CNC 8055 ·TC·

Operating manual

Ref.1705
Soft: V02.2x

FAGOR

FAGOR AUTOMATION

FAGOR AUTOMATION

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

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

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

INDEX

CHAPTER 1   **GENERAL CONCEPTS**

CHAPTER 2   **OPERATING IN JOG MODE**

CHAPTER 3   **WORKING WITH OPERATIONS OR CYCLES**

FAGOR ◢

FAGOR AUTOMATION

CNC 8055
CNC 8055i

Soft: V02.2x

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

SOFT: V02.2x

CHAPTER 7   EXECUTION AND SIMULATION

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

SOFT: V02.2X

ABOUT THE PRODUCT

BASIC CHARACTERISTICS OF THE DIFFERENT MODELS.

	8055i FL EN	8055 FL 8055i FL	8055 Power 8055i Power
Pendant	8055i FL EN	8055i FL	8055i Power
Enclosure	-----	8055 FL	8055 Power
USB	Standard	Standard	Standard
Block processing time	1 ms	3.5 ms	1 ms
RAM memory	1Mb	1Mb	1 Mb
Software for 7 axes	-----	-----	Option
TCP transformation	-----	-----	Option
C axis (Lathe)	-----	-----	Option
Y axis (Lathe)	-----	-----	Option
Look-ahead	100 blocks	100 blocks	200 blocks
Flash Memory 512Mb / 2Gb	512Mb	Option	Option

HARDWARE OPTIONS OF THE 8055I CNC

	Analog	Digital	Engraving
Ethernet	Option	Option	Option
RS232 serial line.	Standard	Standard	Standard
16 digital inputs and 8 outputs (I1 to I16 and O1 to O8)	Standard	Standard	Standard
Another 40 digital inputs and 24 outputs (I65 to I104 and O33 to O56)	Option	Option	Option
Probe inputs	Standard	Standard	Standard
Spindle (feedback input and analog output)	Standard	Standard	Standard
Electronic handwheels	Standard	Standard	Standard
4 axes (feedback and velocity command)	Option	Option	- - -
Remote CAN modules, for digital I/O expansion (RIO).	Option	Option	- - -
Sercos servo drive system for Fagor servo drive connection.	- - -	Option	- - -
CAN servo drive system for Fagor servo drive connection.	- - -	Option	- - -

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

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

SOFTWARE OPTIONS OF THE 8055 AND 8055I CNCS.

	Model							
	GP	**M**	**MC**	**MCO**	**EN**	**T**	**TC**	**TCO**
Number of axes with standard software	4	4	4	4	3	2	2	2
Number of axes with optional software	7	7	7	7	-----	4 or 7	4 or 7	4 or 7
Electronic threading	-----	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.
Tool magazine management:	-----	Stand.	Stand.	Stand.	-----	Stand.	Stand.	Stand.
Machining canned cycles	-----	Stand.	Stand.	-----	Stand.	Stand.	Stand.	-----
Multiple machining	-----	Stand.	Stand.	-----	Stand.	-----	-----	-----
Solid graphics	-----	Stand.	Stand.	Stand.	-----	Stand.	Stand.	Stand.
Rigid tapping	-----	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.
Tool life monitoring	-----	Opt.	Opt.	Opt.	Stand.	Opt.	Opt.	Opt.
Probing canned cycles	-----	Opt.	Opt.	Opt.	Stand.	Opt.	Opt.	Opt.
DNC	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.
COCOM version	Opt.	Opt.	Opt.	Opt.	-----	Opt.	Opt.	Opt.
Profile editor	Stand.	Stand.	Stand.	Stand.	-----	Stand.	Stand.	Stand.
Tool radius compensation	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.
Tangential control	Opt.	Opt.	Opt.	Opt.	-----	Opt.	Opt.	Opt.
Retracing	-----	Opt.	Opt.	Opt.	Stand.	Opt.	Opt.	Opt.
Setup assistance	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.
Irregular pockets with islands	-----	Stand.	Stand.	Stand.	-----	-----	-----	-----
TCP transformation	-----	Opt.	Opt.	Opt.	-----	-----	-----	-----
C axis (on Lathe)	-----	-----	-----	-----	-----	Opt.	Opt.	Opt.
Y axis (on Lathe)	-----	-----	-----	-----	-----	Opt.	Opt.	Opt.
Telediagnosis	Opt.	Opt.	Opt.	Opt.	Stand.	Opt.	Opt.	Opt.

