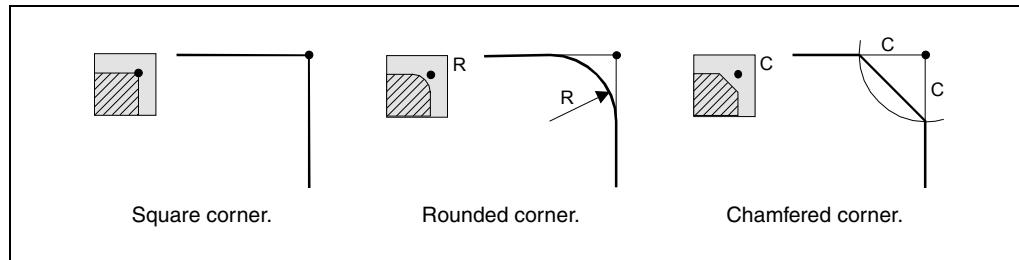
Final diameter (Φ).

Place the cursor on this data, key in the desired value and press [ENTER].

Type of machining for each corner.



To select the type of corner, place the cursor over this icon and press the [-] key.

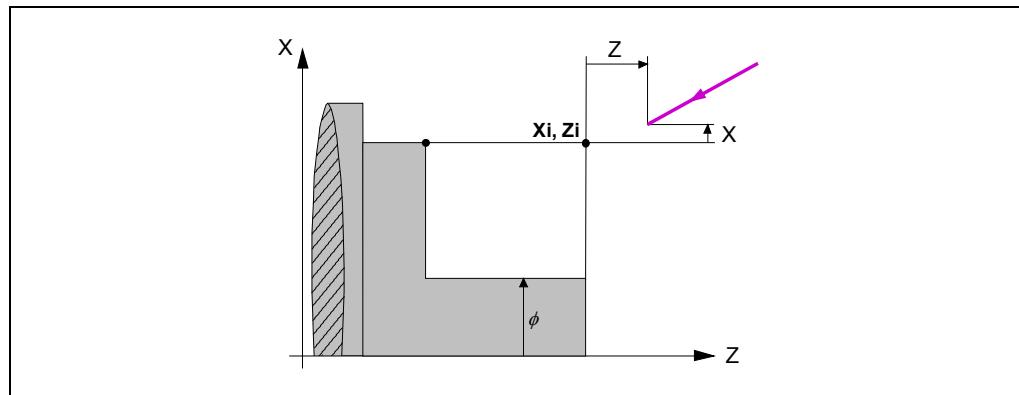


For a rounded corner, define the rounding radius (R); for a chamfer, define the distance from the theoretical corner to the chamfer point (C).

Safety distance.

A part approach point may be set to prevent the tool from colliding with the part. The safety distance indicates the position of the approach point referred to the starting point (X_i, Z_i).

To change one of these values, place the cursor on the corresponding data, key in the desired value and press [ENTER].



The safety distance in X is always defined in radius.

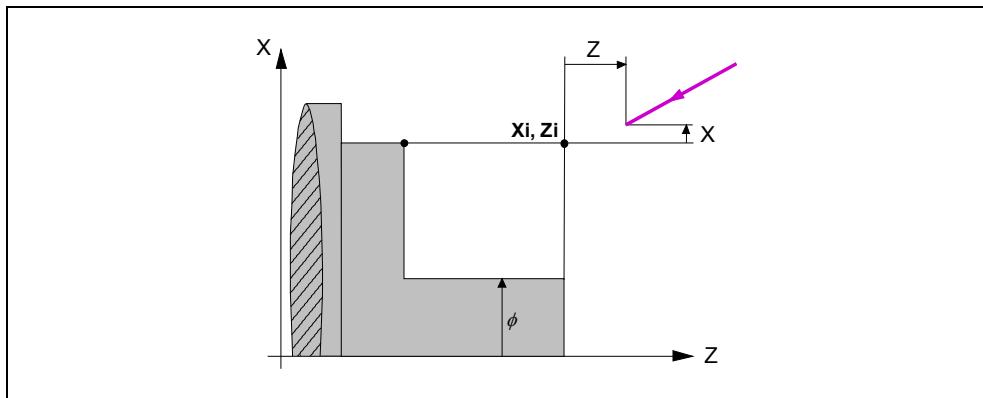
Finishing stocks in X-Z.

2 different residual stocks may be defined, one for each axis (X, Z). To define these residual stocks, place the cursor on the corresponding data, key in the desired value and press [ENTER].

3.5.2 Basic operation

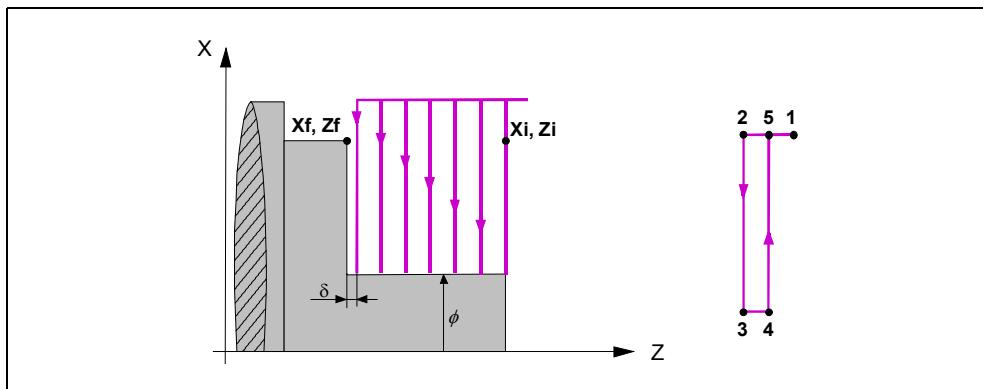
The machining steps for this cycle are:

1. If the roughing 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.
2. The spindle starts at the selected speed and in the indicated direction.
3. The tool approaches in rapid to the starting point (X_i, Z_i) keeping the selected safety distance in X and Z.



4. Roughing operation, with successive facing passes, up to a distance from the selected final Z coordinate (Z_f) equal to the finishing stock.

This operation is carried out with the conditions set for the roughing operations; however, the CNC calculates the actual (real) pass so all the facing passes are identical. This pass will be equal to or under the defined value Δ .



Each facing 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 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 finishing of the part is carried out with the machining conditions set for finishing; feedrate of the axes (F), spindle speed (S) and turning direction.

6. Once the operation or cycle has ended, the tool will return to the position it occupied when calling the cycle; i.e. to the point where [START] was pressed.

When executing an entire part, combination of operations or cycles, the tool does not return to that point after executing each cycle.

7. The CNC stops the spindle, but it keeps the machining conditions set for finishing; tool (T), feedrate of the axes (F) and spindle speed (S).

3.

WORKING WITH OPERATIONS OR CYCLES
Facing cycle

FAGOR

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Considerations

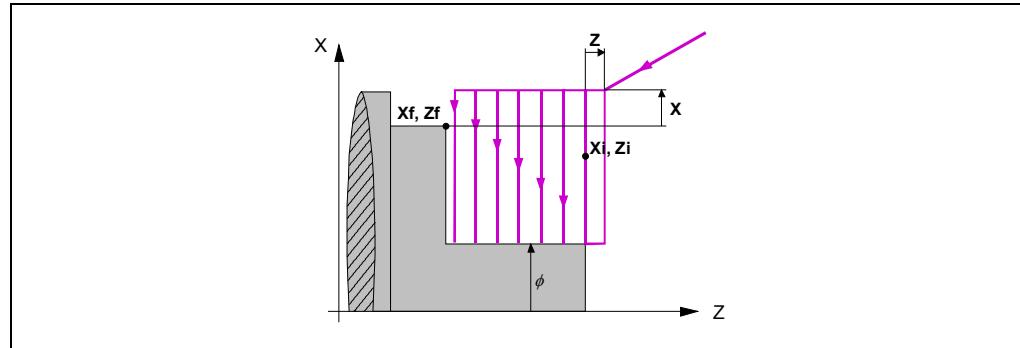
How to leave out the roughing or finishing operations.

By selecting T0 as the roughing tool, the cycle does not execute the roughing operation. In other words, after the approach movement, it will carry out the finishing operation.

By selecting T0 as the finishing tool, the cycle does not execute the finishing operation. In other words, after the roughing operation, the tool will move to the approach point maintaining the safety distance from the starting point (X_i, Z_i).

Different X_i and X_f coordinates.

When the surface to be machined is not totally cylindrical (different X_i and X_f coordinates), the CNC analyzes both coordinates and takes as the X coordinate of the starting point the one farthest away from the final diameter.



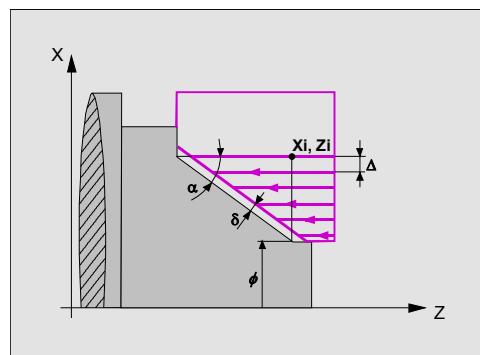
3.6 Taper turning cycle



This key accesses the taper turning cycles.

This cycle may be defined in two ways:

Level 1.



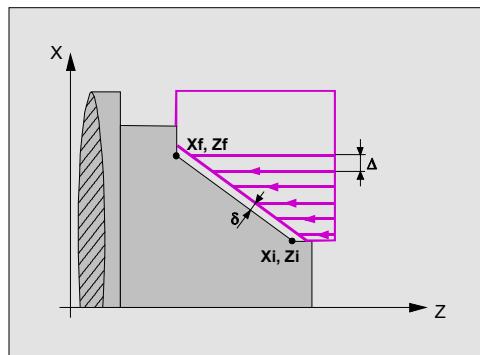
3.

WORKING WITH OPERATIONS OR CYCLES
Taper turning cycle

The following data must be defined:

- The coordinates of the theoretical corner.
- The inclination angle and the final diameter.

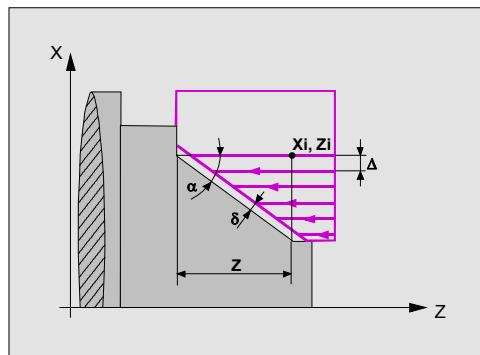
Level 2.



The following data must be defined:

- Coordinates of the starting point.
- Coordinates of the last point.

Level 3.



FAGOR

CNC 8037

The following data must be defined:

- The coordinates of the theoretical corner.
- The taper angle and the distance in Z.

-TC- OPTION
SOFT: V01.4x

3.**WORKING WITH OPERATIONS OR CYCLES**

Taper turning cycle

3.6.1 Definition of data**Type of taper.**

To select the type of taper, place the cursor over this icon and press the [-] key.



Inside taper.

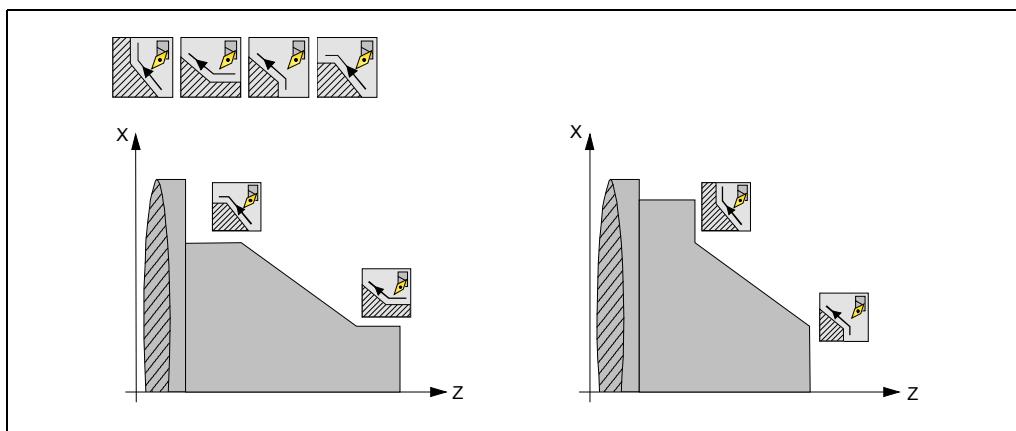


Outside taper.

When changing the type of taper turning operation, the CNC changes the icon and shows the corresponding help screen.

Shape of the part before and after the taper section.

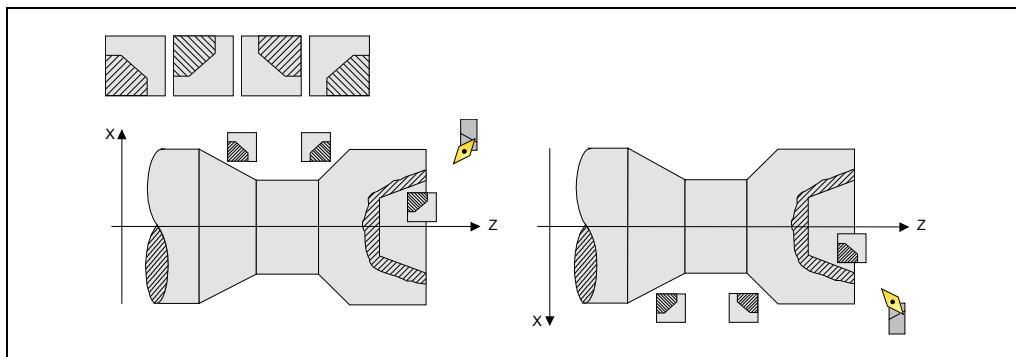
The type of section before and after the taper section is defined with the following icons. To select the desired type, place the cursor over this icon and press the [-] key.



When changing the type of section, the CNC changes the icon and shows the corresponding help screen.

Work quadrant.

The work quadrant is defined with the following icons: To select the desired type, place the cursor over this icon and press the two-color key.


FAGOR

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 ·TC· OPTION
 SOFT: V01.4x

Coordinates of the theoretical corner, of the first point (X_i, Z_i) and of the last point (X_f, Z_f).

The coordinates are defined one by one. After placing the cursor on the coordinates of the axes to be defined, the value is entered in one of the following ways.

- Entering the value manually. Key in the desired value and press [ENTER].
- Assign the current machine position.

Jog the axis with the handwheel or the JOG keys up to the desired point. Press [RECALL] so the selected data assumes the value shown in the top right window and press [ENTER].

The top right window shows the tool position at all times.

3.

WORKING WITH OPERATIONS OR CYCLES
Taper turning cycle

Final diameter (ϕ).

Place the cursor on this data, key in the desired value and press [ENTER].

Angle (α).

Place the cursor on this data, key in the desired value and press [ENTER].

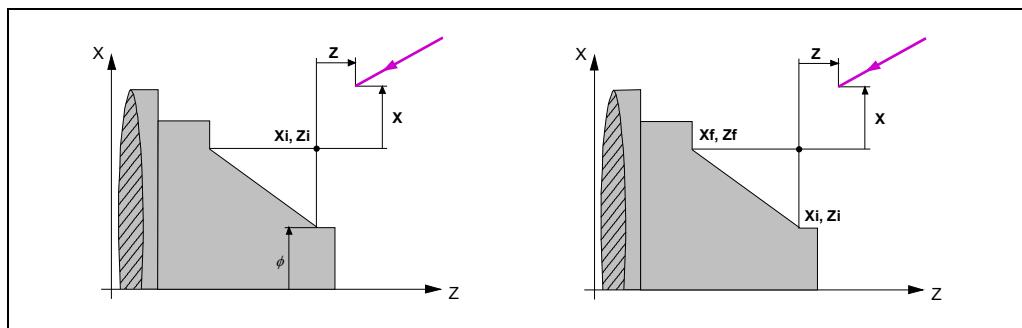
Chamfer (Z).

Length of the taper, measured along the abscissa axis. Place the cursor on this data, key in the desired value and press [ENTER].

Safety distance.

A part approach point may be set to prevent the tool from colliding with the part. The safety distance indicates the position of the approach point referred to the theoretical corner.

To change one of these values, place the cursor on the corresponding data, key in the desired value and press [ENTER].

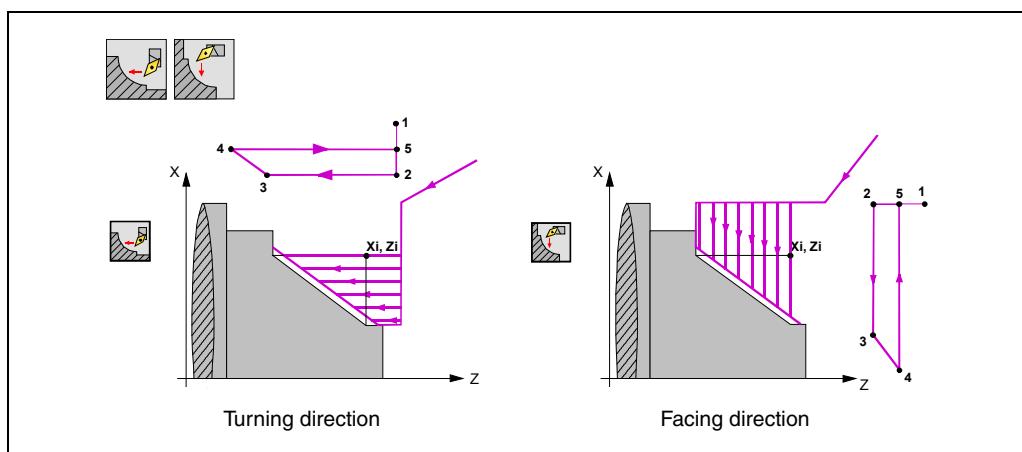


The safety distance in X is always defined in radius.

Machining direction.



The machining direction (turning direction or facing direction) is defined with the following icons. To select the desired type, place the cursor over this icon and press the [-] key.



FAGOR

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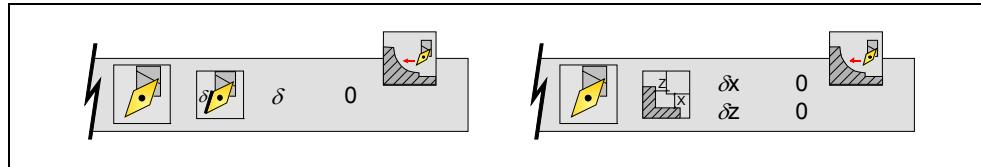
.TC- OPTION
SOFT: V01.4x

When changing the machining direction, the CNC changes the icon and shows the corresponding help screen.

Finishing stocks in X-Z.

Either a single residual stock may be defined that is applied depending on the cutter's edge or 2 different ones, one for each axis (X, Z). It is selected with the following icon located in the finishing area.

- The figure on the left applies a finishing stock depending on the cutter's edge. The finishing stock is measured on the tool's cutting edge.
- The figure on the right lets define 2 finishing stocks, one per axis, regardless of the type of tool used.



To select the type of residual stock, place the cursor over this icon and press the [-] key.
To define these residual stocks, place the cursor on the corresponding data, key in the desired value and press [ENTER].

3.

WORKING WITH OPERATIONS OR CYCLES

Taper turning cycle

FAGOR

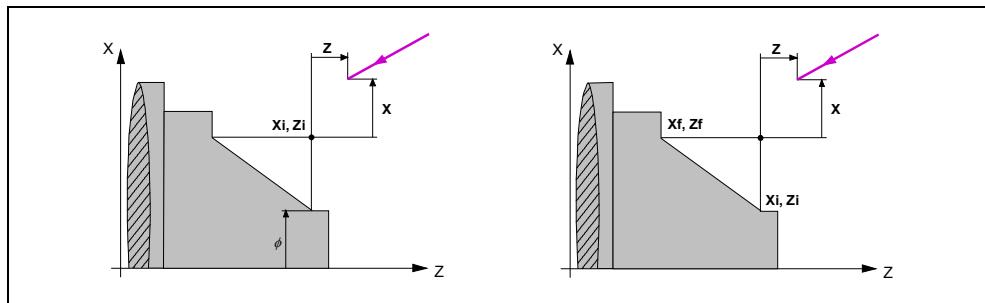
CNC 8037

 .TC. OPTION
 SOFT: V01.4x

3.6.2 Basic operation

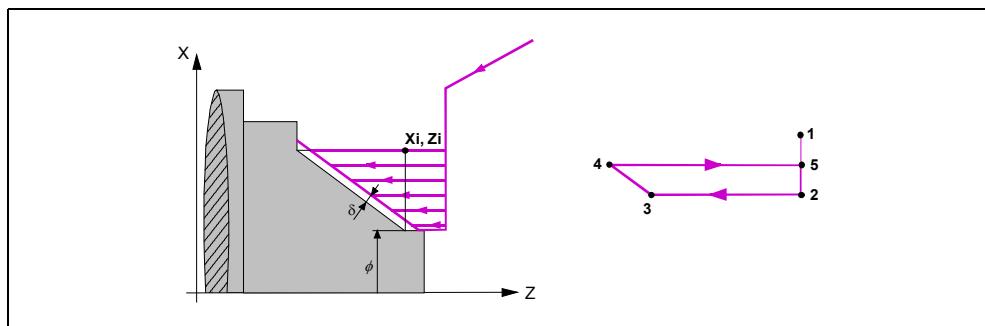
The machining steps for this cycle are:

1. If the roughing 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.
2. The spindle starts at the selected speed and in the indicated direction.
3. The tool approaches in rapid to the theoretical corner keeping the selected safety distance in X and Z.



4. Roughing operation, with successive passes, up to a distance from the selected profile equal to the finishing stock.

This operation is carried out with the conditions set for the roughing operations; however, the CNC calculates the actual (real) pass so all the facing passes are identical. This pass will be equal to or under the defined value Δ .

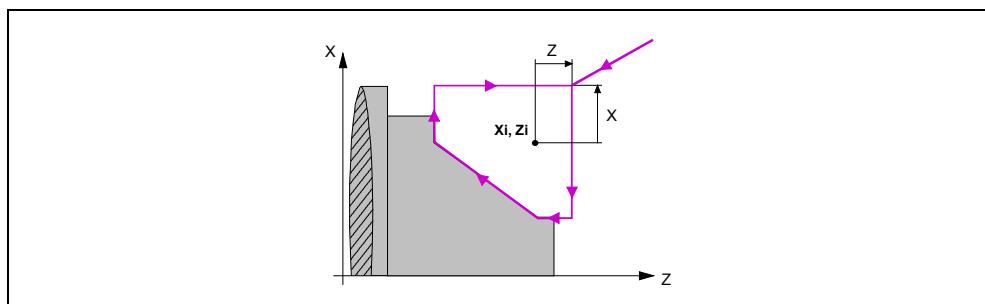


Each machining pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".