DECLARATION OF CONFORMITY AND WARRANTY CONDITIONS

DECLARATION OF CONFORMITY

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

WARRANTY TERMS

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

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

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-MC	Operating manual for the MC option.
OPT-TC	Operating manual for the TC option.
OPT-CO	Manual of the CO manual

Software V01.00   **October 2010**

First version.

Software V01.20   **April 2011**

List of features	Manual
Open communication.	INST
Improvements to Look Ahead machining.	INST
Blocks with helical interpolation in G51.	PRG
G84. Tapping with relief.	PRG

Software V01.08   **August 2011**

List of features	Manual
Spindle parameter OPLDECTI (P86).	INST

Software V01.30   **September 2011**

List of features	Manual
Gear ratio management on Sercos spindles	INST
Improved feedrate limit management (FLIMIT).	INST
New type of penetration in lathe type threading cycles.	PRG
Improved lathe type thread repair. Partial repair.	PRG
MC option: Rigid tapping with relief.	OPT-MC
TC option: New type of penetration in threading cycles.	OPT-TC
TC option: Improved thread repair. Partial and multi-entry (start) thread repair.	OPT-TC
TC option: Zig-zag entry to the groove at the starting point of the groove.	OPT-TC

Software V01.31   October 2011

List of features	Manual
CNC 8055 FL Engraving model	INST / OPT/ PRG

Software V01.40   January 2012

List of features	Manual
Execution of M3, M4 and M5 using PLC marks	INST / PRG
Values 12 and 43 of variable OPMODE in conversational work mode.	INST / PRG

Software V01.60   December 2013

List of features	Manual
Auto-adjustment of axis machine parameter DERGAIN.	INST
New value for axis machine parameter ACFGAIN (P46).	INST
Value 120 of the OPMODE variable.	INST / PRG

Software V01.65   January 2015

List of features	Manual
Block processing time of 1 ms on the "CNC 8055i FL Engraving" model.	INST / OPT/ PRG

Software V02.00   February 2014

List of features	Manual
Profile machining in segments. J parameter for G66 and G68 cycles.	PRG
Calls to subroutines using G functions.	INST / PRG
Anticipated tool management.	INST
Managing "PNG" and "JPG" graphic elements.	INST
New values for parameters MAXGEAR1..4 (P2..5), SLIMIT (P66) and MAXSPEED (P0).	INST
Retracing function of 2000 blocks.	INST
Quick block search.	OPT
Local subroutines within a program.	PRG
Avoid spindle stop with M30 or RESET. Spindle parameter SPDLSTOP (P87).	INST
Programming T and M06 with associated with a subroutine in the same line.	PRG
New values of the OPMODE variable.	INST / PRG
New variables: DISABMOD, GGSN, GGSO, GGSP, GGSQ, CYCCHORDERR.	INST / PRG
Possibility to set the parameters of SERCOS nodes in a non-sequential order.	INST
WRITE instruction: "$" character followed by "P".	PRG
Cancel additive handwheel offset with G04 K0. General parameter ADIMPG (P176).	INST / PRG
Ethernet parameter NFSPROTO (P32). TCP or UDP protocol selection.	INST
Face thread repair cycle.	OPT TC
Penetration increment (step) in thread repair.	INST / OPT TC
API compliant thread.	OPT TC
Roughing by segments in inside profiling cycles 1 and 2.	INST / OPT TC
Programming the Z increment and the angle on threads.	INST / OPT TC
Reversal of the starting and final point of the face thread repair.	INST / OPT TC
Manual tool calibration without stopping the spindle during each step.	INST / OPT TC

Software V02.03   **July 2014**

List of features	Manual
Set PAGE and SYMBOL instructions support PNG and JPG/JPEG formats.	PRG
New values for parameters MAXGEAR1..4 (P2..5), SLIMIT (P66), MAXSPEED (P0) and DFORMAT (P1).	INST

Software V02.10   **November 2014**

List of features	Manual
Incremental zero offset (G158).	INST / PRG
Programs identified with letters.	OPT
Variables PRGN and EXECLEV.	INST
Korean language.	INST
Change of default value for general machine parameters: MAINOFFS (P107), MAINTASF (P162) and FEEDTYPE (P170).	INST
New variable EXTORG.	INST / PRG
Image handling via DNC.	PRG
Save/restore a trace of the oscilloscope.	OPT

Software V02.21   **July 2015**

List of features	Manual
PLC library.	INST
Zero offsets table in ISO mode.	OPT
Compensation of the elastic deformation in the coupling of an axis.	INST
Machine axis parameter DYNDEFRQ (P103).	INST
Change of maximum value of axis and spindle parameter NPULSES.	INST

Software V02.22   **March 2016**

List of features	Manual
Axis filters for movements with the handwheel. General machine parameter HDIFFBAC (P129) and machine axis parameter HANFREQ (P104).	INST
Change of maximum value of axis and spindle parameter NPULSES.	INST

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

SAFETY CONDITIONS

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

The unit can only be repaired by personnel authorized by Fagor Automation.

Fagor Automation shall not be held responsible of any physical or material damage originated from not complying with these basic safety rules.

PRECAUTIONS AGAINST PERSONAL HARM

- 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 electric shocks.

  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 ºC (113º F).

- Do not operate this unit in explosive environments.

  In order to avoid risks, harm or damages, do not work in explosive environments.

Safety conditions

- Work environment.

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

  Fagor Automation shall not be held responsible for any damage that could suffer or cause when installed under other conditions (residential or domestic environments).

- Install this unit in the proper place.

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

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

  – Powerful loads connected to the same mains as the unit.

  – Nearby portable transmitters (radio-telephones, Ham radio transmitters).

  – Nearby radio / TC transmitters.

  – Nearby arc welding machines.

  – Nearby high voltage lines.

  – Etc.

- Enclosures.

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

- Avoid disturbances coming from the machine 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 Ω / 1 W y C=0,2 µF / 600 V.

  – AC motors. RC connected between phases, with values of R=300 Ω / 6 W y C=0,47 µF / 600 V.

- Use the proper power supply.

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

- Connecting the power supply to ground.

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

- Analog inputs and outputs connection.

  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 ºC and +40 ºC (41ºF and 104º F)

  The storage temperature must be between -25 ºC and +70 ºC. (-13 ºF and 158 ºF)

- Monitor enclosure (CNC 8055) or central unit (CNC 8055i)

  Guarantee the required gaps between the monitor or 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.

FAGOR ≡

FAGOR AUTOMATION

CNC 8055
CNC 8055i

PROTECTIONS OF THE UNIT ITSELF (8055)

- "Axes" and "Inputs-Outputs" modules.

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

  They are protected by an external fast fuse (F) of 3.15 A 250V against overvoltage of the external power supply (over 33 Vdc) and against reverse connection of the power supply.

- Monitor.