5. Finishing operation.

If 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 finishing of the part is carried out with the machining conditions set for finishing; feedrate of the axes (F), spindle speed (S) and turning direction.



6. Once the operation or cycle has ended, the tool will return to the position it occupied when calling the cycle; i.e. to the point where [START] was pressed.

When executing an entire part, combination of operations or cycles, the tool does not return to that point after executing each cycle.

7. The CNC stops the spindle, but it keeps the machining conditions set for finishing; tool (T), feedrate of the axes (F) and spindle speed (S).

3.

WORKING WITH OPERATIONS OR CYCLES
Taper turning cycle

FAGOR

CNC 8037

.TC- OPTION
SOFT: V01.4x

Considerations

How to leave out the roughing or finishing operations.

By selecting T0 as the roughing tool, the cycle does not execute the roughing operation. In other words, after the approach movement, it will carry out the finishing operation.

By selecting T0 as the finishing tool, the cycle does not execute the finishing operation. In other words, after the roughing operation, the tool will move to the approach point maintaining the safety distance from the theoretical corner.

3.

WORKING WITH OPERATIONS OR CYCLES

Taper turning cycle



CNC 8037

.TC. OPTION
SOFT: V01.4x

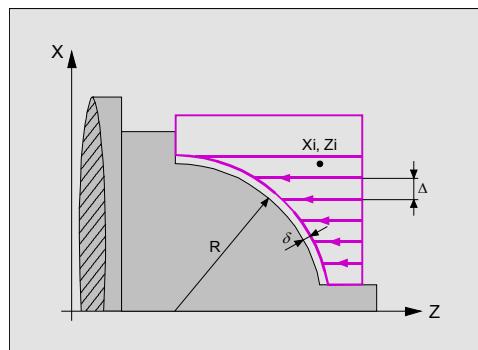
3.7 Rounding cycle



This key accesses the rounding cycles.

This cycle may be defined in two ways:

Level 1.



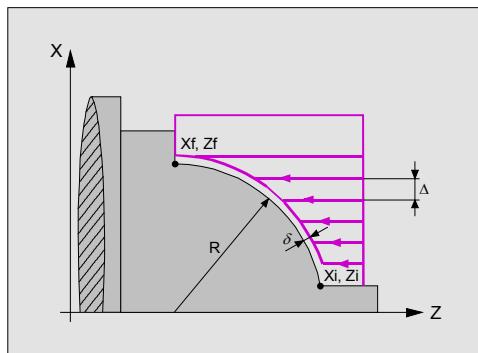
3.

WORKING WITH OPERATIONS OR CYCLES
Rounding cycle

The following data must be defined:

- The coordinates of the theoretical corner.
- The rounding radius.

Level 2.



The following data must be defined:

- Coordinates of the starting point.
- Coordinates of the last point.
- The rounding radius.

FAGOR

CNC 8037

.TC- OPTION
SOFT: V01.4x

3.

3.7.1 Definition of geometry

Type of rounding.



To select the type of rounding, place the cursor over this icon and press the [-] key.



Inside rounding.



Outside rounding.

When changing the type of rounding, the CNC changes the icon and shows the corresponding help screen.

Concave and convex rounding.



The type of section before and after the rounding section is defined with the following icons. To select the desired type, place the cursor over this icon and press the [-] key.



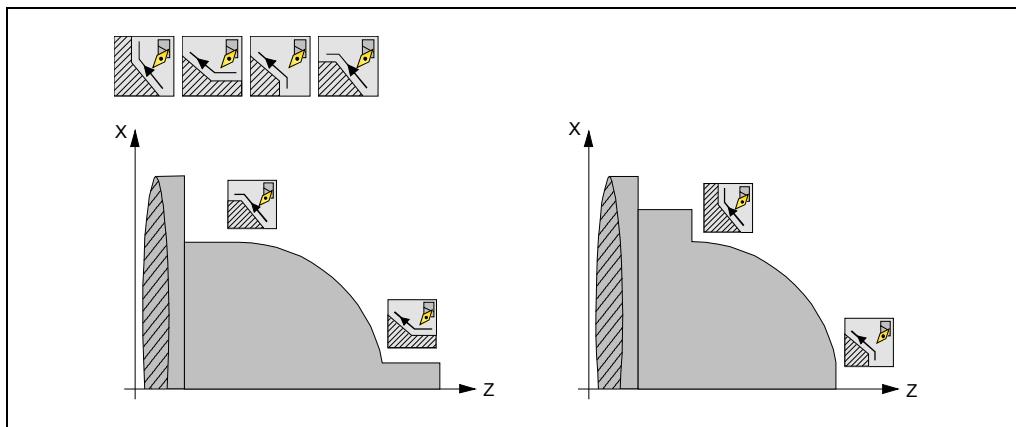
Convex rounding / concave rounding.

When changing one of them, the CNC changes the icon and shows the corresponding help screen.

Shape of the part before and after the rounded section.



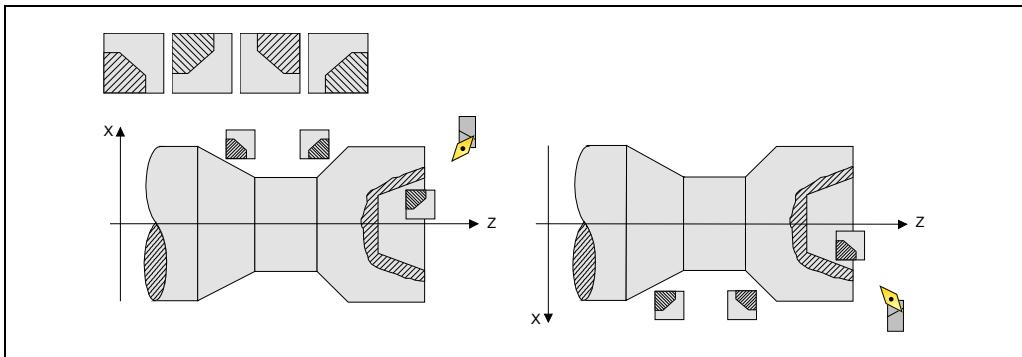
The type of section before and after the rounding section is defined with the following icons. To select the desired type, place the cursor over this icon and press the [-] key.



When changing the type of section, the CNC changes the icon and shows the corresponding help screen.

Work quadrant.

The work quadrant is defined with the following icons: To select the desired type, place the cursor over this icon and press the [-] key.



3.

WORKING WITH OPERATIONS OR CYCLES
Rounding cycle

Coordinates of the theoretical corner or coordinates of the first point (X_i, Z_i) and of the last point (X_f, Z_f).

The coordinates are defined one by one. After placing the cursor on the coordinates of the axes to be defined, the value is entered in one of the following ways.

- Entering the value manually. Key in the desired value and press [ENTER].
- Assign the current machine position.

Jog the axis with the handwheel or the JOG keys up to the desired point. Press [RECALL] so the selected data assumes the value shown in the top right window and press [ENTER].

The top right window shows the tool position at all times.

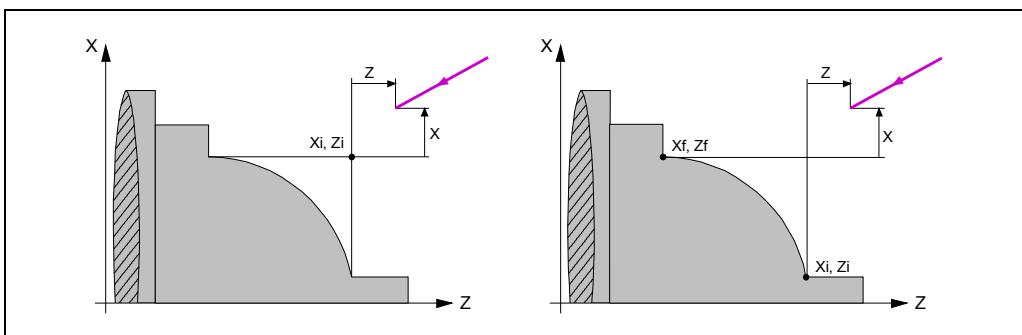
Rounding radius (R).

Place the cursor on this data, key in the desired value and press [ENTER].

Safety distance.

A part approach point may be set to prevent the tool from colliding with the part. The safety distance indicates the position of the approach point referred to the theoretical corner.

To change one of these values, place the cursor on the corresponding data, key in the desired value and press [ENTER].



The safety distance in X is always defined in radius.



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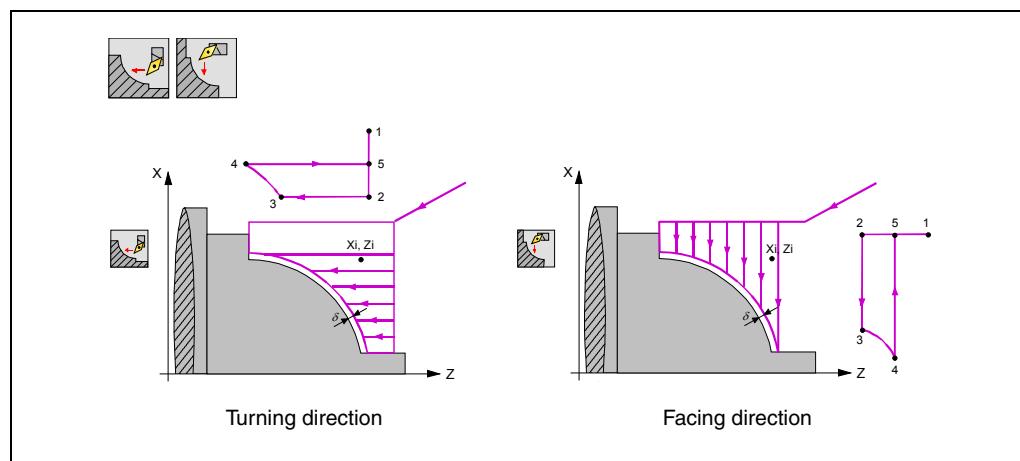
.TC- OPTION
SOFT: V01.4x

Machining direction.

The machining direction (turning direction or facing direction) is defined with the following icons. To select the desired type, place the cursor over this icon and press the [-] key.

3.

WORKING WITH OPERATIONS OR CYCLES
Rounding cycle



When changing the machining direction, the CNC changes the icon and shows the corresponding help screen.

Finishing stocks in X-Z.

Either a single residual stock may be defined that is applied depending on the cutter's edge or 2 different ones, one for each axis (X, Z). It is selected with the following icon located in the finishing area.

- The figure on the left applies a finishing stock depending on the cutter's edge. The finishing stock is measured on the tool's cutting edge.
- The figure on the right lets define 2 finishing stocks, one per axis, regardless of the type of tool used.



To select the type of residual stock, place the cursor over this icon and press the [-] key. To define these residual stocks, place the cursor on the corresponding data, key in the desired value and press [ENTER].

FAGOR

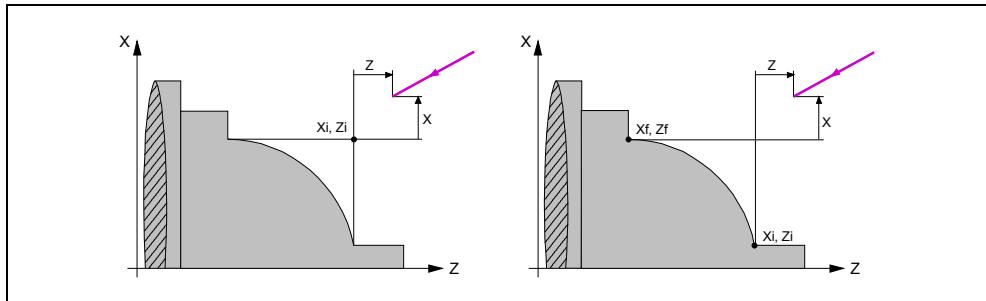
CNC 8037

.TC. OPTION
SOFT: V01.4x

3.7.2 Basic operation

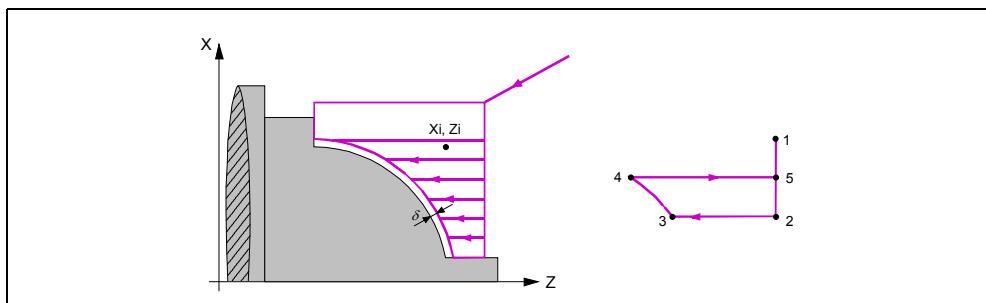
The machining steps for this cycle are:

1. If the roughing 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.
2. The spindle starts at the selected speed and in the indicated direction.
3. The tool approaches in rapid to the theoretical corner keeping the selected safety distance in X and Z.



4. Roughing operation, with successive passes, up to a distance from the selected profile equal to the finishing stock.

This operation is carried out with the conditions set for the roughing operations; however, the CNC calculates the actual (real) pass so all the facing passes are identical. This pass will be equal to or under the defined value Δ .

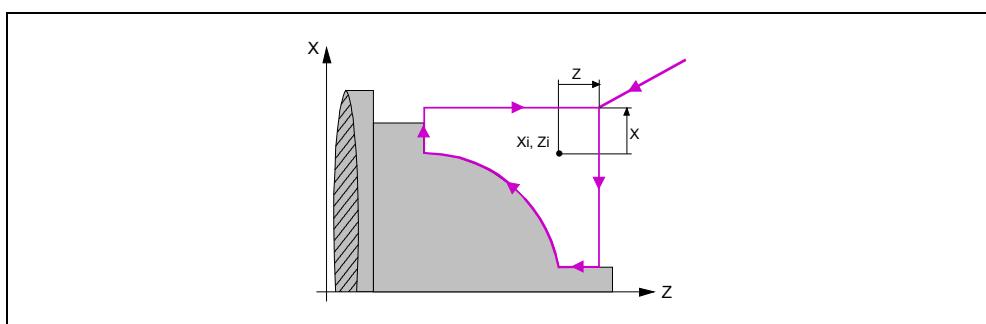


Each machining pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".

5. Finishing operation.

If 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 finishing of the part is carried out with the machining conditions set for finishing; feedrate of the axes (F), spindle speed (S) and turning direction.



6. Once the operation or cycle has ended, the tool will return to the position it occupied when calling the cycle; i.e. to the point where [START] was pressed.

When executing an entire part, combination of operations or cycles, the tool does not return to that point after executing each cycle.

7. The CNC stops the spindle, but it keeps the machining conditions set for finishing; tool (T), feedrate of the axes (F) and spindle speed (S).

3.

WORKING WITH OPERATIONS OR CYCLES
Rounding cycle

FAGOR

CNC 8037

·TC- OPTION
SOFT: V01.4x

Considerations

How to leave out the roughing or finishing operations.

By selecting T0 as the roughing tool, the cycle does not execute the roughing operation. In other words, after the approach movement, it will carry out the finishing operation.

By selecting T0 as the finishing tool, the cycle does not execute the finishing operation. In other words, after the roughing operation, the tool will move to the approach point maintaining the safety distance from the theoretical corner.

3.

WORKING WITH OPERATIONS OR CYCLES

Rounding cycle



CNC 8037

.TC. OPTION
SOFT: V01.4x

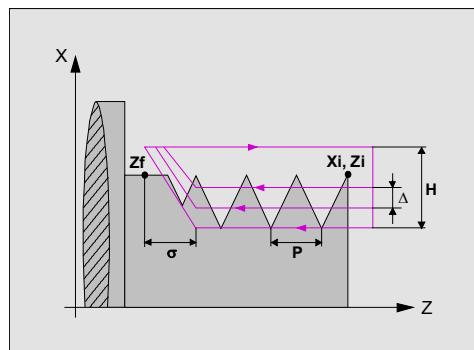
3.8 Threading cycle



This key accesses the threading cycles.

This cycle may be defined in several ways:

Level 1. Longitudinal threadcutting.



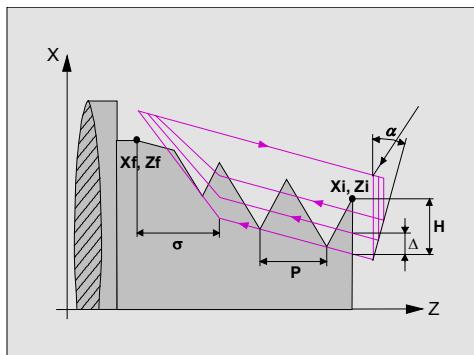
3.

WORKING WITH OPERATIONS OR CYCLES
Threading cycle

The following data must be defined:

- Coordinates of the starting point.
- Z coordinate of the last point.
- The thread pitch.
- The distance to the end of the thread.
- The total depth.
- The spindle angular position.
- The distance to the end of the thread.

Level 2. Taper thread.



The following data must be defined:

- Coordinates of the starting point.
- Coordinates of the last point.
- The thread pitch.
- The distance to the end of the thread.
- The total depth.
- The spindle angular position.
- The distance to the end of the thread.

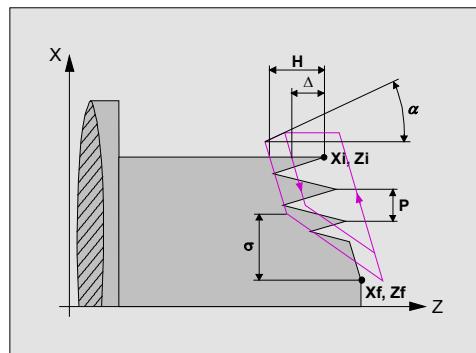
FAGOR

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.TC- OPTION
SOFT: V01.4x

Level 3. Face tapping.**3.****WORKING WITH OPERATIONS OR CYCLES**

Threading cycle



The following data must be defined:

- Coordinates of the starting point.
- Coordinates of the last point.
- The thread pitch.
- The distance to the end of the thread.
- The total depth.
- The spindle angular position.
- The distance to the end of the thread.

FAGOR

CNC 8037

 .TC. OPTION
 SOFT: V01.4X

3.8.1 Definition of geometry

Type of tapping.



To select the type of threading, place the cursor over this icon and press the [-] key.



Inside threading.



Outside threading.

When changing the type of threading, the CNC changes the icon and shows the corresponding help screen.

3.

WORKING WITH OPERATIONS OR CYCLES

Threading cycle

Coordinates of the first point (X_i, Z_i) and of the last point (X_f, Z_f).

The coordinates are defined one by one. After placing the cursor on the coordinates of the axes to be defined, the value is entered in one of the following ways.

- Entering the value manually. Key in the desired value and press [ENTER].
- Assign the current machine position.

Jog the axis with the handwheel or the JOG keys up to the desired point. Press [RECALL] so the selected data assumes the value shown in the top right window and press [ENTER].

The top right window shows the tool position at all times.

Standard threads.

Up to 6 standard thread types may be selected. See "[3.8.2 Standard threads](#)" on page 111.

M (S.I.)	Regular pitch metric thread (international system).
M (S.I.F.)	Fine pitch metric thread (international system).
B.S.W. (W)	Regular pitch Whitworth thread.
B.S.F.	Fine pitch Whitworth thread.
U.N.C.	Regular pitch unified American thread.
U.N.F.	Fine pitch unified American thread.

When selecting one of them, the pitch and depth of the thread are calculated automatically; if none is selected, the pitch and depth of the thread must be defined.

Defining the thread as number of threads per inch.

In any threading cycle, the thread pitch may be entered as number of threads per inch. This may be done regardless of whether it is working in millimeters or in inches.

To define the thread as number of threads per inch, instead of selecting the standard thread, select a free thread and press [ENTER]. In the following data, use the [-] key, select "Threads/inch" and press [ENTER]. Then, place the cursor on the P data, key in the desired value and press [ENTER].

Total depth of the thread (H).

The total depth must be programmed in radius and with a positive value. To define that value, place the cursor on this data, key in the desired value and press [ENTER].

Thread pitch (P).

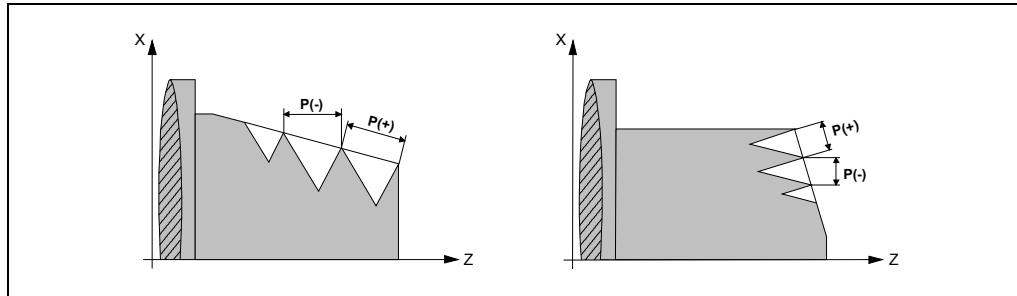
The thread pitch may be defined along the inclination of the thread or along the associated axis. In either case, parameter "P" is be used, but with a different sign.



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.TC- OPTION
SOFT: V01.4x

3.



- "P" with a positive sign to program the pitch is programmed along the inclination of the thread.
- "P" with a negative sign to program the pitch along the associated axis.

To define the pitch, place the cursor on this data, key in the desired value and press [ENTER].

Increment/decrement of thread pitch (ΔP)

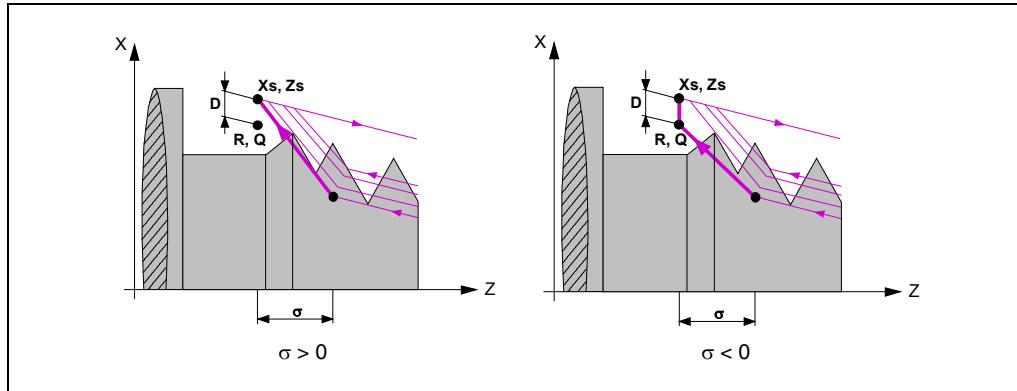
It defines the thread pitch increment or decrement per spindle revolution.