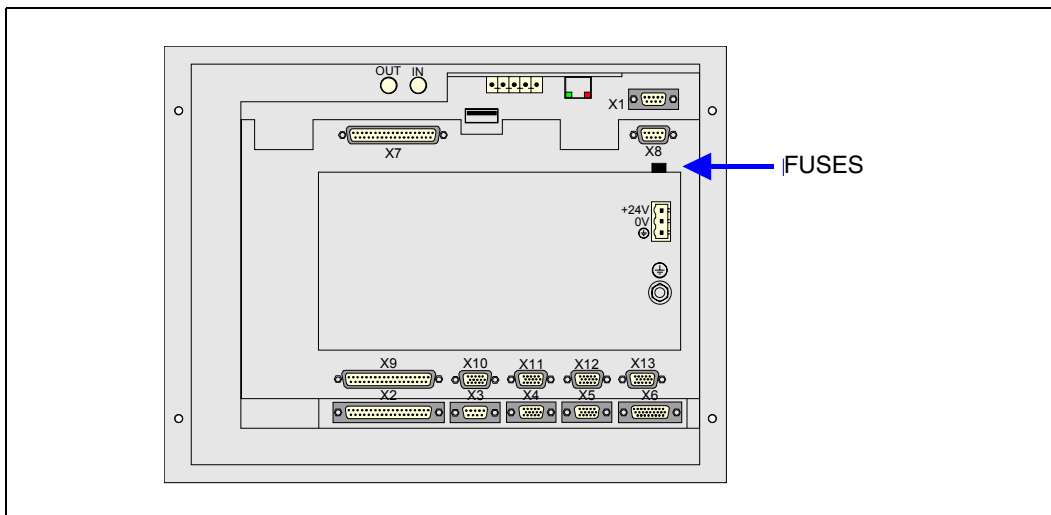 The type of protection fuse depends on the type of monitor. See identification label of the unit itself.

PROTECTIONS OF THE UNIT ITSELF (8055I)

- 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.

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·17·

PRECAUTIONS DURING REPAIRS

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

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

SAFETY SYMBOLS

• Symbols that may appear in the manual.

Symbol for danger or prohibition.
It indicates actions or operations that may cause damage to people or to units.

Warning or caution symbol.
It indicates situations that could be caused by certain operations and the actions to take to prevent them.

Mandatory symbol.
It indicates actions or operations that MUST be carried out.

Information symbol.
It indicates notes, warnings and advises.

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

RETURNING CONDITIONS

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

1. Get a cardboard box whose 3 inside dimensions are at least 15 cm (6 inches) larger than those of the unit itself. The cardboard being used to make the box must have a resistance of 170 kg. (375 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 of the failure.

4. Protect the unit wrapping it up with a roll of polyethylene or with similar material.

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 packaging tape or with industrial staples.

Returning conditions

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

ADDITIONAL NOTES

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 made properly.