It must be borne in mind that if a thread pitch decrement is programmed and the pitch reaches 0 before the end of the thread cutting operation, the CNC will display the corresponding error message.

Distance to end of thread (σ).

This parameter indicates at what distance from the end of the thread the tool starts exiting it. It keeps threading during this exit movement. Place the cursor on this data, key in the desired value and press [ENTER].



- If programmed with a positive value, the tool exits the thread without going through the final point (X_f, Z_f).
- If programmed with a negative value, the tool exits the thread going through the final point (X_f, Z_f).
- If not programmed, a value of 0 is assumed (blind thread).

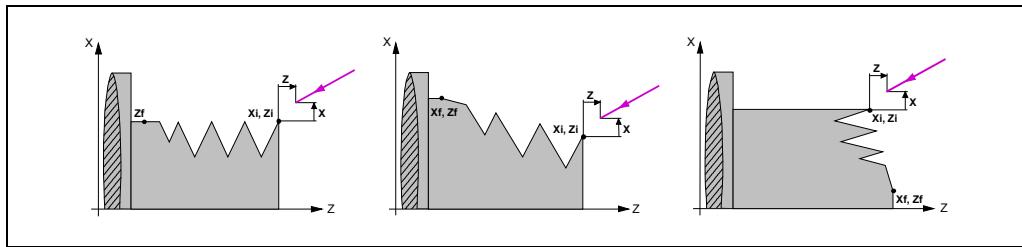
The third set of gains and accelerations for the axes and the spindle may be used to improve the adjustment and the machining of the exit from blind threads. If the thread exit is a short distance, any acceleration set may be used or even remove the acceleration without getting the error "insufficient acceleration while threading".

We recommend to use low or no acceleration.

Safety distance.

A part approach point may be set to prevent the tool from colliding with the part. The safety distance indicates the position of the approach point referred to the theoretical corner.

To change one of these values, place the cursor on the corresponding data, key in the desired value and press [ENTER].



The safety distance in X is always defined in radius.

3.

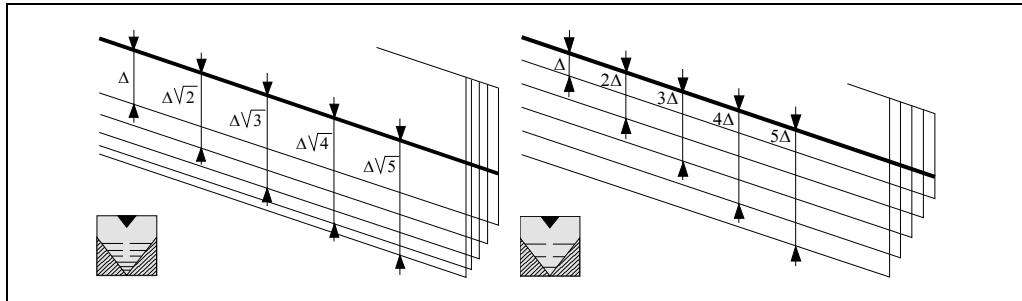
WORKING WITH OPERATIONS OR CYCLES
Threading cycle

 The Δ data sets the maximum penetration pass and the following icons define how the successive machining passes are run. To select the type of pass depth, place the cursor over this icon and press the [-] key.

 The depth of each pass will depend on the number of the corresponding pass. The penetrations are Δ , $\Delta\sqrt{2}$, $\Delta\sqrt{3}$, $\Delta\sqrt{4}$, etc.

If the penetration increment (different between penetrations) calculated by the CNC is smaller than the minimum penetration increment, the CNC assumes this last value.

 The penetration increment is kept constant between passes with a value equal to the one programmed Δ .



For the level 1 threading cycle, the depth of each pass depends on the corresponding pass number (Δ , $\Delta\sqrt{2}$, $\Delta\sqrt{3}$, $\Delta\sqrt{4}$, etc.).

Type of tool penetration.

 To select the type of tool penetration, place the cursor over the icon and press the [-] key.

 Radial penetration.

 First-flank penetration.

 Zig-zag radial penetration from the first center towards the first flank.

 Last-flank penetration.

 Zig-zag radial penetration from the first center towards the last flank.

 Zig-zag penetration by the flanks.

To enable zig-zag penetration by the flanks set bit 10 of g.m.p. CODISET (P147) =1.

When selecting any flank type penetration or in zig-zag, the CNC will request the cutter's penetration angle (α).

For the level 1 threading cycle, the type of penetration is always radial.

FAGOR 

CNC 8037

.TC- OPTION
SOFT: V01.4x

Repeat the last threading pass.

To select whether to repeat the last pass or not, place the cursor over this icon and press the two-color key.



Do not repeat the last pass.



Repeat the last pass.

3.**WORKING WITH OPERATIONS OR CYCLES**

Threading cycle

FAGOR **CNC 8037**.TC. OPTION
SOFT: V01.4x

3.8.2 Standard threads

At all the levels except in face threading, the diameter value may be entered so the CNC calculates the relevant pitch and depths.

A field (window) may be used to select the type of standard thread; if none is selected, the pitch and depth of the thread must be defined.

The available thread types are:

- | | |
|------------|---|
| M (S.I.) | Regular pitch metric thread (international system). |
| M (S.I.F.) | Fine pitch metric thread (international system). |
| B.S.W. (W) | Regular pitch Whitworth thread. |
| B.S.F. | Fine pitch Whitworth thread. |
| U.N.C. | Regular pitch unified American thread. |
| U.N.F. | Fine pitch unified American thread. |

3.

WORKING WITH OPERATIONS OR CYCLES
Threading cycle



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.TC- OPTION
SOFT: V01.4x

Regular pitch metric thread: M (S.I.)

3.

WORKING WITH OPERATIONS OR CYCLES

Threading cycle



CNC 8037

.TC: OPTION
SOFT: V01.4x

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

Depth in internal threads = 0.5413 x Pitch

Depth in external threads = 0.6134 x Pitch

Fine pitch metric thread: M (S.I.F.)

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

Depth in internal threads = 0.5413 x Pitch

Depth in external threads = 0.6134 x Pitch

3.

WORKING WITH OPERATIONS OR CYCLES

Threading cycle



CNC 8037

.TC- OPTION
SOFT: V01.4x

Whitworth thread with normal pitch: B.S.W. (W)**3.****WORKING WITH OPERATIONS OR CYCLES**

Threading cycle

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

The threads must be defined in mm 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:

$$\text{Pitch in mm} = 25.4 / \text{number of edges}$$

$$\text{Pitch in inches} = 1 / \text{number of edges}$$

$$\text{Depth in internal threads} = 0.6403 \times \text{Pitch}$$

$$\text{Depth in external threads} = 0.6403 \times \text{Pitch}$$



CNC 8037

.TC. OPTION
SOFT: V01.4x

Whitworth thread with fine pitch: B.S.F

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

The threads must be defined in mm 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:

$$\text{Pitch in mm} = 25.4 / \text{number of edges}$$

$$\text{Pitch in inches} = 1 / \text{number of edges}$$

$$\text{Depth in internal threads} = 0.6403 \times \text{Pitch}$$

$$\text{Depth in external threads} = 0.6403 \times \text{Pitch}$$

3.

WORKING WITH OPERATIONS OR CYCLES

Threading cycle



CNC 8037

·TC· OPTION
SOFT: V01.4x

Regular pitch unified American thread: UNC (NC,USS)**3.****WORKING WITH OPERATIONS OR CYCLES**

Threading cycle

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

The threads must be defined in mm 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:

$$\text{Pitch in mm} = 25.4 / \text{number of edges}$$

$$\text{Pitch in inches} = 1 / \text{number of edges}$$

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

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



CNC 8037

.TC. OPTION
SOFT: V01.4x

Fine pitch unified American thread: UNF (NF,SAE)

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

The threads must be defined in mm 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:

$$\text{Pitch in mm} = 25.4 / \text{number of edges}$$

$$\text{Pitch in inches} = 1 / \text{number of edges}$$

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

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

3.

WORKING WITH OPERATIONS OR CYCLES

Threading cycle



CNC 8037

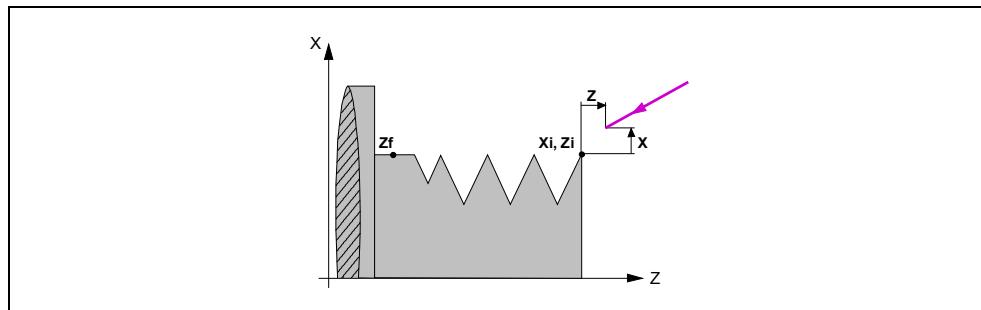
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 SOFT: V01.4x

3.

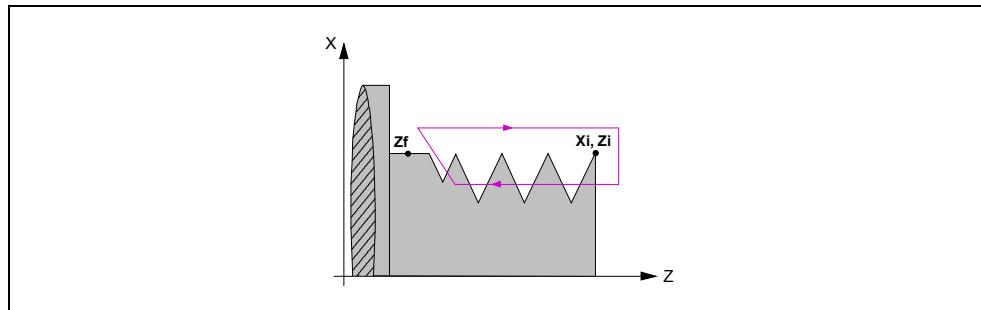
WORKING WITH OPERATIONS OR CYCLES
Threading cycle**3.8.3 Basic operation. Longitudinal threadcutting**

The machining steps for this cycle are:

1. If the 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.
2. The spindle starts at the selected speed and in the indicated direction. Depending on the spindle turning direction, the thread will be clockwise or counterclockwise.
3. The tool approaches in rapid to the theoretical corner keeping the selected safety distance in X and Z.



4. The threading is done with radial penetration and in successive passes until the total depth is reached. The depth of each pass depends on the corresponding pass number Δ , $\Delta\sqrt{2}$, $\Delta\sqrt{3}$, $\Delta\sqrt{4}$, etc.



Each threading step is carried out as follows:

- Rapid move to the corresponding programmed depth.
 - Threading of the programmed section, first along the Z up to the distance from the end of the thread (σ) and then exit threading up to the final coordinate. The feedrate F cannot be varied with the feedrate override switch while threading; it will stay at 100%. At the beginning of the machining operation on large lathes, when cutting long threads, the spindle speed may be overridden in the first passes to prevent the part from bending.
 - Withdrawal in rapid to the approach point.
5. Once the operation or cycle has ended, the tool will return to the position it occupied when calling the cycle; i.e. to the point where [START] was pressed.
- When executing an entire part, combination of operations or cycles, the tool does not return to that point after executing each cycle.
6. The CNC stops the spindle, but it keeps the machining conditions set; tool (T), feedrate of the axes (F) and spindle speed (S).



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.TC. OPTION
SOFT: V01.4x

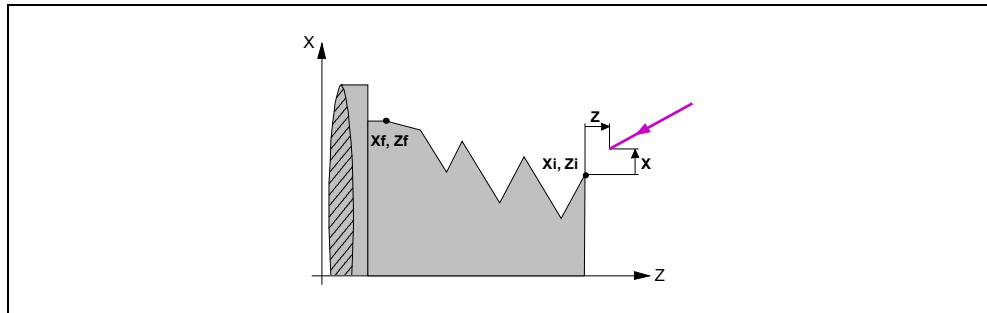
3.8.4 Basic operation. Taper thread

The machining steps for this cycle are:

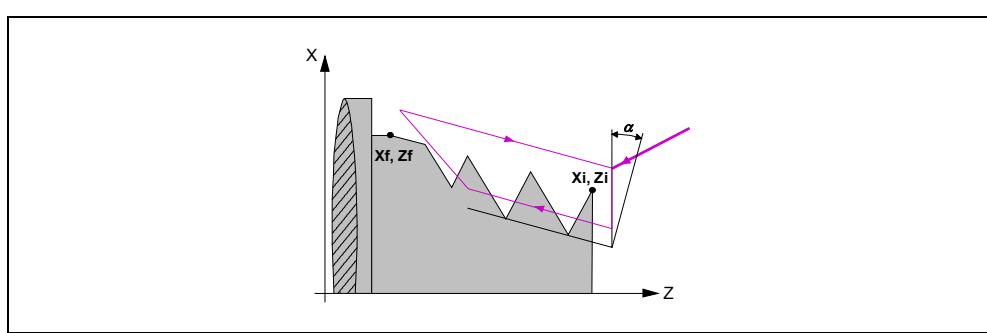
1. If the operation has been programmed with another tool, the CNC will change the tool and will move to the change position if so required by the machine.
2. The spindle starts at the selected speed and in the indicated direction. Depending on the spindle turning direction, the thread will be clockwise or counterclockwise.
3. The tool approaches in rapid to the theoretical corner keeping the selected safety distance in X and Z.

3.

WORKING WITH OPERATIONS OR CYCLES
Threading cycle



4. The threading is done in successive passes until the total depth is reached. The depth of each pass will depend on the selected model.
 - Depth depending on the pass number Δ , $\Delta\sqrt{2}$, $\Delta\sqrt{3}$, $\Delta\sqrt{4}$, etc.
 - Depth keeping the increment between passes Δ constant.



Each threading step is carried out as follows:

- Rapid move to the corresponding programmed depth. This move will be made according to the penetration angle (α) of the selected tool.
 - Threading of the programmed section, first along the defined profile up to the distance from the end of the thread (σ) and then exit threading up to the final coordinate. The feedrate F cannot be varied with the feedrate override switch while threading; it will stay at 100%. At the beginning of the machining operation on large lathes, when cutting long threads, the spindle speed may be overridden in the first passes to prevent the part from bending.
 - Withdrawal in rapid to the approach point.
5. Once the operation or cycle has ended, the tool will return to the position it occupied when calling the cycle; i.e. to the point where [START] was pressed.
- When executing an entire part, combination of operations or cycles, the tool does not return to that point after executing each cycle.
6. The CNC stops the spindle, but it keeps the machining conditions set; tool (T), feedrate of the axes (F) and spindle speed (S).

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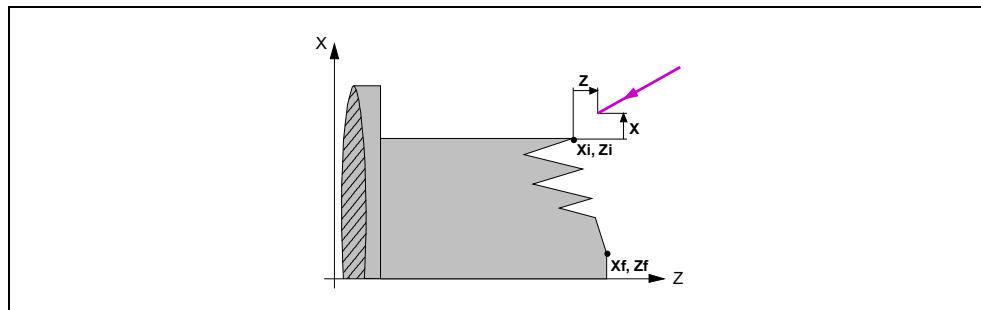
.TC- OPTION
SOFT: V01.4x

3.

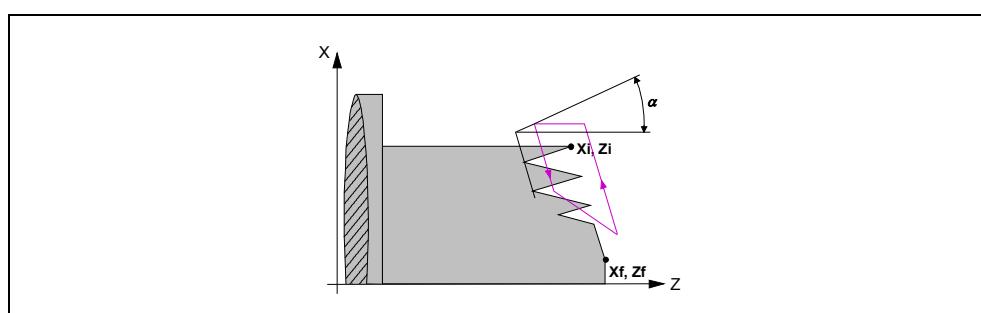
3.8.5 Basic operation. Face tapping

The machining steps for this cycle are:

1. If the 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.
2. The spindle starts at the selected speed and in the indicated direction. Depending on the spindle turning direction, the thread will be clockwise or counterclockwise.
3. The tool approaches in rapid to the theoretical corner keeping the selected safety distance in X and Z.



4. The threading is done in successive passes until the total depth is reached. The depth of each pass will depend on the selected model.
 - Depth depending on the pass number Δ , $\Delta\sqrt{2}$, $\Delta\sqrt{3}$, $\Delta\sqrt{4}$, etc.
 - Depth keeping the increment between passes Δ constant.



Each threading step is carried out as follows:

- Rapid move to the corresponding programmed depth. This move will be made according to the penetration angle (α) of the selected tool.
 - Threading of the programmed section, first along the defined profile up to the distance from the end of the thread (σ) and then exit threading up to the final coordinate. The feedrate F cannot be varied with the feedrate override switch while threading; it will stay at 100%. At the beginning of the machining operation on large lathes, when cutting long threads, the spindle speed may be overridden in the first passes to prevent the part from bending.
 - Withdrawal in rapid to the approach point.
5. Once the operation or cycle has ended, the tool will return to the position it occupied when calling the cycle; i.e. to the point where [START] was pressed.
- When executing an entire part, combination of operations or cycles, the tool does not return to that point after executing each cycle.
6. The CNC stops the spindle, but it keeps the machining conditions set; tool (T), feedrate of the axes (F) and spindle speed (S).

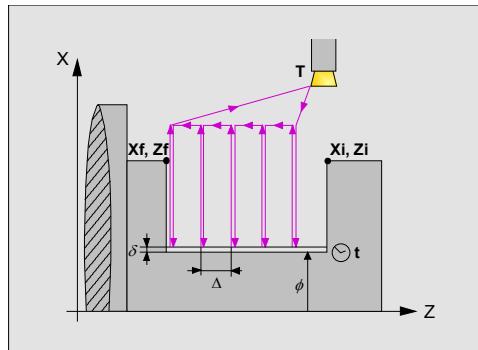
3.9 Grooving cycle



This key accesses the grooving cycles.

This cycle may be used to groove on the cylindrical side of the part or on its face, all of them with vertical walls or incline walls.

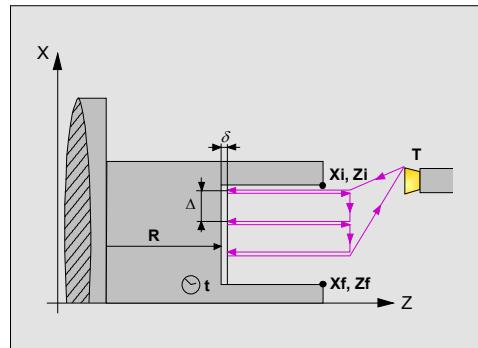
Level 1. Grooving on the cylindrical side.



The following data must be defined:

- The coordinates of the first and last points.
- The final diameter.
- The dwell at the bottom.
- Number of grooves and offset.

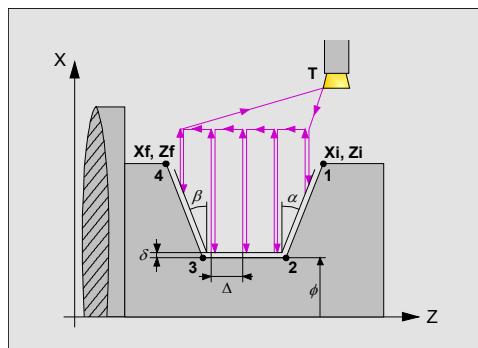
Level 2. Face grooving.



The following data must be defined:

- The coordinates of the first and last points.
- The coordinate of the bottom of the groove.
- The dwell at the bottom.
- Number of grooves and offset.

Level 3. Grooving on the side of the cylinder with incline walls.



3.

WORKING WITH OPERATIONS OR CYCLES
Grooving cycle

FAGOR

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.TC- OPTION
SOFT: V01.4x

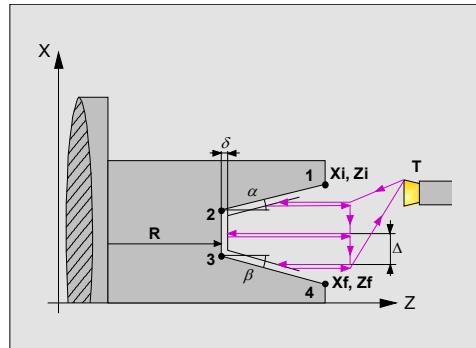
3.

WORKING WITH OPERATIONS OR CYCLES
Grooving cycle

The following data must be defined:

- The coordinates of the first and last points.
- The final diameter.
- The Inclination angles of the incline walls.
- Number of grooves and offset.

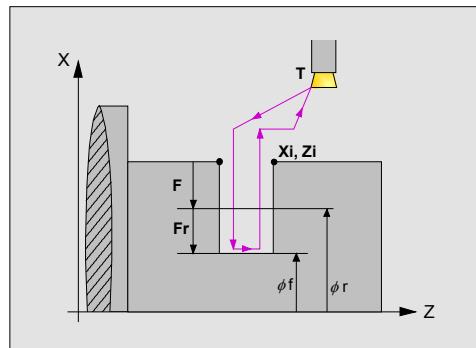
Level 4. Face grooving with incline walls.



The following data must be defined:

- The coordinates of the first and last points.
- The coordinate of the bottom of the groove.
- The Inclination angles of the incline walls.
- Number of grooves and offset.

Level 5. Cut-off.



The following data must be defined:

- Coordinates of the starting point.
- The final diameter of the cut-off.
- The intermediate diameter to decrease the feedrate.
- The cut-off feedrate.



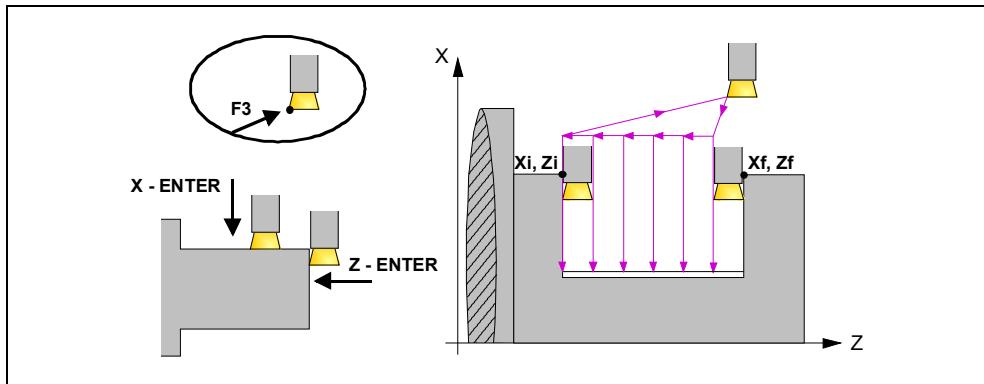
CNC 8037

.TC. OPTION
SOFT: V01.4x

3.9.1 Calibration of the grooving tool

When calibrating the grooving tool, you must properly indicate the location code for the corner that has been calibrated. Thus, the same tool may be calibrated in three different ways, as shown next:

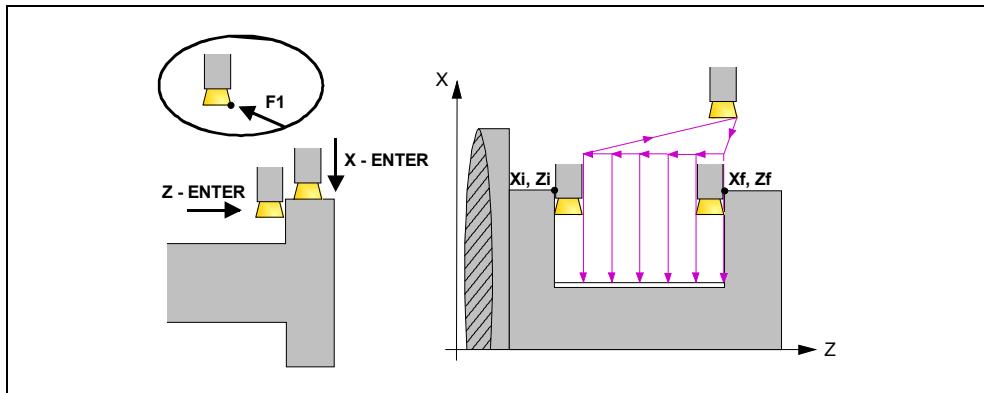
- Location code F3. It calibrates the left corner of the cutter.



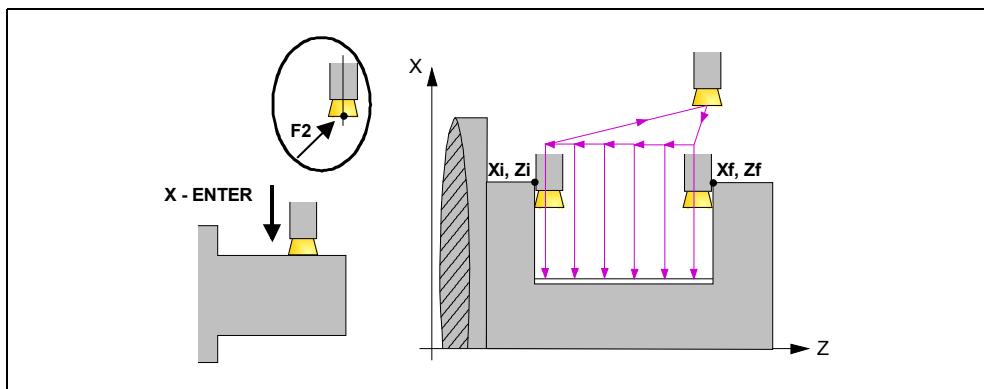
3.

WORKING WITH OPERATIONS OR CYCLES
Grooving cycle

- Location code F1. It calibrates the right corner of the cutter.



- Location code F2. The CNC only calibrates along the X axis and it assumes the center of the cutter as the calibrated point.



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·TC· OPTION
SOFT: V01.4x

3.

3.9.2 Definition of geometry

Type of grooving.

To select the type of grooving, place the cursor over this icon and press the [-] key.



Inside grooving.



Outside grooving.

When changing the type of grooving, the CNC changes the icon and shows the corresponding help screen.

Coordinates of the first point (X_i, Z_i) and of the last point (X_f, Z_f).

The coordinates are defined one by one. After placing the cursor on the coordinates of the axes to be defined, the value is entered in one of the following ways.

- Entering the value manually. Key in the desired value and press [ENTER].
- Assign the current machine position.

Jog the axis with the handwheel or the JOG keys up to the desired point. Press [RECALL] so the selected data assumes the value shown in the top right window and press [ENTER].

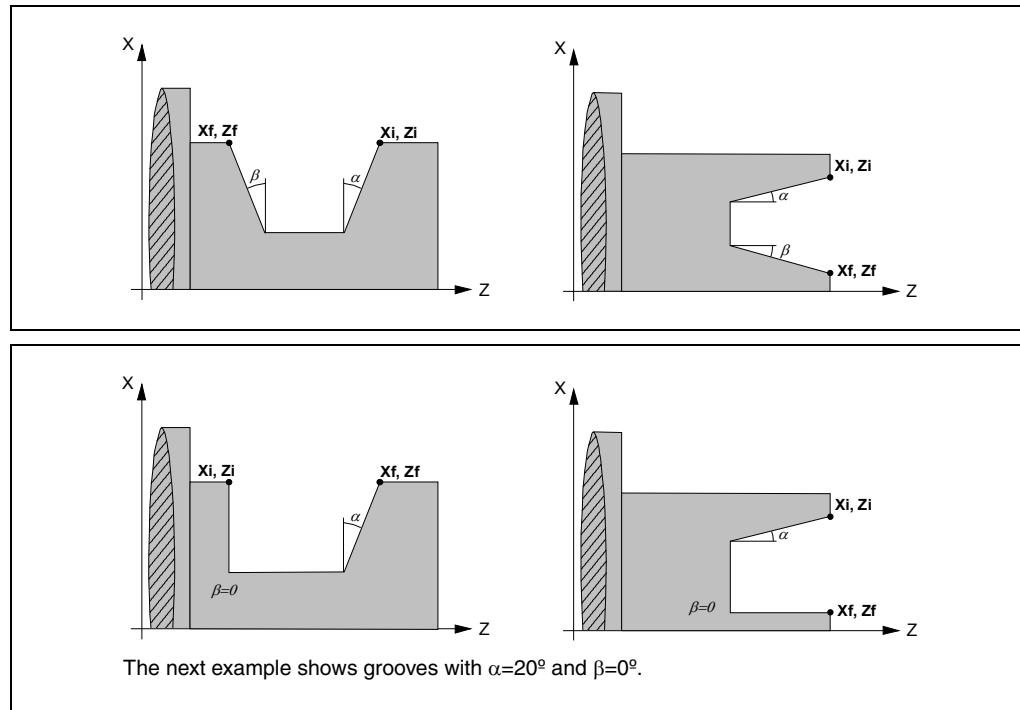
The top right window shows the tool position at all times.

Intermediate diameter (\varPhi_r) and final diameter (\varPhi_f).

For the cut-off operation, you must define the final diameter and an intermediate diameter where start decreasing the feedrate (slowing down). To define these data, key in the desired value and press [ENTER].

Inclination angles (α, β).

These data must be defined when grooving with incline walls. To do this, place the cursor on the data, key in the desired value and press [ENTER].



Type of machining for each corner.

To select the type of corner, place the cursor over this icon and press the [-] key.



Square corner.

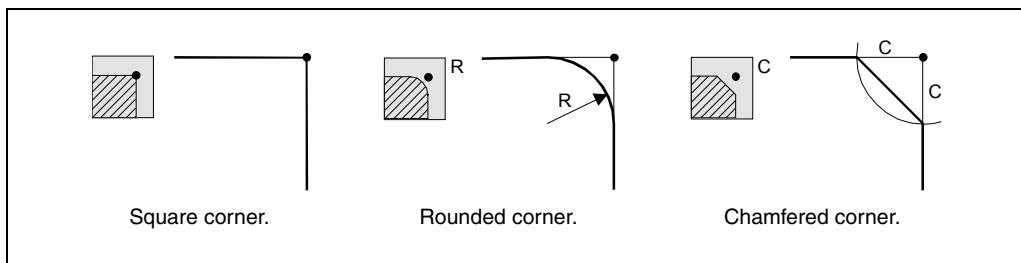


Rounded corner.



Chamfered corner.

These data must be defined when grooving with incline walls. The type of machining to perform must be defined in all four corners of the groove.



For a rounded corner, define the rounding radius (R); for a chamfer, define the distance from the theoretical corner to the chamfer point (C). For cut-off, it is possible to also define the chamfer angle.

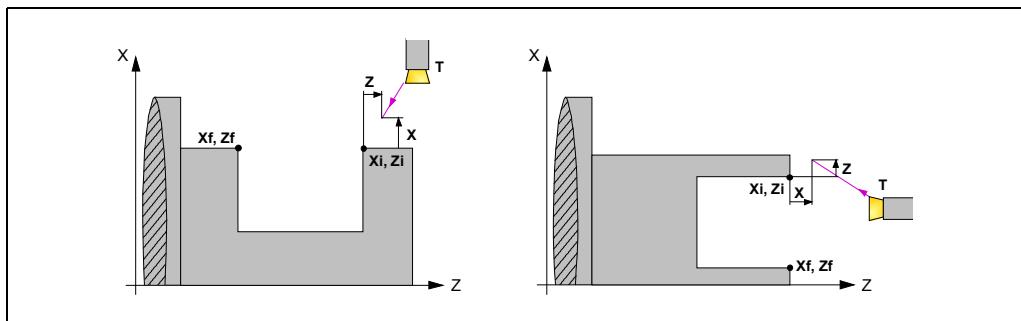
Dwell at the bottom (t).

Defines the dwell, in seconds, after each penetration until the withdrawal begins. To define it, place the cursor on this data, key in the desired value and press [ENTER].

Safety distance.

A part approach point may be set to prevent the tool from colliding with the part. The safety distance indicates the position of the approach point referred to the starting corner.

To change one of these values, place the cursor on the corresponding data, key in the desired value and press [ENTER].



The safety distance in X is always defined in radius.

3.

WORKING WITH OPERATIONS OR CYCLES
Grooving cycle

FAGOR

CNC 8037

.TC- OPTION
SOFT: V01.4x

Type of machining for the roughing pass.

To select the type of grooving, place the cursor over the icon and press the [-] key.

Selection of the starting point of the grooving operation in the center of the groove or at the starting point of the groove:



The roughing process of the grooving starts at the center and goes on in the direction of the starting point. Once the starting point has been reached, it returns to the center and goes on in the direction of the final point until the end of the roughing operation.



The groove roughing process starts at the starting point of the groove.

This option is available in all 4 grooving levels. If zig-zag grooving is defined at levels 3 and 4, the walls of the groove are not vertical enough, they will not be machined in zig-zag.

Selection of the penetration step (pass) P and dwell t (in seconds) for chip removal in the first grooving pass:



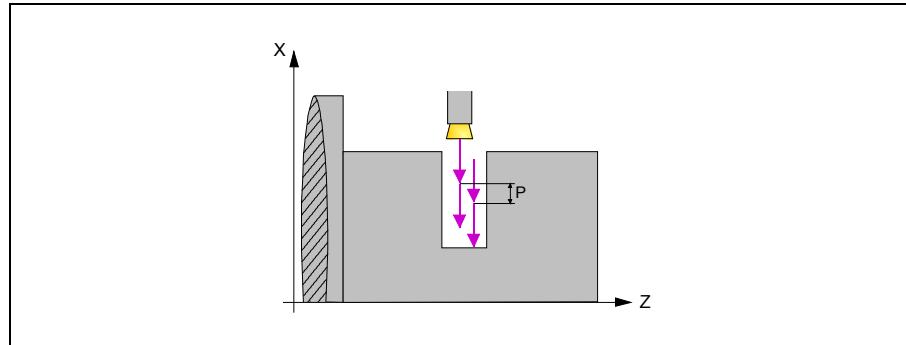
Selection of the penetration step (pass) to keep the tool from overheating and dwell for chip removal after each pass.

This penetration in steps takes place during the first penetration and it is valid regardless of the selected starting point (center or corner).

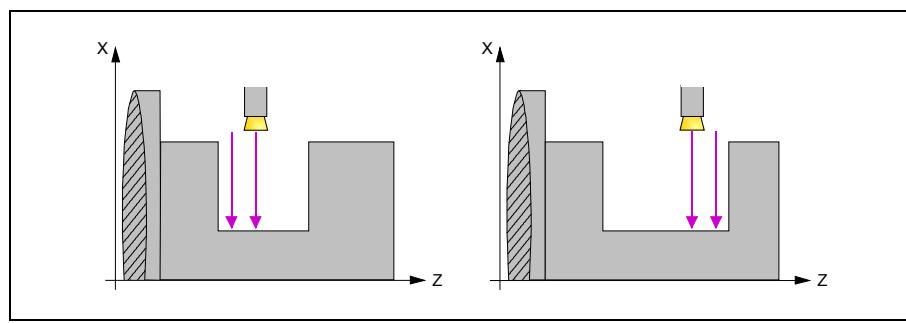
Selection of the zig-zag grooving for chip removal in the first grooving pass:



The chip relief selecting icon offers the zig-zag grooving option.



In the first pass, it machines in zig-zag with penetration according to parameter P.



It runs the rest of the passes machining down to the bottom. In order to take care of the tool, first machine the center area towards the origin with one side of the cutter (left figure) and then from the center to the opposite side with the other side of the cutter (right figure).

In any of the grooving options, even if a dwell has not been programmed in the last roughing passes, a dwell will be applied in order to machine the whole surface of the part with a uniform radius. The length of this delay will be the time it takes the spindle to rotate a full revolution.

3.

WORKING WITH OPERATIONS OR CYCLES

Grooving cycle

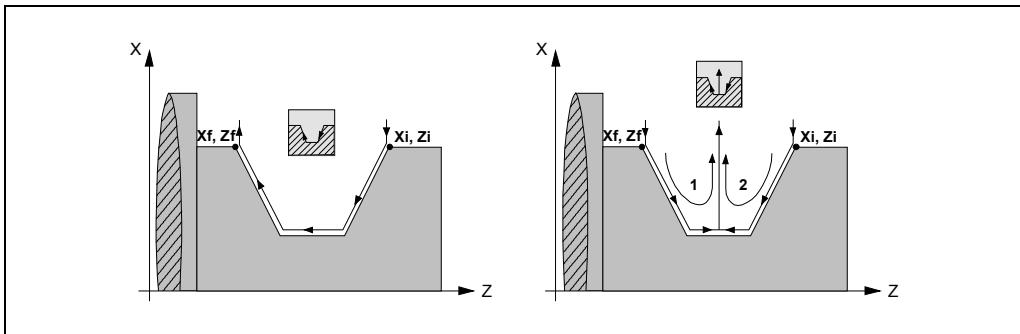
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 ·TC· OPTION
 SOFT: V01.4x

Type of machining for the finishing pass.

This data must be defined when grooving with incline walls. To select the type of machining operation, place the cursor over this icon and press the [-] key.



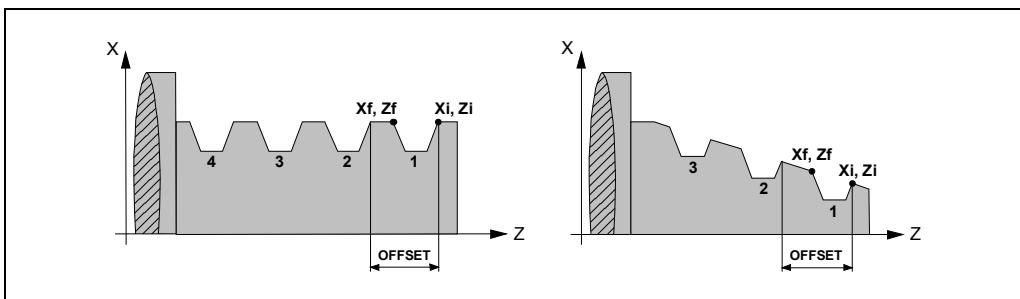
3.

WORKING WITH OPERATIONS OR CYCLES
Grooving cycle

Groove repetition.

The "number of grooves" and "offset" data may be used to repeat a groove several times along the Z axis in cylindrical grooving or along the X axis in face grooving.

If the first groove is tapered, Xi different from Xf, that taper is maintained for the rest of the grooves.



If the number of grooves is defined with a value of 0 or 1, it only makes one groove.

Intermediate diameter (ϕr) and feedrate (F_r).

For the cut-off operation, you must define an intermediate diameter where to start machining at low feedrate (F_r) until the end of the cut-off operation. To define these data, key in the desired value and press [ENTER].



CNC 8037

.TC- OPTION
SOFT: V01.4x

3.

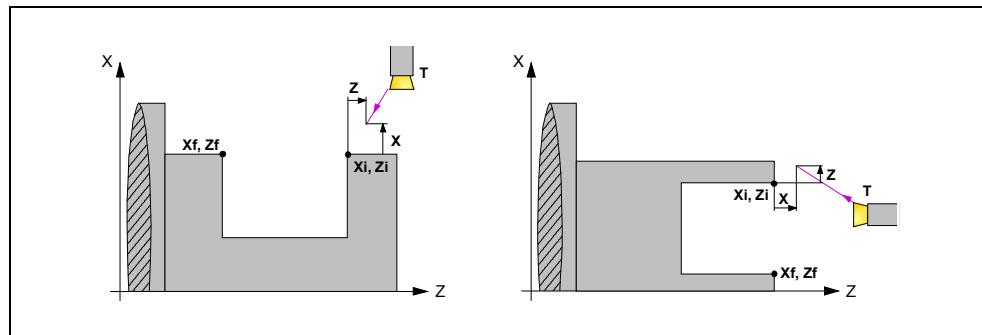
WORKING WITH OPERATIONS OR CYCLES

Grooving cycle

3.9.3 Basic operation. Slot milling

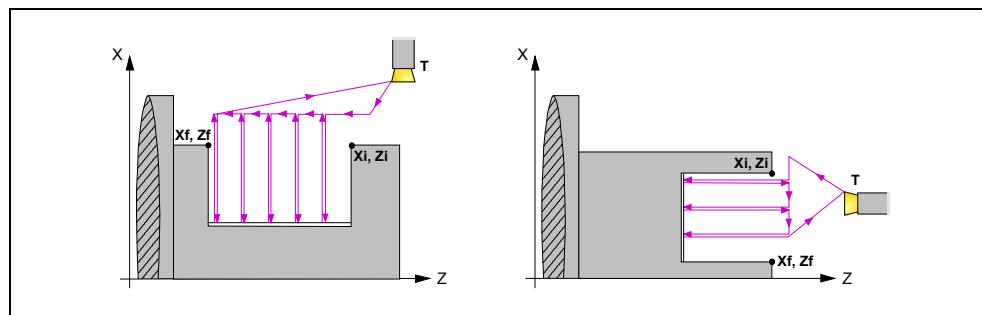
The machining steps for this cycle are:

1. If the 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.
2. The spindle starts at the selected speed and in the indicated direction.
3. The tool approaches in rapid to the starting point (X_i, Z_i) keeping the selected safety distance in X and Z.



4. Roughing operation, with successive grooving passes, up to a distance from the final depth equal to the finishing stock.

This operation is carried out with the conditions set for the roughing operations; however, the CNC calculates the actual (real) pass so all the facing passes are identical. This pass will be equal to or under the defined value Δ .

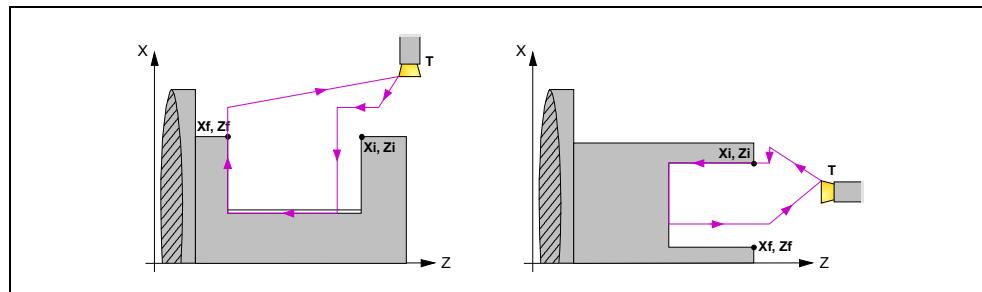


Each machining pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".

5. Finishing operation.

If 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 finishing of the part is carried out with the machining conditions set for finishing; feedrate of the axes (F), spindle speed (S) and turning direction.



6. Once the operation or cycle has ended, the tool will return to the position it occupied when calling the cycle; i.e. to the point where [START] was pressed.

When executing an entire part, combination of operations or cycles, the tool does not return to that point after executing each cycle.

7. The CNC stops the spindle, but it keeps the machining conditions set; tool (T), feedrate of the axes (F) and spindle speed (S).