To prevent electrical shock at the central unit of the 8055 CNC, use the proper mains AC connector at the power supply module. Use 3-wire power cables (one for ground connection).
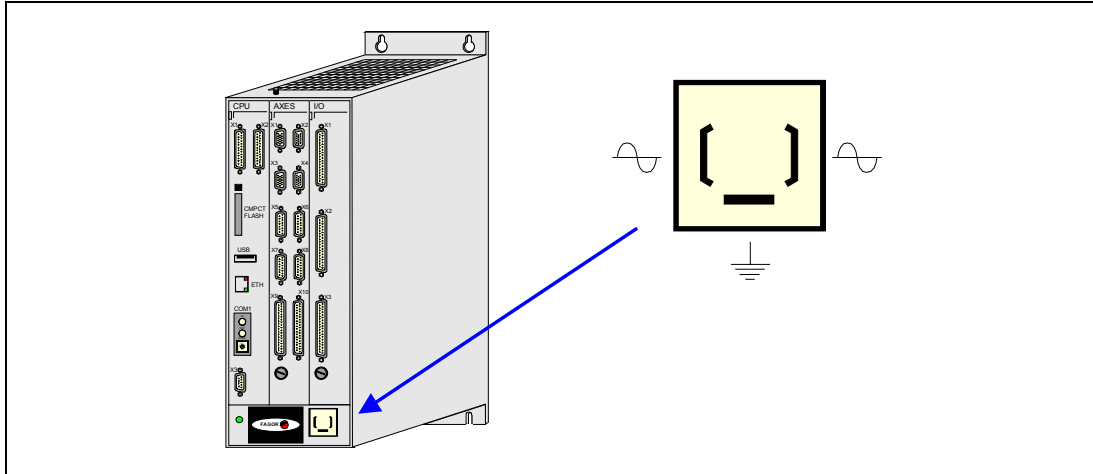
To prevent electrical shock at the monitor of the 8055 CNC, use the proper mains AC connector (A) with 3-wire power cables (one of them for ground connection).
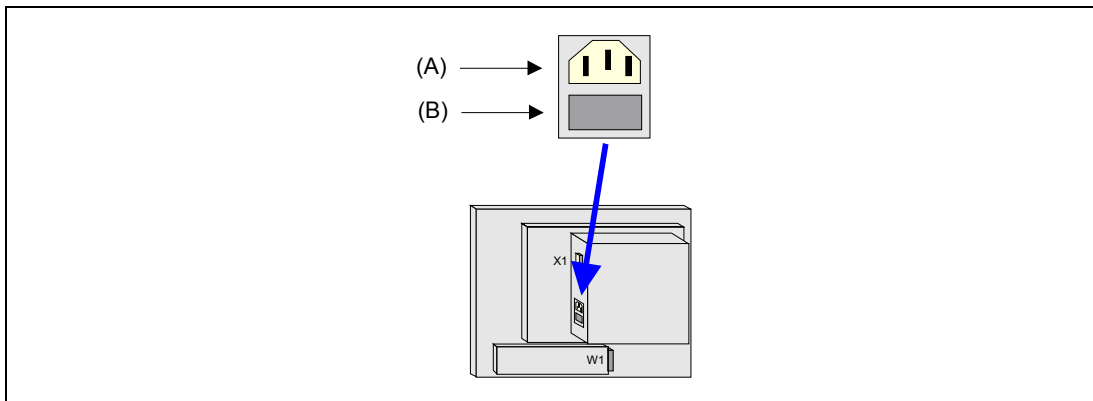
Before turning on the monitor of the 8055 CNC and verifying that the external AC line (B) fuse of each unit is the right one. See identification label of the unit itself.

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

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

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.

MC Manual

Directed to the end user.

It describes how to operate and program in MC mode.

It contains a self-teaching manual.

TC Manual

Directed to the end user.

It describes how to operate and program in TC mode.

It contains a self-teaching manual.

MCO/TCO model

Directed to the end user.

It describes how to operate and program in MCO and TCO mode.

Examples-M manual

Directed to the end user.

It contains programming examples for the M mode.

Examples-T manual

Directed to the end user.

It contains programming examples for the T mode.

WINDNC Manual

It is directed to people using the optional DNC communications software.

It is supplied in a floppy disk with the application.

WINDRAW55 Manual

Directed to people who use the WINDRAW55 to create screens.

It is supplied in a floppy disk with the application.