Considerations

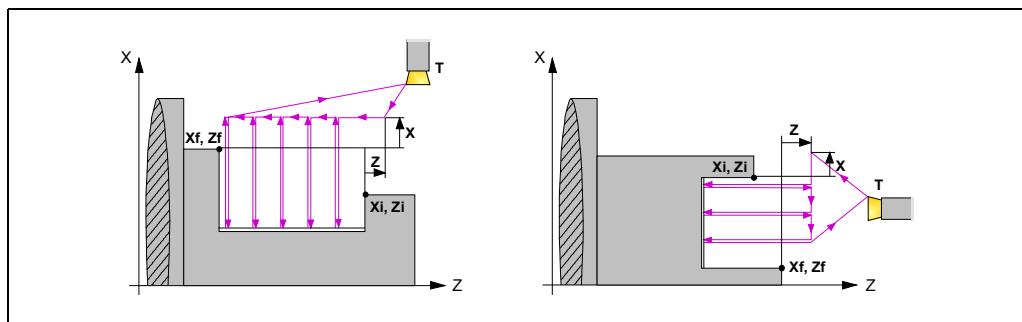
How to leave out the roughing or finishing operations.

By selecting T0 as the roughing tool, the cycle does not execute the roughing operation. In other words, after the approach movement, it will carry out the finishing operation.

By selecting T0 as the finishing tool, the cycle does not execute the finishing operation. In other words, after the roughing operation, the tool will move to the approach point maintaining the safety distance from the starting point (Xi, Zi).

Different Xi and Xf coordinates.

When the surface to be machined is not totally cylindrical (different Xi and Xf coordinates), the CNC analyzes both coordinates and takes as the X coordinate of the starting point the one farthest away from the final depth.



3.

WORKING WITH OPERATIONS OR CYCLES
Grooving cycle

FAGOR 

CNC 8037

.TC- OPTION
SOFT: V01.4x

3.9.4 Basic operation. Cut-off

The machining steps for this cycle are:

1. If the 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.
2. The spindle starts at the selected speed and in the indicated direction.
3. The tool approaches in rapid to the starting point (X_i, Z_i) keeping the selected safety distance in X and Z.
4. Cut-off operation, at feedrate F, up to the intermediate diameter. From this point on, the cut-off operation continues at feedrate F up to the final diameter.
5. Once the operation or cycle has ended, the tool will return to the position it occupied when calling the cycle; i.e. to the point where [START] was pressed.
When executing an entire part, combination of operations or cycles, the tool does not return to that point after executing each cycle.
6. The CNC stops the spindle, but it keeps the machining conditions set; tool (T), feedrate of the axes (Fr) and spindle speed (S).

3.

WORKING WITH OPERATIONS OR CYCLES

Grooving cycle



CNC 8037

.TC. OPTION
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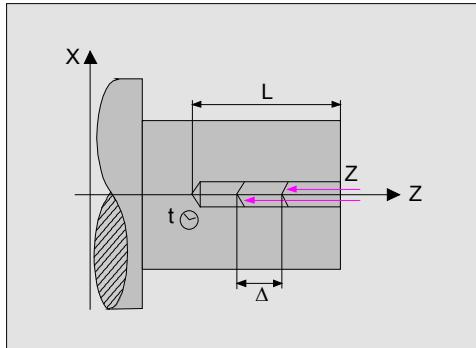
3.10 Drilling and tapping cycles



This key accesses the drilling and tapping cycles.

- Drilling cycle.
- Tapping cycle.

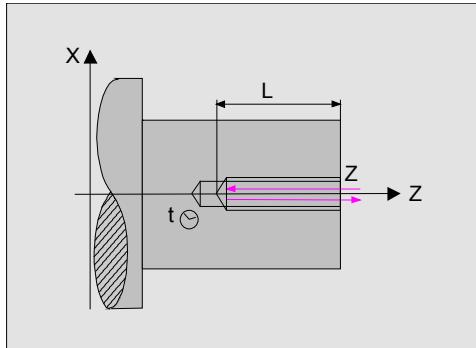
Level 1. Drilling cycle.



The following data must be defined:

- Coordinates of the drilling point.
- The total depth.
- The dwell at the bottom.

Level 2. Tapping cycle.



The following data must be defined:

- Z coordinate of the tapping point.
- The total depth.
- The dwell at the bottom.

3.

WORKING WITH OPERATIONS OR CYCLES
Drilling and tapping cycles

FAGOR 

CNC 8037

.TC- OPTION
SOFT: V01.4x

3.

3.10.1 Definition of geometry

Machining on the face of the part.



To select the type of machining operation, place the cursor over this icon and press the [-] key.

Coordinates of the starting point (X, Z).

The coordinates are defined one by one. After placing the cursor on the coordinates of the axes to be defined, the value is entered in one of the following ways.

- Entering the value manually. Key in the desired value and press [ENTER].
- Assign the current machine position.

Jog the axis with the handwheel or the JOG keys up to the desired point. Press [RECALL] so the selected data assumes the value shown in the top right window and press [ENTER].

The top right window shows the tool position at all times.

Both tapping and drilling must always be axial, at the rotation center (X0).

Total depth (L).

Place the cursor on this data, key in the desired value and press [ENTER].

Withdrawal distance (H) and approach distance (C).

Parameter H defines the distance for the tool to withdraw after each drilling peck. Parameter C defines the distance from the previous drilling peck where the tool approaches in rapid. To define these parameters, key in the desired value and press [ENTER].

Dwell at the bottom (t).

Defines the dwell, in seconds, after the drilling or threading until the withdrawal begins. To define it, place the cursor on this data, key in the desired value and press [ENTER].

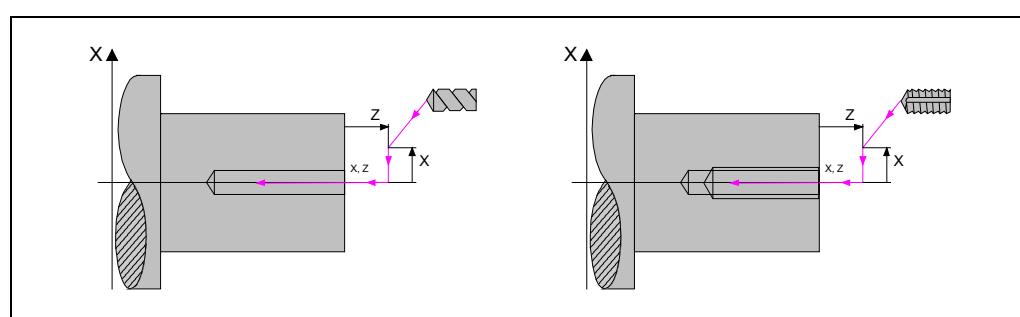
Penetrating feedrate (F).

Place the cursor on this data, key in the desired value and press [ENTER].

Safety distance.

A part approach point may be set to prevent the tool from colliding with the part. The safety distance indicates the position of the approach point referred to the drilling or threading point.

To change one of these values, place the cursor on the corresponding data, key in the desired value and press [ENTER].

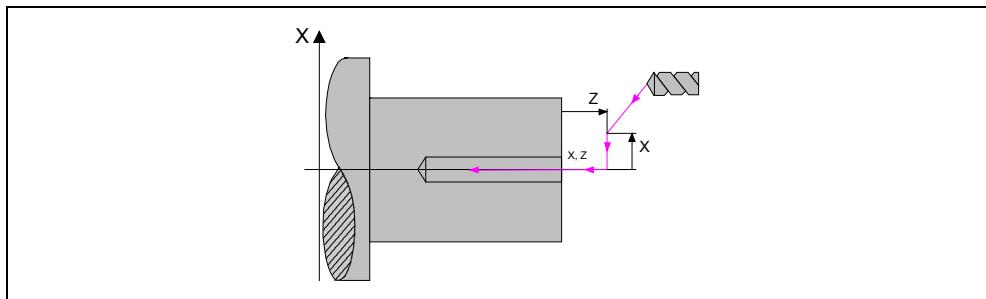


The safety distance in X is always defined in radius.

3.10.2 Drilling cycle. Basic operation

The machining steps for this cycle are:

1. If the 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.
2. The spindle starts at the selected speed and in the indicated direction.
3. The tool approaches in rapid to the starting point (X_i, Z_i) keeping the selected safety distance in X and Z.



4. Drilling loop. The steps are repeated, penetrating each time the amount indicated by $K\Delta$ and Δ , up to the depth L. Δ defines the drilling peck and $K\Delta$ the reduction factor of that peck.
 - The first drilling peck will be Δ , the second one " $K\Delta \Delta$ ", The third one " $K\Delta (K\Delta \Delta)$ " and so on; in other words, from the second peck on, the new peck will be the product of the $K\Delta$ factor times the previous peck.
 - After each drilling peck, the tool withdraws a distance "H". Then, it approaches in rapid up to a "C" distance from the previous penetration; if "C" has not been defined, the tool approaches up to 1 mm from the previous penetration.
 - Drilling down to the next penetration.
 - Withdrawal in rapid to the approach point.
5. Dwell "t" at the bottom of the hole.
6. Withdrawal in rapid to the approach point.
7. Once the operation or cycle has ended, the tool will return to the position it occupied when calling the cycle; i.e. to the point where [START] was pressed.
When executing an entire part, combination of operations or cycles, the tool does not return to that point after executing each cycle.
8. The CNC stops the spindle, but it keeps the machining conditions set; tool (T), feedrate of the axes (F) and spindle speed (S).

3.

WORKING WITH OPERATIONS OR CYCLES

Drilling and tapping cycles

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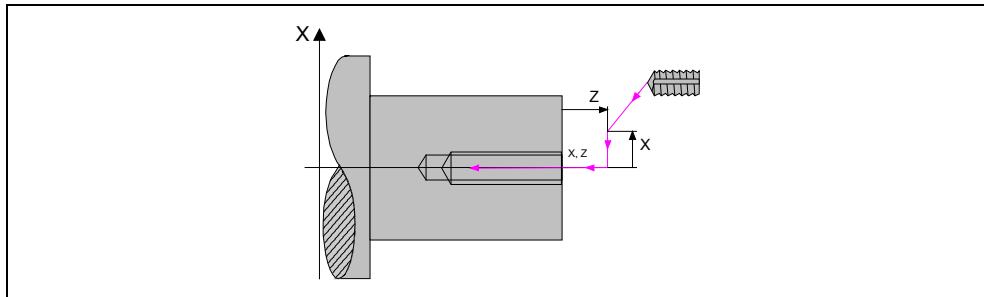
3.**WORKING WITH OPERATIONS OR CYCLES**

Drilling and tapping cycles

3.10.3 Tapping cycle. Basic operation

The machining steps for this cycle are:

1. If the 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.
2. The spindle starts at the selected speed and in the indicated direction.
3. The tool approaches in rapid to the starting point (X_i, Z_i) keeping the selected safety distance in X and Z.



4. Threading of the part at work feedrate F up to the "L" depth.
 5. Reverse the spindle turning direction.
- If a dwell has been defined for the bottom of the hole, the spindle stops and after the programmed dwell has elapsed, the spindle starts turning in the other way.
6. Withdrawal in work feedrate to the approach point.
 7. Once the operation or cycle has ended, the tool will return to the position it occupied when calling the cycle; i.e. to the point where [START] was pressed.
- When executing an entire part, combination of operations or cycles, the tool does not return to that point after executing each cycle.
8. The CNC stops the spindle, but it keeps the machining conditions set; tool (T), feedrate of the axes (F) and spindle speed (S).

General logic output "TAPPING" (M5517) stays active while executing the cycle.



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.TC. OPTION
SOFT: V01.4x

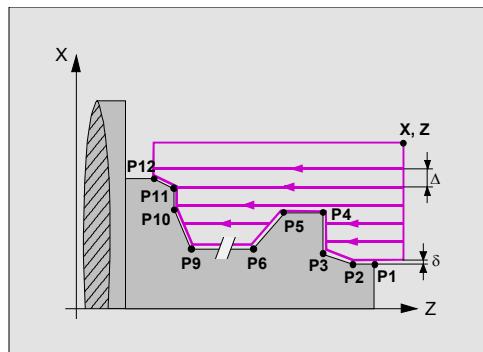
3.11 Profiling cycle



This key accesses the profiling cycles.

This cycle may be defined in several ways.

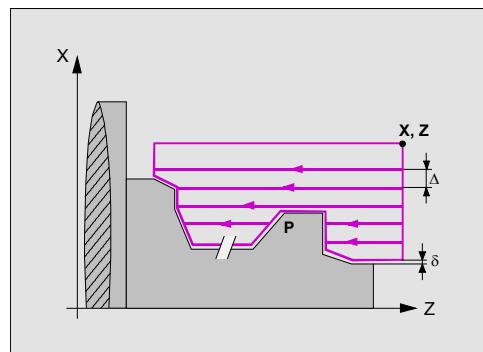
Level 1. Defining all the points of the profile.



3.

WORKING WITH OPERATIONS OR CYCLES
Profiling cycle

Level 2. Using a part-program that contains a profile.



FAGOR

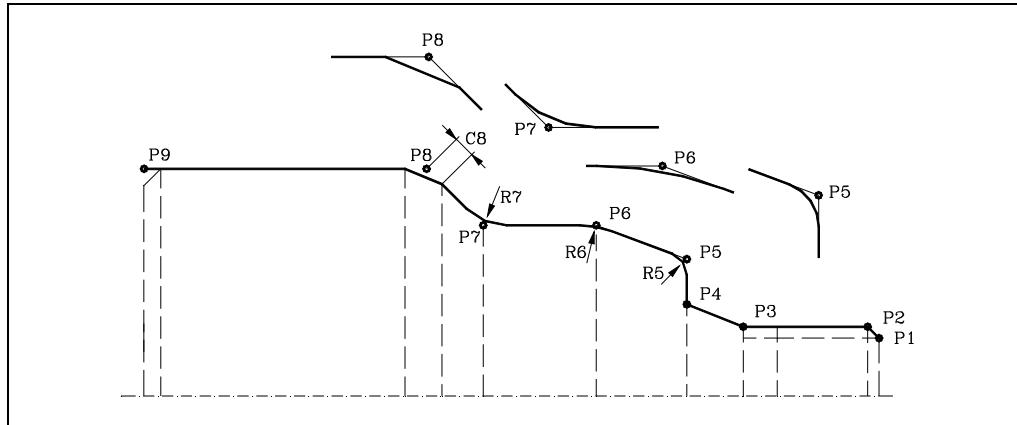
CNC 8037

.TC- OPTION
SOFT: V01.4x

3.

3.11.1 Level 1. Profile definition

This mode may be used to define the profile by describing its theoretical corners. Up to 12 points may be used in the cycle to define those corners. Point P1 is the starting point of the profile. The rest of the points must be sequential.



Use the [\downarrow] [\uparrow] keys to select and exit the window that contains the profile defining points and the [\leftarrow] [\rightarrow] keys to define those points.

Coordinates of the first and last points.

The coordinates of each point are defined one by one. After placing the cursor on the coordinate to be defined, the value is entered in one of the following ways.

- Entering the value manually. Key in the desired value and press [ENTER].
- Assign the current machine position.

Jog the axis with the handwheel or the JOG keys up to the desired point. Press [RECALL] so the selected data assumes the value shown in the top right window and press [ENTER].

The top right window shows the tool position at all times.

Intermediate points.

The intermediate points are defined one by one. If a coordinate is left blank, the cycle will assume that it is the same as that of the previous one. When not using all 12 definition points, the following conditions must be met:

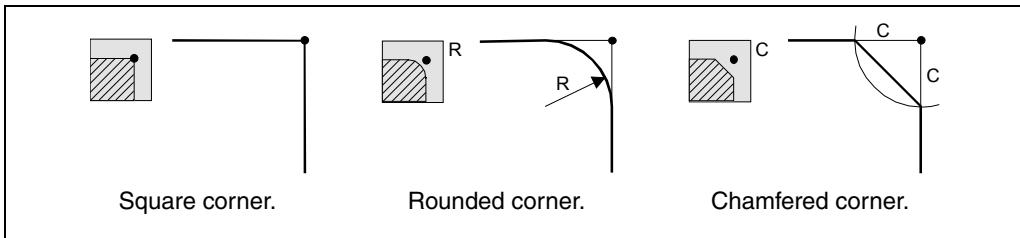
- The CNC ignores the machining type of the last point of the profile.
- The first unused point must be defined with the same coordinates as the last point of the profile. P10 must be set equal to P9 in the example of the figure above.

If defined...	The CNC assumes...
X1 25.323	Z1 26.557 Point: X1 25.323 Z1 26.557
X2	Z2 78.998 Point: X2 25.323 Z2 78.998
X3 67.441	Z3 83.231 Point: X3 67.441 Z3 83.231
X4	Z4 Point: X4 67.441 Z4 83.231
X5	Z5 There are no more points, it is a repetition of the previous point.

The coordinates of each point may also be defined incrementally. To do that, select the desired coordinate with the cursor and press the [INS] key. Both coordinates of the selected point will be shown preceded by the " Δ " icon that indicates the incremental value with respect to the previous point.



The type of machining to perform at the corner must be defined in all the intermediate points of the profile. To select the type of corner operation, place the cursor over this icon and press the [-] key.



For a rounded corner, define the rounding radius (R); for a chamfer, define the distance from the theoretical corner to the chamfer point (C).

Deleting all the points of a profile.

Once all the desired points have been programmed, they may all be erased at once. To delete all the programmed points at the same time, proceed as follows:

- Place the cursor on the text "DEF. PROFILE (max 12 points)" of the window where the points are edited.
- In the instant the cursor is placed in this position, the text changes to: "CLEAR - Delete all the points".
- At this moment, pressing [CLEAR] displays a window requesting confirmation to delete all the points: Press the [ENTER] to delete all the points or [ESC] not to delete them.

3.

WORKING WITH OPERATIONS OR CYCLES
Profiling cycle

FAGOR 

CNC 8037

.TC- OPTION
SOFT: V01.4x

3.11.2 Level 2. Profile definition

Defining the profile program.

The profile program may be defined as follows.

- Key in the profile program number directly.
If the "profile program" is known, key in the program number and press [ENTER].
- Access the "profile programs" to select one of them.
Press the [**↓**] key. The canned cycle will show a window with the profile programs currently defined. Use the [**↑**] and [**↓**] keys to move around this window. After positioning the cursor on the desired program, press [ENTER].
To exit this window without selecting any program, use the [**←**] y [**→**] keys.

Editing a new profile program.

To edit a new profile program, key in the program number (between 0 and 999) and press [RECALL]. The CNC will show the window for the profile editor (see operating manual).

Once the profile has been edited, the CNC requests the comment to be associated with the profile program being edited. Enter the desired comment and press [ENTER]. If no comment is desired, press [ESC].

Copying a profile program.

Press the [**↓**] key. The canned cycle will show the profile programs currently defined. Place the cursor on the profile program to be copied and press [EDIT]. The CNC requests the number of the new profile and it allows changing its comment. If the number entered is the same as that of an existing profile, the CNC requests confirmation to replace it.

Modifying an existing profile program.

To modify a profile program, key in the program number and press [RECALL]. The CNC will show, in the window for the profile editor, the profile that is currently defined.

With this profile, the following actions are possible:

- Adding new elements at the end of the current profile.
- Modifying the data of any element.
- Modifying or including chamfers, rounding, etc.
- Deleting elements of the profile.

Deleting an existing profile program.

Press the [**↓**] key. The canned cycle will show the profile programs currently defined. Place the cursor on the profile program to be deleted and press [CLEAR]. The CNC requests confirmation.

Notes

The profile programs can also be accessed in the "T" mode because the CNC saves them internally as:

P998xxx (ZX profile, level 2).

Profile program 11 is saved as P998011.

P997xxx (ZC profile, level 3).

Profile program 22 is saved as P997022.

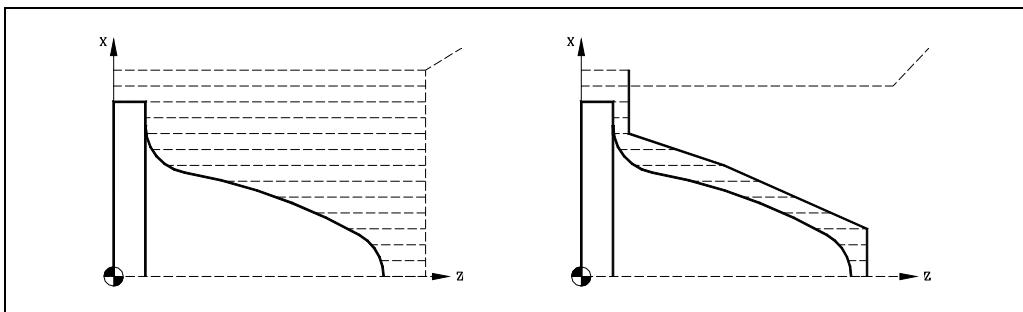
P996xxx (XC profile, level 4).

Profile program 33 is saved as P996033.

When saving a part-program that contains a profile cycle into an external device, also save its associated profile cycle (P998xxx, P997xxx, P996xxx).

3.11.3 Level 2. Optimizing of the machining of a profile

If only the desired profile is defined, the CNC assumes that the original stock is cylindrical and machines it as shown on the left.



When the stock profile is known, it is recommended to define both profiles: The profile of the original stock and the desired final profile. The machining operation is much faster because only the stock delimited by both profiles is removed.

Follow this order to define both profiles:

1. Access the profile editor.
2. Edit the desired final profile.
3. Press the softkey for a new profile.
4. Edit the profile of the original stock.
5. Quit the profile editor saving the profile.

Remember that the desired final profile must be defined first and then the profile of the original stock.

3.

WORKING WITH OPERATIONS OR CYCLES
Profiling cycle

FAGOR 

CNC 8037

.TC- OPTION
SOFT: V01.4x

3.11.4 Definition of geometry levels 1 and 2. ZX profile

Outside or inside profile.



To select the type of profile, place the cursor over this icon and press the [-] key.



Inside profile.



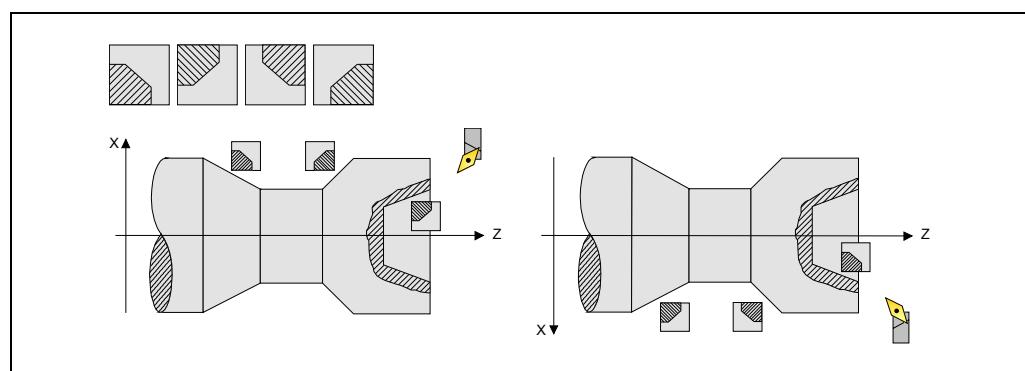
Outside profile.

When changing the type of profile, the CNC changes the icon and shows the corresponding help screen.

Work quadrant.



The work quadrant is defined with the following icons: To select the desired type, place the cursor over this icon and press the [-] key.



Tool of machining.



The type of machining is defined with the following icons: To select the desired type, place the cursor over this icon and press the [-] key.

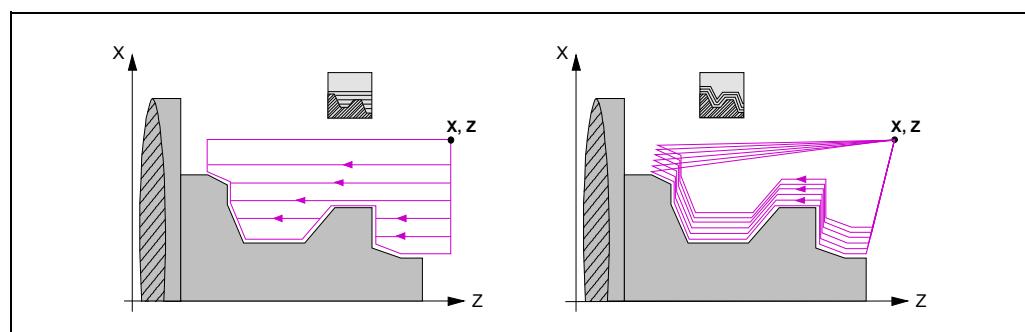


Paraxial machining (one axis at a time).



Machining following the profile

When changing the type of machining, the CNC changes the icon and shows the corresponding help screen.



In paraxial machining you must define the feedrate (F) for the tool to penetrate into the valleys. The machining feedrate will be the one indicated in the roughing and finishing windows.

In pattern repeat, the amount of stock to be removed from the original stock (ϵ) must be defined. That value is defined in radius.

3.

WORKING WITH OPERATIONS OR CYCLES
Profiling cycle

FAGOR

CNC 8037

.TC. OPTION
SOFT: V01.4x

Coordinates of the starting point (X, Z).

The coordinates are defined one by one. After placing the cursor on the coordinates of the axes to be defined, the value is entered in one of the following ways.

- Entering the value manually. Key in the desired value and press [ENTER].
- Assign the current machine position.

Jog the axis with the handwheel or the JOG keys up to the desired point. Press [RECALL] so the selected data assumes the value shown in the top right window and press [ENTER].

The top right window shows the tool position at all times.

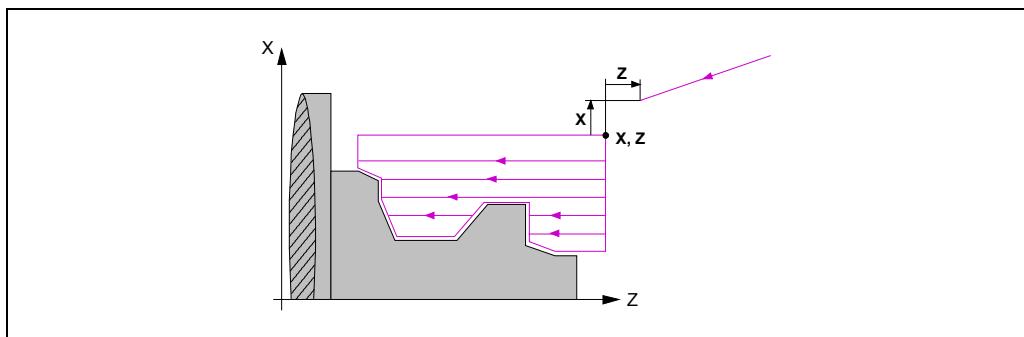
3.

WORKING WITH OPERATIONS OR CYCLES
Profiling cycle

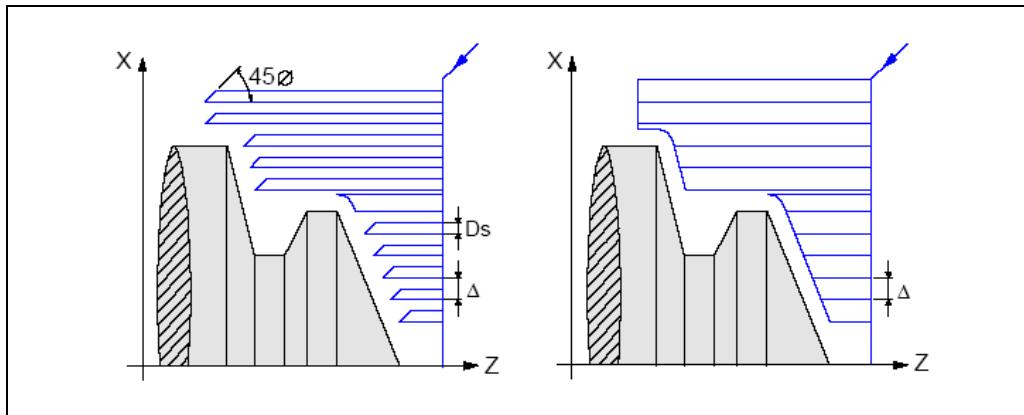
Safety distance.

A part approach point may be set to prevent the tool from colliding with the part. The safety distance indicates the position of the approach point referred to the starting corner.

To change one of these values, place the cursor on the corresponding data, key in the desired value and press [ENTER].

***Withdrawal distance at 45° (Ds).***

The tool can withdraw to a safety distance after each pass. If the "Ds" data is validated, the display graphic changes and there is a withdrawal. This withdrawal is done at 45° (left figure). The right figure shows a standard withdrawal without a 45° angle.



FAGOR 

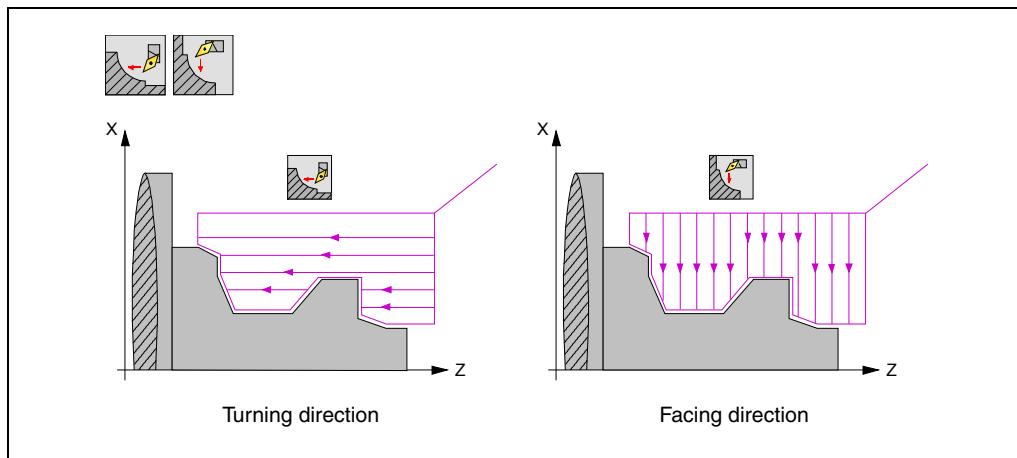
CNC 8037

.TC- OPTION
SOFT: V01.4x

3.

Machining direction.

The machining direction (turning direction or facing direction) is defined with the following icons. To select the desired type, place the cursor over this icon and press the [-] key.

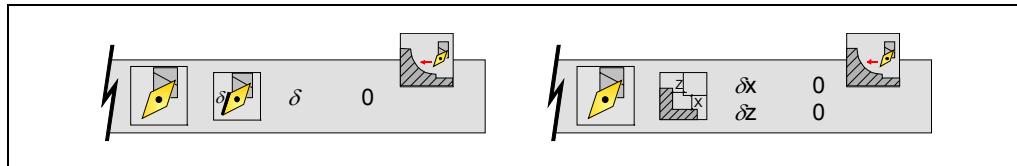


When changing the machining direction, the CNC changes the icon and shows the corresponding help screen.

Finishing stocks in X-Z.

Either a single residual stock may be defined that is applied depending on the cutter's edge or 2 different ones, one for each axis (X, Z). It is selected with the following icon located in the finishing area.

- The figure on the left applies a finishing stock depending on the cutter's edge. The finishing stock is measured on the tool's cutting edge.
- The figure on the right lets define 2 finishing stocks, one per axis, regardless of the type of tool used.

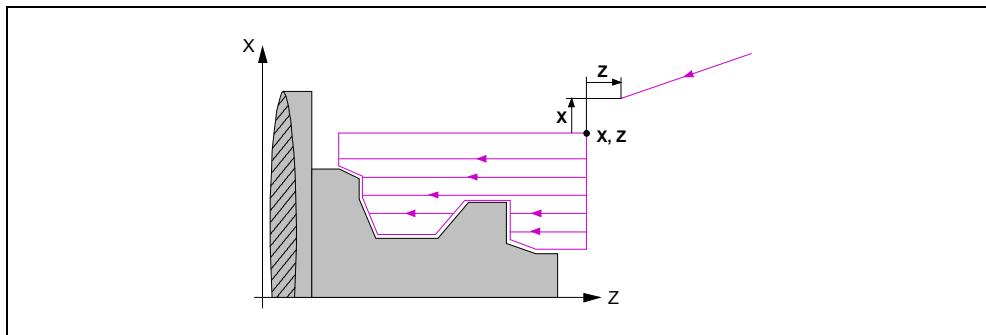


To select the type of residual stock, place the cursor over this icon and press the [-] key. To define these residual stocks, place the cursor on the corresponding data, key in the desired value and press [ENTER].

3.11.5 Basic operation at levels 1 and 2. ZX profile

The machining steps for this cycle are:

1. If the 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.
2. The spindle starts at the selected speed and in the indicated direction.
3. The tool approaches in rapid to the starting point (X, Z) keeping the selected safety distance in X and Z.



4. Roughing operation, with successive passes, up to a distance from the profile equal to the finishing stock.

This operation is carried out under the conditions set for the roughing operation.

5. Finishing operation.

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 finishing of the part is carried out with the machining conditions set for finishing; feedrate of the axes (F), spindle speed (S) and turning direction.

6. Once the operation or cycle has ended, the tool will return to the position it occupied when calling the cycle; i.e. to the point where [START] was pressed.

When executing an entire part, combination of operations or cycles, the tool does not return to that point after executing each cycle.

7. The CNC stops the spindle, but it keeps the machining conditions set; tool (T), feedrate of the axes (F) and spindle speed (S).

3.

WORKING WITH OPERATIONS OR CYCLES
Profiling cycle

Considerations

How to leave out the roughing or finishing operations.

By selecting T0 as the roughing tool, the cycle does not execute the roughing operation. In other words, after the approach movement, it will carry out the finishing operation.

By selecting T0 as the finishing tool, the cycle does not execute the finishing operation. In other words, after the roughing operation, the tool will move to the approach point maintaining the safety distance from the starting point (X, Z).



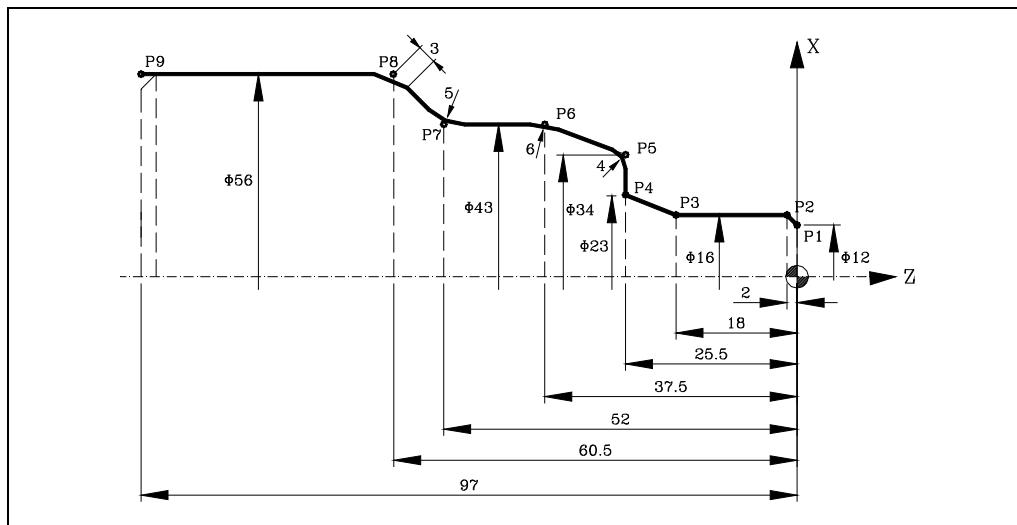
CNC 8037

.TC- OPTION
SOFT: V01.4x

3.11.6 Example. Level 1

3.

WORKING WITH OPERATIONS OR CYCLES
Profiling cycle



Geometry definition.



Outside profile.

Tool of machining.

Work quadrant.

Profile definition.

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

Coordinates (X, Z).

X 80.0000 Z 10.0000

Safety distance.

X 0.0000 Z 0.0000

Roughing.

F 1.000 S 1000 T 3 Δ 2



Finishing.

F 0.800 S 1000 T 3 δ 0.25

Spindle.

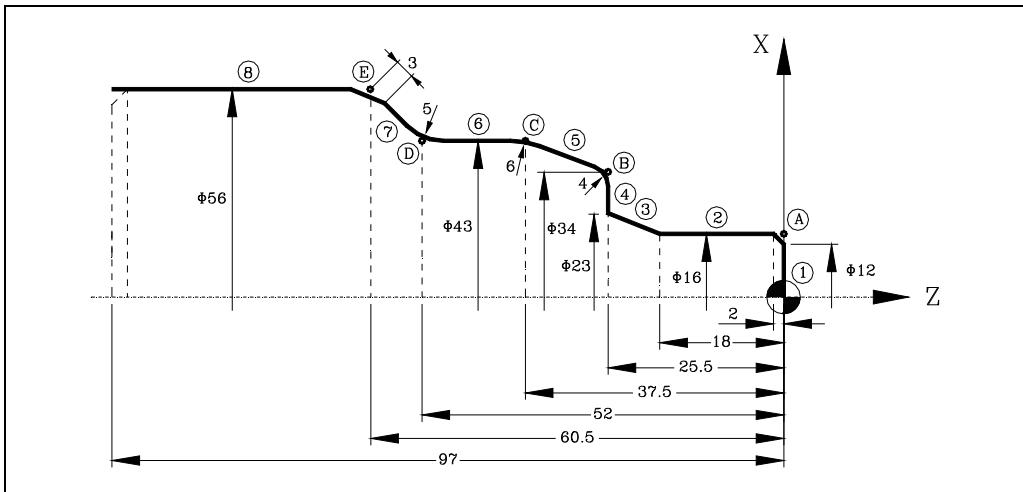
RPM

.TC. OPTION
SOFT: V01.4x

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3.11.7 Examples. Level 2



3.

WORKING WITH OPERATIONS OR CYCLES
Profiling cycle

Geometry definition.

Outside profile.

Tool of machining.

Work quadrant.

Profile definition.

Abscissa and ordinate of the starting point.	Z = 0	X = 0
Section 1	Straight	Z = 0 X = 16
Section 2	Straight	Z = -18 X = 16
Section 3	Straight	Z = -25.5 X = 23
Section 4	Straight	Z = -25.5 X = 34
Section 5	Straight	Z = -37.5 X = 43
Section 6	Straight	Z = -52 X = 43
Section 7	Straight	Z = -60.5 X = 56
Section 8	Straight	Z = -97 X = 56

Modify.

Chamfer	Select point "A". Press [ENTER] and assign to it Radius = 2.
Rounding	Select point "B". Press [ENTER] and assign to it Radius = 4.
Rounding	Select point "C". Press [ENTER] and assign to it Radius = 6.
Rounding	Select point "D". Press [ENTER] and assign to it Radius = 5.
Chamfer	Select point "E". Press [ENTER] and assign to it Radius = 3.

Coordinates (X, Z).

X 65.0000 Z 10.0000

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.TC- OPTION
SOFT: V01.4x

Safety distance.

X 0.0000 Z 0.0000

Roughing.

F 1.000 S 1000 T 3 Δ 2

***Finishing.***

F 0.800 S 1000 T 3 δ 0.25

Spindle.

RPM

**3.**

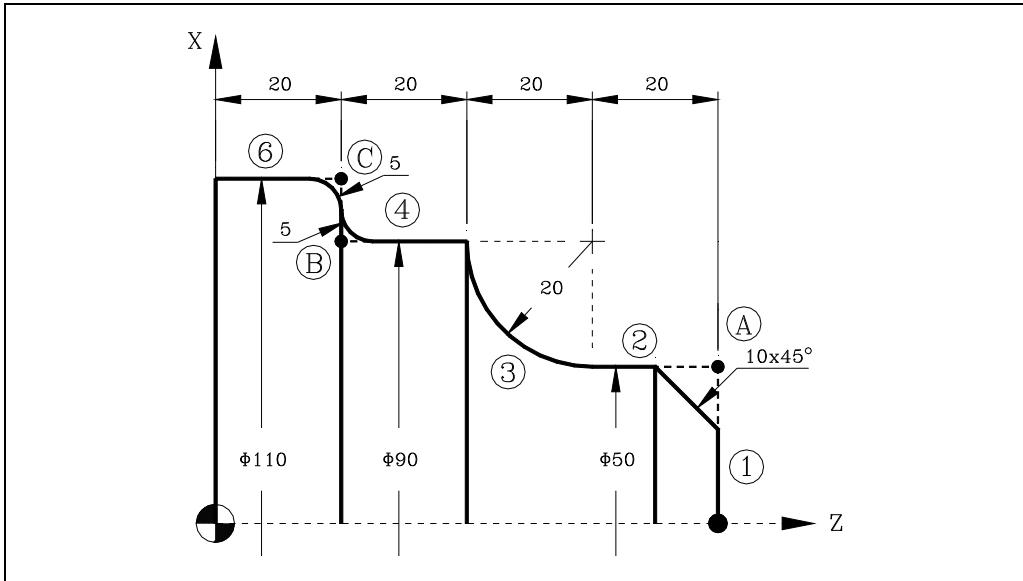
WORKING WITH OPERATIONS OR CYCLES

Profiling cycle

FAGOR 

CNC 8037

·TC· OPTION
SOFT: V01.4x



3.

WORKING WITH OPERATIONS OR CYCLES

Profiling cycle

Geometry definition.



- Outside profile.
 - Tool of machining.
 - Work quadrant.

Profile definition.

Abscissa and ordinate of the starting point.				Z = 80	X = 0
Section 1	Straight	Z = 80	X = 50		
Section 2	Straight	Z = 60	X = 50		
Section 3	Clockwise arc	Z = 40	X = 90	Zc = 60	Xc = 90 R = 20
Section 4	Straight	Z = 20	X = 90		
Section 5	Straight	Z = 20	X = 110		
Section 6	Straight	Z = 0	X = 110		

Modify.

Chamfer	Select point "A". Press [ENTER] and assign to it Radius = 10.
Rounding	Select point "B". Press [ENTER] and assign to it Radius = 5.
Rounding	Select point "C". Press [ENTER] and assign to it Radius = 5.

Coordinates (X, Z).

X 120.0000 Z 90.0000

Safety distance.

X 0.0000 Z 0.0000

FAGOR 

CNC 8037

·TC· OPTION
SOFT: V01.4X

Roughing.

F 1.000 S 1000 T 3 Δ 2

**Finishing.**

F 0.800 S 1000 T 3 δ 0.25

Spindle.

RPM

**3.**

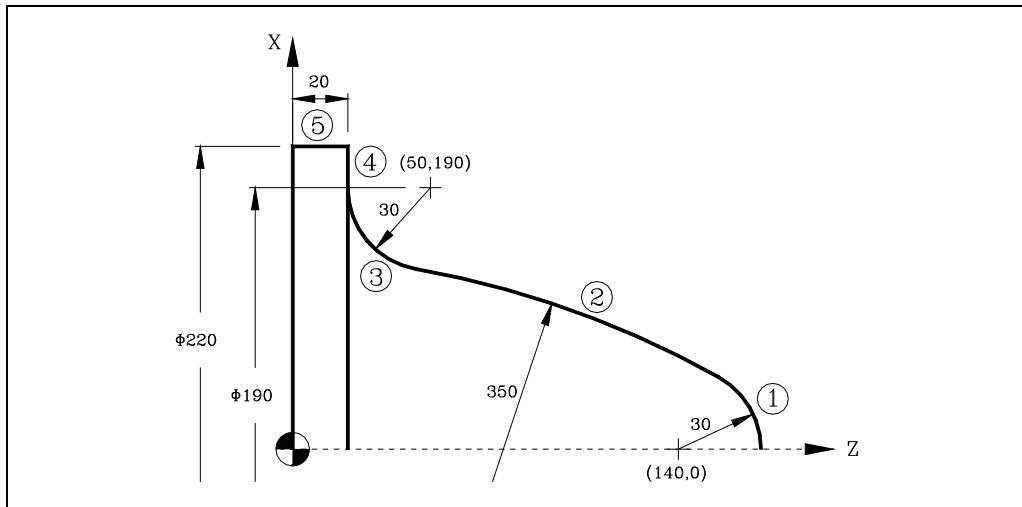
WORKING WITH OPERATIONS OR CYCLES

Profiling cycle

FAGOR 

CNC 8037

.TC. OPTION
SOFT: V01.4x



3.

WORKING WITH OPERATIONS OR CYCLES
Profiling cycle

Geometry definition.

Outside profile.

Tool of machining.

Work quadrant.

Profile definition.

Abscissa and ordinate of the starting point.			Z = 170	X = 0
Section 1	Counterclockwise arc	Zc = 140	Xc= 0	R = 30
Section 2	Counterclockwise arc		R = 350	Tangent = Yes
Section 3	Clockwise arc	Zc = 50	Xc =190	R = 30
The CNC shows the possible options for section 2. Select the correct one.				
Section 4	Straight	Z = 20	X = 220	Tangent = Yes
The CNC displays the possible options between sections 3-4. Select the correct one.				
Section 5	Straight	Z = 0	X = 220	

Coordinates (X, Z).

X 230.0000 Z 180.0000

Safety distance.

X 0.0000 Z 0.0000

Roughing.