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

GENERAL CONCEPTS
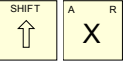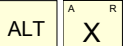
1.1   Keyboard
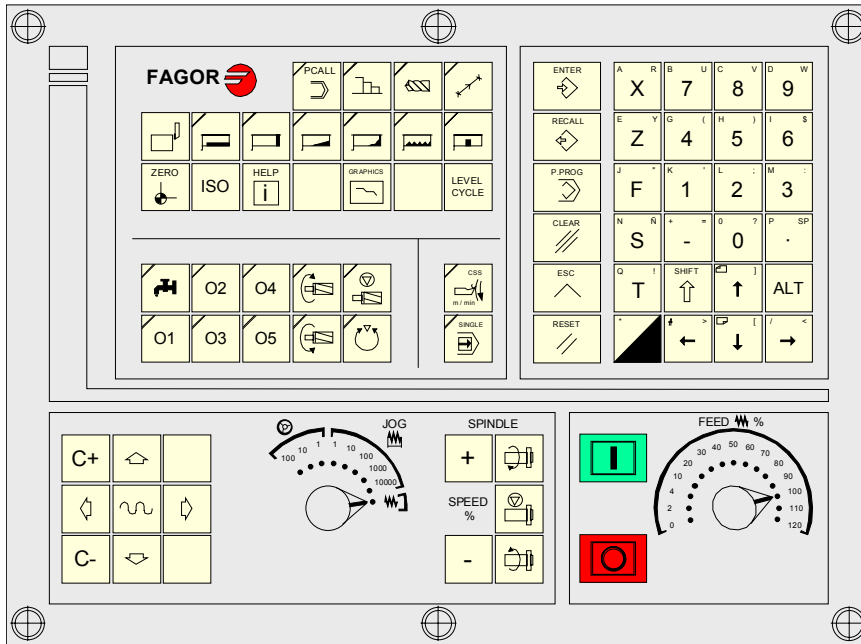
Alphanumeric keyboard and command keys
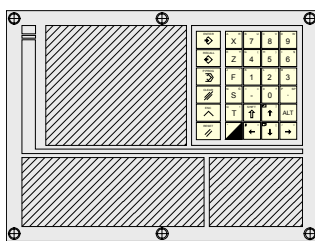
X	Select the X character.
SHIFT ⇧ X	Select the A character.
ALT X	Select the R character.

Specific keys of the TC model
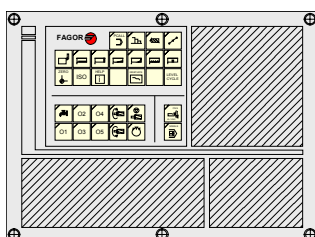
These keys may be used for:

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

JOG keys
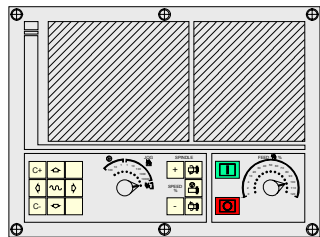
These keys may be used for:

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

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.*

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.

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.

> ℹ️ 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 or TCO 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, TCO 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

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

1.3   Power-up

| SHIFT ⇧ | RESET ⁄⁄ | 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:

```
15:28:42    SBK  P000002 IN POSITION

X    00044.000   φ         T 02  ▽
     REFERENCE  ZERO   X 0000.000       D 12

Z    -00443.331              CHANGE POSITION
     REFERENCE  ZERO   Z 0000.000    X  25.000    Z  85.000

S         0    00025.000     S 0100
               00000.013
               00014.480     ◎    % 115

                             SMAX 1000    RANGE 1
F 0100.000      % 080        ⟳    020.0000
```

| SHIFT ⇧ | ESC ⋀ | There are 2 work modes: TC work mode and T work mode. To change from one work mode to another, press the key sequence [SHIFT] [ESC]. |

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

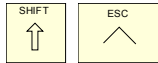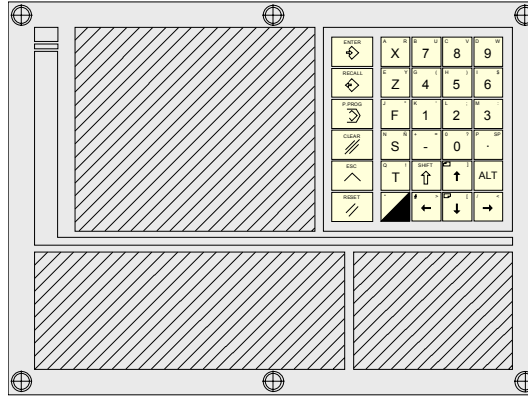## 1.4   Working in T mode with the TC keyboard

There are 2 work modes: TC work mode and T work mode. To change from one work mode to another, press the key sequence [SHIFT] [ESC].

The TC keyboard is designed to also be able to work in T mode. In T mode, use the alphanumeric keyboard and the keys that replace the softkeys F1 through F7.
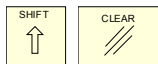
Alphanumeric keyboard:

The keys that replace the softkeys F1 through F7 are:
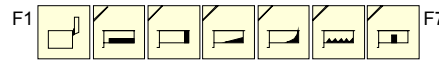
1.5   Video off

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.