F 1.000 S 1000 T 3 Δ 2

**Finishing.**

F 0.800 S 1000 T 3 δ 0.25

Spindle.

RPM

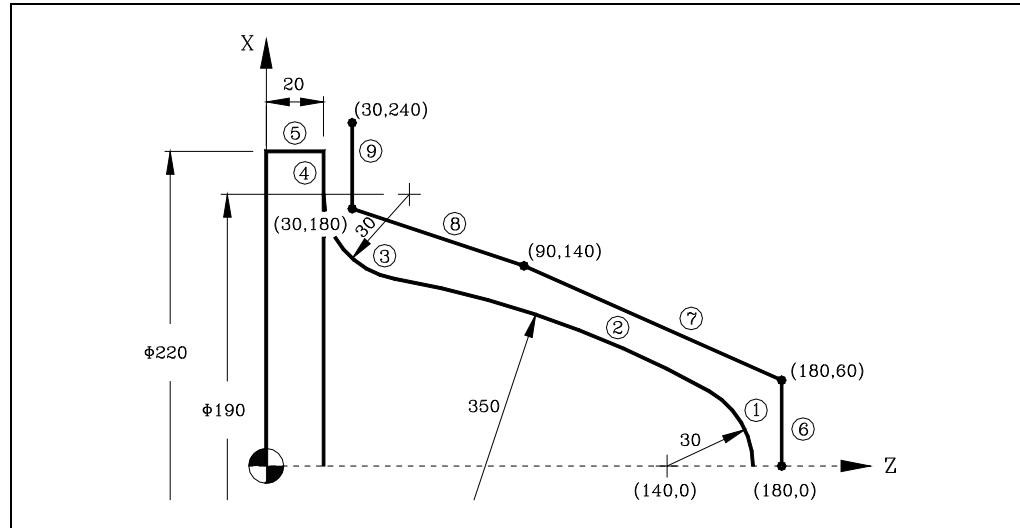

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CNC 8037

 ·TC· OPTION
SOFT: V01.4x

3.

WORKING WITH OPERATIONS OR CYCLES
Profiling cycle

**Geometry definition.**

Outside profile.

Tool of machining.

Work quadrant.

Profile definition.

Abscissa and ordinate of the starting point.	Z = 170	X = 0
Section 1 Counter-clockwise arc	Zc = 140	Xc = 0 R = 30
Section 2 Counter-clockwise arc	Zc = 50	R = 350 Tangent = Yes
Section 3 Clockwise arc	Zc = 50	Xc = 190 R = 30 Tangent = Yes
The CNC shows the possible options for section 2. Select the correct one.		
Section 4 Straight	Z = 20	X = 220 Tangent = Yes
The CNC displays the possible options between sections 3-4. Select the correct one.		
Section 5 Straight	Z = 0	X = 220

Definition of the profile of the original stock (new profile).

Abscissa and ordinate of the starting point.	Z = 180	X = 0
Section 6 Straight	Z = 180	X = 60
Section 7 Straight	Z = 90	X = 140
Section 8 Straight	Z = 30	X = 180
Section 9 Straight	Z = 30	X = 240

Coordinates (X, Z).

X 230.0000 Z 180.0000

Safety distance.

X 0.0000 Z 0.0000

Roughing.

F 1.000 S 1000 T 3 Δ 2



Finishing.

F 0.800 S 1000 T 3 δ 0.25

Spindle.

RPM



3.

WORKING WITH OPERATIONS OR CYCLES
Profiling cycle

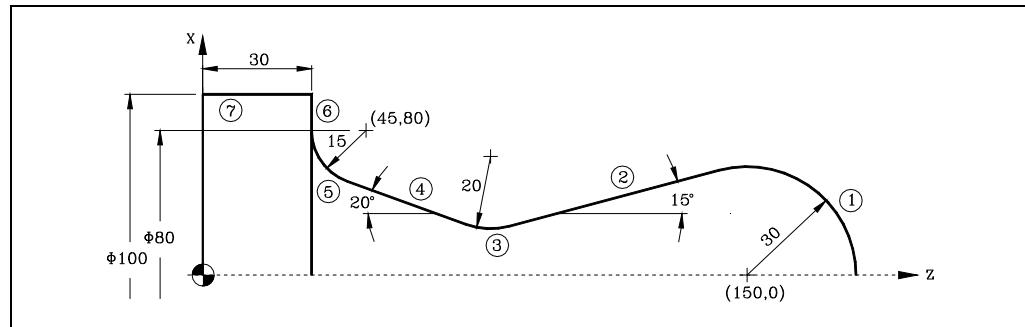
FAGOR 

CNC 8037

.TC- OPTION
SOFT: V01.4x

3.

WORKING WITH OPERATIONS OR CYCLES
Profiling cycle

**Geometry definition.**

Outside profile.

Tool of machining.

Work quadrant.

Profile definition.

Abscissa and ordinate of the starting point.	Z = 180	X = 0	
Section 1	Counterclockwise arc	Zc = 150	Xc = 0 R = 30
Section 2	Straight	Angle = 195	Tangent = Yes
The CNC displays the possible options between sections 1-2. Select the correct one			
Section 3	Clockwise arc	R = 20	Tangent = Yes
Section 4	Straight	Angle = 160	Tangent = Yes
Section 5	Clockwise arc	Z = 30 Zc = 45 R = 15 Xc = 80	X = 80 Tangent = Yes
The CNC displays the possible options between sections 4-5. Select the correct one			
The CNC shows the possible options for section 3. Select the correct one			
Section 6	Straight	Z = 30 X = 100	
Section 7	Straight	Z = 0 X = 100	

Coordinates (X, Z).

X 110.0000 Z 190.0000

Safety distance.

X 0.0000 Z 0.0000

Roughing.

F 1.000 S 1000 T 3 Δ 2

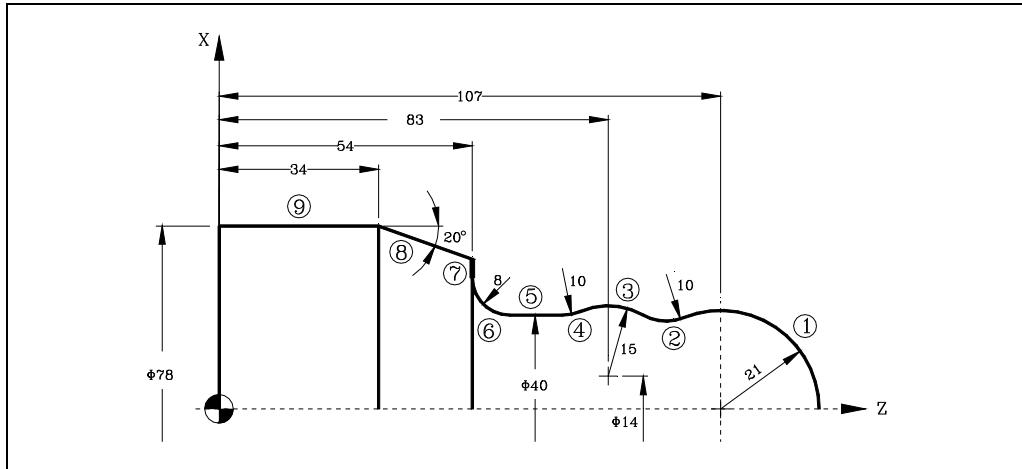
**Finishing.**

F 0.800 S 1000 T 3 δ 0.25

Spindle.

RPM

.TC. OPTION
SOFT: V01.4x**FAGOR** **CNC 8037**



3.

WORKING WITH OPERATIONS OR CYCLES

Profiling cycle

Geometry definition.



- Outside profile.
Tool of machining
Work quadrant.

Profile definition.

Abscissa and ordinate of the starting point.			Z = 128	X = 0		
Section 1	Counterclockwise arc Zc = 107	Xc= 0	R = 21			
Section 2	Clockwise arc	R = 10	Tangent = Yes			
Section 3	Counterclockwise arc Zc = 83	Xc= 14	R = 15	Tangent = Yes		
The CNC shows the possible options for section 2. Select the correct one						
Section 4	Clockwise arc	R = 10	Tangent = Yes			
Section 5	Straight	X = 40	Angle = 180	Tangent = Yes		
The CNC shows the possible options for section 4. Select the correct one						
Section 6	Clockwise arc	Z = 54				
X = 56	Zc = 62					
Xc= 56	R = 8	Tangent = Yes				
Section 7	Straight	Z = 54	Angle = 90	Tangent = Yes		
Section 8	Straight	Z = 34	X = 78	Angle = 160		
Section 9	Straight	Z = 0	X = 78			

Coordinates (X, Z).

X 85.0000 Z 135.0000

Safety distance.

X 0.0000 Z 0.0000

FAGOR 

CNC 8037

·TC· OPTION
SOFT: V01.4X

Roughing.

F 1.000 S 1000 T 3 Δ 2

**Finishing.**

F 0.800 S 1000 T 3 δ 0.25

Spindle.

RPM

**3.**

WORKING WITH OPERATIONS OR CYCLES
Profiling cycle

FAGOR

CNC 8037

.TC. OPTION
SOFT: V01.4x

OPERATING IN ISO MODE

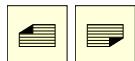
4

The ISO mode is accessed with the [ISO] key.

- When operating with operations or cycles, press the [ISO] key once.
- When operating in jog mode, press the [ISO] key twice; the first time to access the MDI mode and the second time to access the ISO mode. See "[2.9 ISO management](#)" on page 70.

Cycle levels

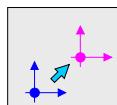
The ISO mode offers several editing levels. Each level has its own screen and the main window of the cycle indicates, with tabs, the available levels and which one is selected.



To change levels, use the [page up] and [page down] keys to scroll up and down through the various levels.



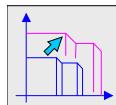
Block editing



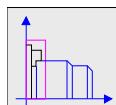
Zero offsets and presets



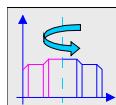
Place labels and repetitions from label to label



Scaling factor



Work zones



Mirror image

Simulating, executing or saving ISO cycles

Once the cycle data or the blocks have been edited, press [ESC]. The upper right side of the screen will show the "start" symbol. From this moment one, the edited blocks may be simulated, executed or saved like any operation or cycle.

- To simulate the blocks, press the [SIMUL] key.
- To execute the blocks, press the [START] key.
- To save the edited blocks, press the [EDIT] key.

It is possible to combine ISO-coded blocks with standard and/or user machining cycles to create part-programs. See chapter "[5 Saving programs](#)".



CNC 8037

.TC- OPTION
SOFT: V01.4x

4.

OPERATING IN ISO MODE
Editing blocks in ISO mode

4.1 Editing blocks in ISO mode

When accessing the ISO mode, the CNC displays a special screen for editing up to 6 program blocks either in ISO code or in high level language. After editing a block, press [ENTER] to validate it.

Example:

ISO

G95 G96 S120 M3



G0 Z100



G1 X30 F0.1



Once the block or blocks have been edited, press [ESC]. The upper right side of the screen will show the "start" symbol. From this moment one, the edited blocks may be simulated, executed or saved like any operation or cycle.

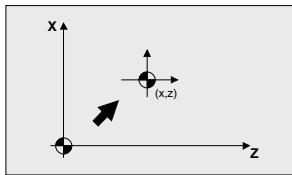


CNC 8037

.TC. OPTION
SOFT: V01.4x

4.2 Programming assistance

4.2.1 Zero offsets and presets



The icon may be used to select the following options:

- Machine zero. It cancels any zero offset and assumes the machine zero (home) as reference.
- Absolute zero offset. It permits defining, enabling or defining + enabling the absolute zero offsets (G54 ... G57, G159N7 ... G159N20).
- The zero offset is selected with an icon.
- Incremental offset: It permits defining, enabling or defining + enabling the incremental zero offsets (G58 or G59). The zero offset to be activated is selected with an icon.
- Preset: The preset value for the active axes may be edited on the screen. If any of them is not to be preset, leave its field empty.

The cycle internally generates a block with function G53, G54...G59, G159N7...G159N20 or G92.

Pressing the [ZERO] key accesses the zero offset table.

4

OPERATING IN ISO MODE

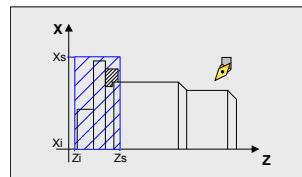
Programming assistance

FAGOR 

CNC 8037

.TC- OPTION
SOFT: V01.4x

4.2.2 Work zones



The icons may be used to defined the following options:

- Select the work zone to act upon. Up to five different zones may be defined.
- Type of action to carry out with the zone. A zone may be defined, enabled, defined+enabled or disabled.
- Define the type of zone. Each one of them may be defined either as a no-entry or no-exit zone.

The numeric fields permit editing the lower and upper limits of the zone. The limits are defined on the X and Z axes. To only define either the upper or the lower limit, leave the values of the other limit blank.

The cycle internally generates up to three blocks with functions G20, G21 and G22.

4.

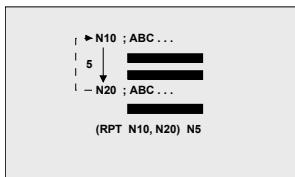
OPERATING IN ISO MODE
Programming assistance

FAGOR 

CNC 8037

.TC. OPTION
SOFT: V01.4x

4.2.3 Place labels and repetitions from label to label



This cycle permits editing labels and repetition blocks between two labels. It is selected with an icon.

- Edit labels. There is a field to enter the block number and another one to add a comment.

The cycle generates this type of a block: N10; -> Comment

- Block repetition. The repetition takes place between the first and final labels programmed, the number of times that is indicated.

The cycle internally generates an RPT block.

4

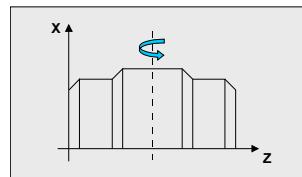
OPERATING IN ISO MODE
Programming assistance

FAGOR 

CNC 8037

.TC- OPTION
SOFT: V01.4x

4.2.4 Mirror image



The icons may be used to select the following options:

- Select the action to be carried out. It is possible to cancel the active mirror image, define a new one canceling the previous ones or define a new one and add it to the one that is currently active.
- Select the axes upon which mirror image is applied. This icon is not displayed if mirror image cancellation has been selected.

The cycle internally generates an ISO block that contains a combination of functions G10, G11 and G12.

4.

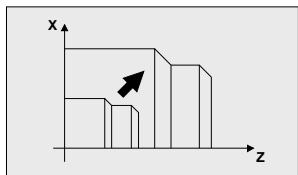
OPERATING IN ISO MODE
Programming assistance

FAGOR 

CNC 8037

.TC. OPTION
SOFT: V01.4x

4.2.5 Scaling factor



An icon is used to indicate whether to cancel the existing scaling factor or activate a new one. In the latter case, a numeric field is displayed to define the value of the scaling factor.

The cycle internally generates a block with function G72.

4.

OPERATING IN ISO MODE
Programming assistance

FAGOR 

CNC 8037

.TC- OPTION
SOFT: V01.4x

4.

OPERATING IN ISO MODE
Programming assistance



CNC 8037

.TC. OPTION
SOFT: V01.4x

SAVING PROGRAMS

5

Part-programs may be edited, simulated and executed.

Each one of these programs is made up by concatenating simple operations or cycles and/or blocks edited in ISO code. Chapter "*3 Working with operations or cycles*" describes how to edit or define those operations or cycles.

This chapter indicates how to operate with those part-programs through the following sections and sub-sections.

- List of saved programs.
- Seeing the contents of a program.
- Editing a new part-program.
- Saving an ISO block or a cycle.
- Deleting a new part program.
- Copying a part-program into another one.
- Modifying a part-program.
- Managing programs using the explorer



CNC 8037

.TC- OPTION
SOFT: V01.4x

5.1 List of saved programs

SAVING PROGRAMS
List of saved programs

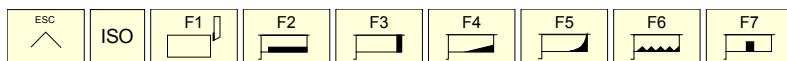
5.

 Press [EDIT] to access the list of saved part-programs.
It is not possible to access the list of part-programs directly if the "Tool calibration" mode is selected. First, quit this mode; in other words, first press [ESC] and then [P.PROG].

The CNC will display the following information:

15:28:42	IN POSITION
PROGRAMS - PARTS 1 - XFT123 2 - ABZ 2343 22 - 23 - 128 - MTB 234A 285 - XFT 127B 764 - 777 - 832 - ABZ2347C 833 - 1234 - 1236 - MTB 238 1245 - MTB 3434	CYCLES 1.- POSITIONING 1 2.- TURNING CYCLE 1 3.- TAPER TURNING CYCLE 1 4.- ROUNDING CYCLE 2

Press one of these keys to exit the directory or list of part-programs:



List of part-programs.

The left side shows the list of part-programs that are saved in the CNC's memory. Use the [\uparrow][\downarrow] keys to move the pointer over the list of programs. Use the [SHIFT][\uparrow] y [SHIFT][\downarrow] keys to scroll a page at a time.

It is possible to select the program by typing its number directly. If the desired program does not exist, the cursor will go to the nearest previous one. For example, to find program number 123, press the keystroke sequence [1], [2] and [3]. The time interval between keystrokes must be shorter than 1.5 seconds. A longer interval starts a new sequence.

Cycles that make up the part-program.

The right column shows the cycles and/or blocks edited in ISO code that make up the part. Once the list of programs has been selected, the following operations are possible:

- Creating a new part-program.
- Seeing the contents of a part-program.
- Deleting a new part program.
- Copying a part-program into another one.
- Modifying a part-program.

FAGOR 

CNC 8037

.TC. OPTION
SOFT: V01.4x

5.2 See the contents of a program

To see the contents of a part-program, select it on the left column using the pointer. To do that, use the [\uparrow][\downarrow] keys. The right column shows the cycles that make up the part.



When pressing [ENTER] or one of the [\leftarrow][\rightarrow] keys, the pointer switches to the right column. Now, the [\uparrow][\downarrow] keys may be used to move the pointer over the blocks or cycles that make up the part.



So, use the [\uparrow][\downarrow] keys to move up and down on each column and [\leftarrow][\rightarrow] keys to switch columns.

Once an operation has been selected, the following is possible:

- To see the operation in detail.
- To delete the operation.
- To move the operation to another position.
- To modify the operation.

5.

SAVING PROGRAMS

See the contents of a program

FAGOR 

CNC 8037

.TC- OPTION
SOFT: V01.4x

5.**SAVING PROGRAMS**

See the contents of a program



CNC 8037

.TC. OPTION
SOFT: V01.4x

Once the desired operation has been selected using the pointer, press [RECALL]. The CNC will show all the data for that operation.

In this moment it is possible to:

- Simulate the operation. See chapter "[6 Execution and simulation](#)".
- Execute the operation. See chapter "[6 Execution and simulation](#)".
- To modify the operation.
- Save the operation. Replacing the previous one or inserting it as a new one.

5.3 Edit a new part-program

To edit a new program, proceed as follows:



1. Press [EDIT] to access the list of part-programs stored.
2. Select, with the pointer, the option "Creating new part" on the left column.
3. Press the [EDIT] key. The bottom of the screen will request the number to be assigned to the new part-program and it will offer the first one available.



4. Enter the number of the desired program and press [ENTER].
It must be a number between 1 and 899999. Both included.
5. The CNC requests the comment to be associated with the part-program.
It is not necessary to associate a comment.



6. Press the [ENTER] or [ESC].
The CNC includes the new part-program in the list (left column).

From this moment on, all the desired operations may be saved in the desired order.

5.

SAVING PROGRAMS

Edit a new part-program

FAGOR

CNC 8037

.TC- OPTION
SOFT: V01.4x

5.**SAVING PROGRAMS**

Saving an ISO block or a cycle

5.4 Saving an ISO block or a cycle

The block or cycle may be added at the end of the program, after the last operation or insert it between 2 existing operations.

To save the block or cycle, proceed as follows:

1. Define the desired block or cycle assigning the relevant data to it.
2. Press [EDIT] to access the list of part-programs stored.
3. Select, with the pointer, the desired program number on the left column and switch over to the right column.
4. Place the pointer on the operation after which the block or cycle is to be saved and press [ENTER].

Example:

Current program	Desired program
Positioning cycle 2	Positioning cycle 2
Facing cycle	Facing cycle
Taper turning cycle 2	Taper turning cycle 2
	Turning cycle
Rounding cycle 2	Rounding cycle 2
Taper turning cycle 1	Taper turning cycle 1
	Threading cycle 1

To insert the "turning cycle" operation, once the operation has been defined, place the pointer on the "taper turning cycle 2" operation and press [ENTER].

To insert the "threading cycle 1" operation, once the operation has been defined, place the pointer on the last "taper turning cycle 1" operation and press [ENTER].



CNC 8037

.TC. OPTION
SOFT: V01.4x

5.5 Delete a new part program

To delete a part-program, proceed as follows:

1. Press [EDIT] to access the list of part-programs stored.
2. Select, with the pointer, the part-program to be deleted from the left column.



3. Press [CLEAR].

The bottom of the screen will show a message requesting confirmation of the deleting operation.

- If [ENTER] is pressed, the CNC will delete the selected program and will update the list of saved programs.
- If [ESC] is pressed, the program will not be deleted and it will quit deleting operation.

5.

SAVING PROGRAMS

Delete a new part program

FAGOR 

CNC 8037

.TC- OPTION
SOFT: V01.4x

5.**SAVING PROGRAMS**

Copying a part-program into another one

5.6 Copying a part-program into another one

To copy a part-program into another one, proceed as follows:

1. Press [EDIT] to access the list of part-programs stored.
 2. Select, with the pointer, the part-program to be copied from the left column.
 3. Press the [EDIT] key.
- The bottom of the screen will show a message requesting the number to be assigned to the copy.
4. Enter the number of the desired program and press [ENTER].
 - It must be a number between 1 and 899999. Both included.
 5. If there is already a part-program with that number, the bottom of the screen will show a message asking to either replace it or cancel the operation?
 - If [ENTER] is pressed, the CNC will request a new program. If [ESC] is pressed, the CNC will not delete the current program and will copy the program.
 6. The CNC requests the comment to be associated with the new part-program (the copy). It is not necessary to associate a comment.
 7. Press the [ENTER] or [ESC]. The CNC updates the list of saved part-programs.



CNC 8037

.TC. OPTION
SOFT: V01.4x

5.7 Modify a part-program

To modify a part-program, proceed as follows:

1. Press [EDIT] to access the list of part-programs stored.
2. Select, with the pointer, the part-program to be modified from the left column.

Once the program has been selected, the following operations are possible:

- To delete an operation.
- To move an operation to another position.
- To add or insert a new operation.
- To modify an existing operation.

5.

SAVING PROGRAMS
Modify a part-program



CNC 8037

.TC- OPTION
SOFT: V01.4x

5.7.1 Delete an operation

To delete an operation, proceed as follows:

1. Select, with the pointer, the operation to be deleted from the right column.
2. Press [CLEAR].

The bottom of the screen will show a message requesting confirmation of the deleting operation.

- If [ENTER] is pressed, the CNC will delete the selected operation and will update the right column.
- If [ESC] is pressed, the operation will not be deleted and it will quit deleting operation.

5.

SAVING PROGRAMS

Modify a part-program



CNC 8037

.TC. OPTION
SOFT: V01.4x

5.7.2 Add or insert a new operation

To add or insert an operation follow the same steps as for saving an operation.

1. Define the desired block or cycle assigning the relevant data to it.
2. Press [EDIT] to access the list of part-programs stored.
3. Place the pointer on the operation after which the block or cycle is to be saved and press [ENTER].

5.

SAVING PROGRAMS
Modify a part-program



CNC 8037

.TC- OPTION
SOFT: V01.4x

5.7.3 Move an operation to another position

To move an operation to another position, proceed as follows:

1. Select, with the pointer, the operation to be moved from the right column.
-  2. Press the [-] key.
The CNC will highlight that operation.
3. Place the cursor on the operation after which the operation is to be placed and press [ENTER].

5.

SAVING PROGRAMS

Modify a part-program

Example:

Current program	Desired program
Positioning cycle 2	Positioning cycle 2
Facing cycle	Taper turning cycle 2
Taper turning cycle 2	Turning cycle
Turning cycle	Rounding cycle 2
Rounding cycle 2	Taper turning cycle 1
Taper turning cycle 1	Facing cycle
Threading cycle 1	Threading cycle 1

Select the "facing cycle" operation and press the two-color key. Then position the cursor on the "taper turning cycle 1" operation and press [ENTER].



CNC 8037

.TC. OPTION
SOFT: V01.4x

5.7.4 Modify an existing operation

To modify an operation, proceed as follows:

1. Select, with the pointer, the block or cycle to be modified from the right column.
2. Press the [RECALL] key.
The CNC will show the editing page for that operation.
3. Modify all the desired data.

To save the modified operation, proceed as follows:

1. Press [EDIT] to access the list of part-programs stored.

The CNC shows the pointer on the same operation. To select another position, use the [\uparrow][\downarrow] keys. The new operation will be inserted after the selected operation.

2. Press the [ENTER] key.

To place the modified operation in its previous location, the CNC will show a message asking to either replace the previous operation or keep it inserting the new one after it.

The following example modifies the "facing cycle" operation:

Current program	Option "Replace"	Option "Insert"
1.- Facing cycle	1.- Facing cycle	1.- Facing cycle
2.- Taper turning cycle 2	2.- Taper turning cycle 2	2.- Facing cycle
		3.- Taper turning cycle 2



It is possible to select an existing operation, modify it and then insert it in another position and even in another part-program.

5.

SAVING PROGRAMS

Modify a part-program

FAGOR

CNC 8037

.TC- OPTION
SOFT: V01.4x

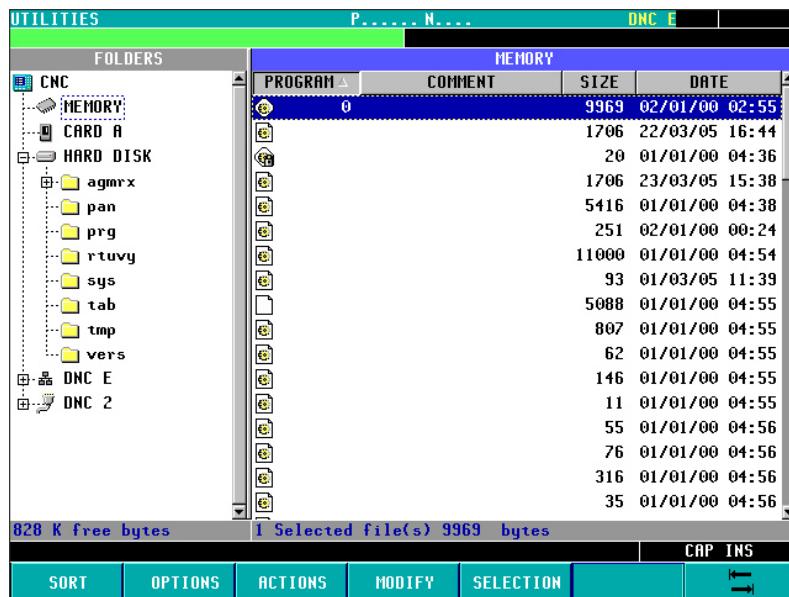
5.**SAVING PROGRAMS**

Managing programs using the explorer

5.8 Managing programs using the explorer

Windows Explorer may be accessed from the PPROG screen by placing the cursor in the "user programs" area and pressing [RECALL]. Pressing the [ESC] key returns to the PPROG screen.

Accessing the explorer displays a window divided in two areas (left panel and right panel) as shown in the next figure:



Once in the explorer, it will be possible to select any program of the Ram memory or hard disk (KeyCF) to edit it or execute it. When selecting the hard disk (KeyCF), the CNC will select the PRG folder automatically because it is the only folder where programs may be executed.



When selecting programs of the hard disk (KeyCF), only programs from the PRG folder may be selected.

When returning from the explorer to the PPROG screen with a different device, the CNC will display a message warning that the device has been changed and requesting confirmation.

Once out of the explorer, the CNC will show an indicator of the device that has been selected. This indicator will appear to the left of the selected program.



The subroutines and profiles defined in the profile cycles (the profiles defined in the cycle as well as the associated profile programs) must be in user RAM memory although the call to the cycle may be in a program of the hard disk (KeyCF).

*Programs contained in USB or in DNC (infinite program) can be neither edited nor executed.
The selected device will be maintained even after turning the CNC off or after a Shift/Reset.*



CNC 8037

.TC. OPTION
SOFT: V01.4x

EXECUTION AND SIMULATION

6

The simulation may be used to graphically show a part-program or an operation with the data used to define it. This way, the simulation may be used to check the part-program or the operation before executing it or saving it and, therefore, correct or modify its data.



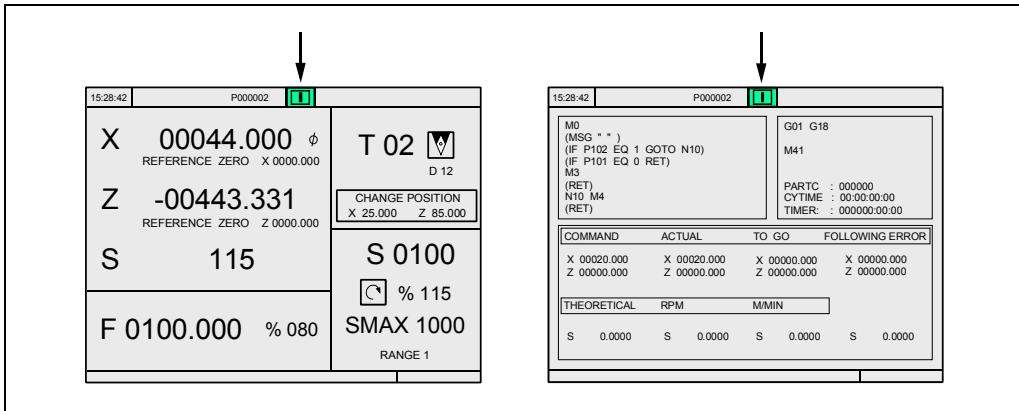
It is possible to execute or simulate a part-program or any operation. The simulation or execution may be done from beginning to end or step by step by pressing the [SINGLE] key.

It is possible to simulate or execute:

- An operation or Cycle.
- A part-program.
- An operation saved as part of a part-program.

Selecting a program for simulation or execution

When selecting a part-program or operation saved as part of a part-program for simulation or execution, the CNC selects that part-program and shows it highlighted next to the green "start" symbol in the top center window.



When the top center window shows the part-program selected next to the green "start" symbol, the CNC acts as follows:

- If [START] is pressed, the CNC executes the part-program that is selected.
- If [CLEAR] is pressed, the CNC de-selects the part-program and removes it from the top center window.



CNC 8037

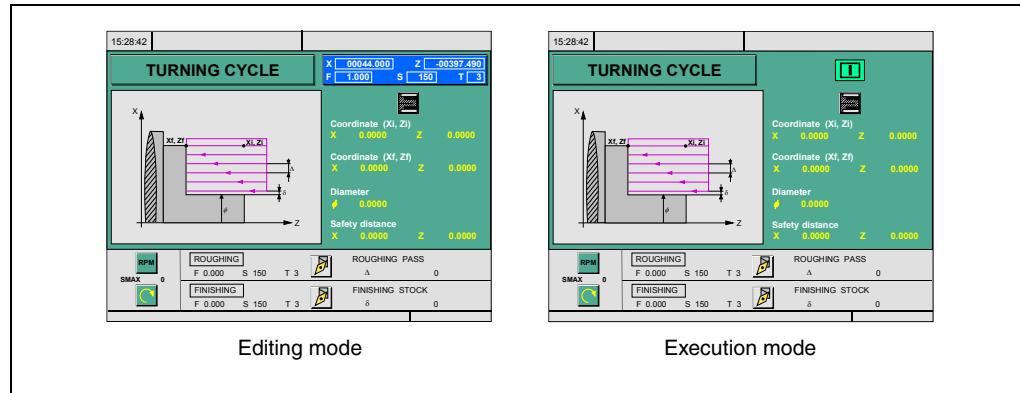
.TC- OPTION
SOFT: V01.4x

6.**EXECUTION AND SIMULATION**

Simulating or executing an operation or cycle

6.1 Simulating or executing an operation or cycle

All the operations or cycles have 2 work modes; execution and editing.

**Simulation**

The operation or cycle may be simulated in both work modes. To do that, press the [GRAPHICS] key. The CNC shows the graphic screen of the T model.

Execution

An operation or cycle can only be executed in the cycle execution mode.

The operation or cycle cannot be executed when the cycle editing mode is selected. Press [ESC] to quit the editing mode and switch to execution mode.

Press [START] to execute an operation or cycle.



CNC 8037

.TC. OPTION
SOFT: V01.4x

6.2 Simulating or executing a part-program

Proceed as follows to simulate or execute a part-program:

1. Press [EDIT] to access the list of part-programs stored.
2. Select on the left column the program to be simulated or executed.

Press [SIMUL] to simulate the part-program and [START] to execute it. When executing a program, the CNC executes the initial subroutine 9998 and the final subroutine 9999.

If an error occurs in a cycle during simulation or execution, when accessing the list of programs again, the cursor will be placed on the cycle that came up with the error. When program 999998 is displayed on the screen or the error is not an execution error, the cursor will go to the beginning or to the end of the program depending on its length.

6.

EXECUTION AND SIMULATION

Simulating or executing a part-program



CNC 8037

.TC- OPTION
SOFT: V01.4x

6.2.1 Simulating or executing a portion of a part-program

Proceed as follows to simulate or execute a portion of a part-program:

1. Press [EDIT] to access the list of part-programs stored.
2. Select the program from the left column and the operation (on the right column) from which to begin executing or simulating the part program.

Press [SIMUL] to simulate the selected portion and [START] to execute it. When executing a portion of a program, the CNC does not execute the initial subroutine 9998, but it executes the final subroutine 9999. If the program is executed from the first operation on, it executes both subroutines.

6.

EXECUTION AND SIMULATION

Simulating or executing a part-program



CNC 8037

.TC. OPTION
SOFT: V01.4x

6.3 Simulating or executing an operation that has been saved

Proceed as follows to simulate or execute an operation that has been saved as a part of program:

1. Press [EDIT] to access the list of part-programs stored.
2. Select the program that contains it from the left column and the operation to be simulated or executed from the right column.
3. Press the [RECALL] key.

Press [SIMUL] to simulate the operation and [START] to execute it.

6.

EXECUTION AND SIMULATION

Simulating or executing an operation that has been saved



CNC 8037

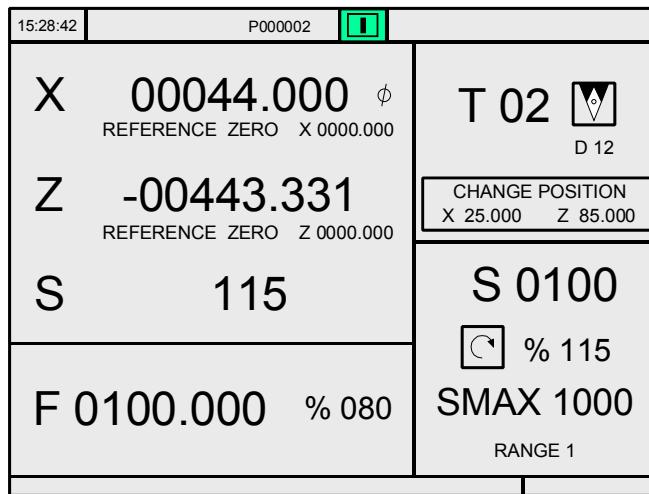
.TC- OPTION
SOFT: V01.4x

6.

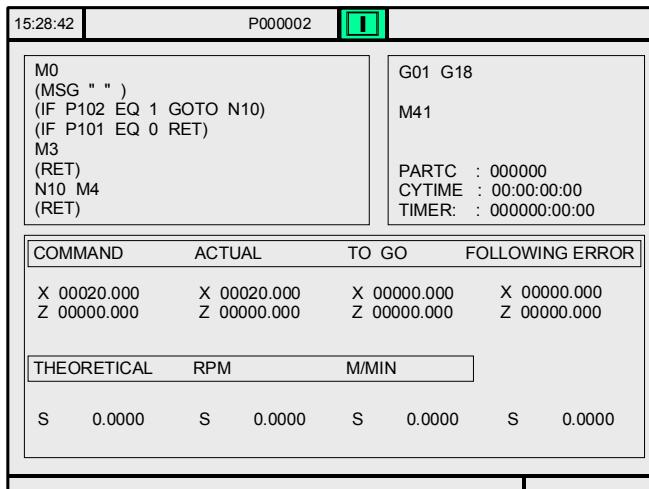
EXECUTION AND SIMULATION
Execution mode



When pressing [START] to execute an operation or part-program, the CNC shows the standard screen of the TC mode.



When pressing the [-] key, the CNC shows the special screen of the TC mode.



During execution, both screens show (in the top center window) the number of the program and the number of the cycle being executed. However, when detecting an RPT or GOTO instruction, the cycle number will disappear.



Once the operation or part has been selected, it may be executed as often as desired; to do that, once the execution has finished, pressed the [START] again.



To interrupt the execution, press [STOP]. Once the execution has been interrupted, a tool change may be done. See "["6.4.1 Tool inspection"](#) on page 183.



While executing an operation or a part, the [SIMUL] key may be pressed to access the graphics display mode.

FAGOR

CNC 8037

.TC. OPTION
SOFT: V01.4x

6.4.1 Tool inspection

The PLC mark M5050 "TOOLINSP" determines when tool inspection is enabled.



TOOLINSP=0 Tool inspection is possible after pressing [STOP].

TOOLINSP=1 Pressing [STOP] interrupts the execution of the program. To move the axes and do a tool inspection, press the [T] key once the execution of the program has been interrupted.

Once tool inspection has been selected, it is possible:

- To move the axes to the tool change point.
- To select another tool.
- To modify the tool values.
- Resuming program execution.

To move the axes to the tool change point.

To move the axes with the handwheels or the jog keypad to the point where the tool will be changed.

To select another tool.

The standard screen of the TC mode must be selected in order to be able to make a tool change.

Press the [T] key. The CNC highlights the tool number.



Type in the number of the tool to be selected and press [START] so the CNC selects the new tool. The CNC will manage the tool change.

Modifying the tool values (dimensions and geometry).



Press the key associated with tool calibration. The CNC shows the screen for tool calibration.

It is possible to modify the dimensions of the tool (offsets I, K to compensate for wear) or the values for tool geometry.

Press the [ESC] key to exit this screen and go back to the previous one (still in tool inspection).

Resuming program execution.



Press the [START] key to resume program execution. The CNC will reposition the tool by moving it to the point where tool inspection began.

There may be 2 cases; either only one axis moved or several did.

- Only one axis moved.
The CNC repositions it and resumes the execution.
- Both axes moved.

The CNC will show a window with the following options to choose the order the axes will be repositioned.

PLANE The axes of the X-Y plane move at the same time.

Z-X When moving the axes of the plane, the Z axis moves first and then the X.

X-Z When moving the axes of the plane, the X axis moves first and then the Z.

6.

EXECUTION AND SIMULATION
Execution mode

FAGOR 

CNC 8037

.TC- OPTION
SOFT: V01.4x

6.

6.5 Graphic representation



When pressing the [SIMUL] key, the CNC shows the graphics screen of the T model. To quit the graphics mode, press either [SIMUL] or [ESC].

During simulation, the top center window shows the number of the program and the number of the cycle being executed. However, when detecting an RPT or GOTO instruction, the cycle number will disappear.

The operating manual, T model, "Graphics" section of the "Execution / Simulation" chapter describes how to use the graphics display. However, here is a brief description of the softkeys.

Type of graphic

The graphics may be "X-Z", "X-C", "Z-C", "Solid X-Z", "Solid X-C" or "Solid Z-C".

The "X-Z", "X-C" and "Z-C" graphics are line graphics that describe the tool tip movement with color lines.

The graphics start from a three-dimensional block and the tool removes material during the execution or simulation showing the shape of the resulting part.

Display area

XZ, XC, ZC They draw the part in the selected plane.

SOLID It shows a three-dimensional block that will be "machined" as the part-program is being run.

With this option it is possible to re-define the display area by defining the maximum and minimum values for each axis.

Use the [\uparrow][\downarrow] keys to select the maximum and minimum values. Once all the data has been defined, press [ENTER].

When selecting a new display area, the CNC clears the screen and shows the axes or the part unmachined.



The display area cannot be changed while executing or simulating the part. If it is, interrupt the execution or simulation by pressing the [STOP] key.

Zoom

With this option, it is possible to enlarge or reduce the graphics display area.

It shows a window superimposed on the current graphics and another one over the drawing at the lower right-hand side of the screen. These new windows indicate the new display area being selected.

Use the [\leftarrow][\rightarrow][\uparrow][\downarrow] keys to move the window around, the softkeys "ZOOM+" "ZOOM-" to enlarge or shrink its size and press [ENTER] for those new values to be assumed.

Every time a new display area is selected, the CNC keeps the current graphic representation. It does not delete it.



When pressing [START] to resume or re-start the execution or simulation, it deletes the current graphic representation and starts a new one with the new values.



The zoom function cannot be used while executing or simulating the part. If it is, interrupt the execution or simulation by pressing the [STOP] key.

Graphic parameters

- Simulation speed.

Select on the top right side of the screen the percentage of simulation speed to be applied.

Use the [**◀**][**▶**] keys to select the percentage and press [ENTER] for the CNC to assume that value.

- Tool path colors.

It only makes sense in line graphics (not in solid ones). It lets you choose the colors to represent rapid traverse, the tool path without compensation, the tool path with compensation and threading.

Use the [**↑**][**↓**] keys to select, on the right side of the screen, the type of path and use the [**◀**][**▶**] keys to select the color to be applied.

Press [ENTER] for the CNC to assume those values.

- Colors of solid.

It only makes sense in solid graphics (not in line graphics). It lets you use the colors to represent the cutter, the part, the axes and the clamps.

Use the [**◀**][**▶**] keys to select, on the right side of the screen, the type of path and use the [**↑**][**↓**] keys to select the color to be applied.

Press [ENTER] for the CNC to assume those values.

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EXECUTION AND SIMULATION

Graphic representation

Clear screen

When selecting this option, the CNC clears the screen and shows the axes or the part unmachined.



The screen cannot be cleared while executing or simulating the part. If it is, interrupt the simulation by pressing the [STOP] key.

Starting graphic simulation



Once the type of graphics, the display area, the graphic parameters, etc have been selected, press the [START] key to start graphic simulation.

During graphic simulation, the CNC takes into account the simulation speed and the position of the feedrate override switch (FEED 0%-120%).

When selecting a new simulation speed, the CNC applies 100% of it regardless of the position of the switch. Once the switch is turned, the CNC begins to apply the selected %.

To interrupt the simulation, press [STOP].



To quit the simulation mode, press either [GRAPHICS] or [ESC].

ISO

In the conversational mode of the CNC, it is possible to generate an ISO-coded program with some basic G functions as well as M and T functions from an operation (cycle) or on a part-program.

To enable this feature, set general machine parameter ISOSIMUL (P183) with a value other than zero. This parameter identifies the number of the ISO program generated in user RAM memory.

The program generated from the conversational program is an ISO-coded program created in RAM memory. This program may be later edited, copied or executed at the CNC itself.

To generate the ISO program, use the simulation in conversational mode through the "SIMUL" key. This can be done in a whole program on the simulation screen or in any of the particular TC cycles.



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Once inside the graphic simulation screen, the ISO generation maybe selected using the <ISO> softkey. Then, pressing [CYCLE START] will generate the program defined by machine parameter ISOSIMUL (that will only contain ISO instructions) while it simulates it graphically.

When generating that program, parametric programming, rounding (G36), tangential entry (G37), tangential output (G38), chamfer (G39) and path tangent to previous path (G8) are resolved and generated only using G1, G2 and G3 blocks.

If the program already exists, it will be deleted without requesting confirmation. If the generated program exceeds the available user memory, the CNC will issue the corresponding error message, but it will keep in memory the portion of the ISO program generated so far.

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EXECUTION AND SIMULATION

Graphic representation

Description of the functions used to generate an ISO-coded program:

The ISO-coded program is mainly generated to control axes. It provides information of the moving conditions and indications on feedrate using the following set of functions

- **G functions:** Motion preparatory functions to determine the geometry and work conditions.

Function	Meaning
G2 (G3) G6 X Y I J	In the circular interpolation, the center will be programmed with respect to the origin, not referred to the starting point of the circle.

Rigid tapping cycles G84 (mill) and G86 (lathe) only generate the equivalent ISO block.

- **"F" and "S" functions:** Functions to control the feedrate of the axes and the spindle turning speed.

- **T and D functions:** Tool control functions.

If the T function has an associated subroutine, the blocks of this subroutine are ignored when generating the program set by g.m.p. ISOSIMUL (P183).

- **M functions:** Complementary or auxiliary functions.

If the M functions have an associated subroutine, the blocks of this subroutine are ignored when generating the program set by g.m.p ISOSIMUL (P183).



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