1.6   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.

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

OPERATING IN JOG MODE

The standard screen of the TC mode is the following:

```
┌──────────────────────────────────────────────────────────┐
│ 15:28:42   │SBK│ P000002 IN POSITION                       │
├────────────────────────────┬─────────────────────────────┤
│                            │                             │
│  X    00044.000  φ         │  T 02  [V]                   │
│  REFERENCE  ZERO  X 0000.000│             D 12            │
│                            ├─────────────────────────────┤
│  Z   -00443.331            │  CHANGE POSITION            │
│  REFERENCE  ZERO  Z 0000.000│  X 25.000    Z 85.000      │
│                ┌───────────┤                             │
│                │ 00025.000 │  S 0100                     │
│  S        0    │ 00000.013 │                             │
│                │ 00014.480 │  [▽]  % 115                 │
│                └───────────┤                             │
│                            │  SMAX 1000    RANGE 1       │
│  F 0100.000     % 080      │  [◯]  020.0000             │
└────────────────────────────┴─────────────────────────────┘
```

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


```
┌──────────────────────────────────────────────────────────┐
│ 15:28:42   │SBK│ P000002 IN POSITION                       │
├────────────────────────────┬─────────────────────────────┤
│ M0                         │ G01  G18                    │
│ (MSG " " )                 │                             │
│ (IF  P102  EQ  1  GOTO N10)│ M41                         │
│ (IF  P101  EQ  0  RET)     │                             │
│ M3                         │                             │
│ (RET)                      │ PARTC  : 000000             │
│ N10  M4                    │ CYTIME : 00:00:00:00        │
│ (RET)                      │ TIMER:  : 000000:00:00      │
├────────────────────────────┴─────────────────────────────┤
│ COMMAND        ACTUAL       TO GO      FOLLOWING ERROR    │
│                                                          │
│ X 00020.000   X 00020.000  X 00000.000   X 00000.000     │
│ Z 00089.520   Z 00089.520  Z 00000.000   Z 00000.000     │
│ C 00014.480   C 00014.480  C 00000.000   C 00000.000     │
├──────────────────────────────────────────────────────────┤
│ THEORETICAL    RPM          M/MIN                        │
│ S    0.0000   S    0.0000  S    0.0000   S    0.0000     │
│                                                          │
│ U  00025.000    B  00000.013                             │
└──────────────────────────────────────────────────────────┘
```
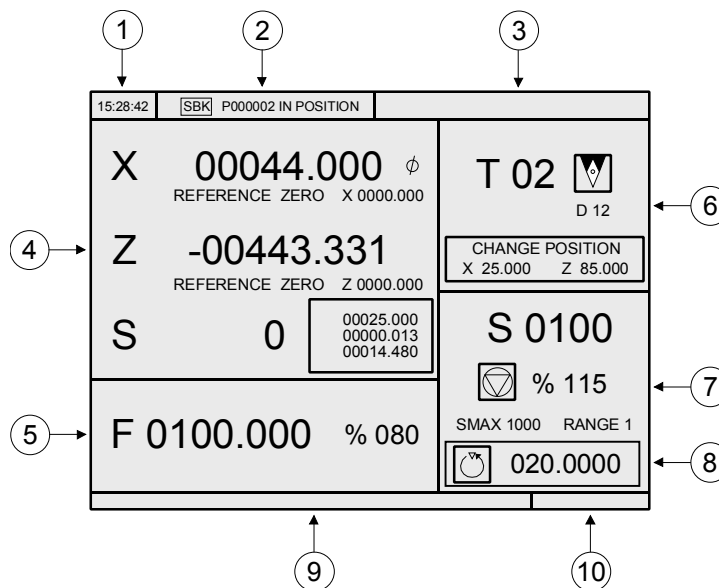
2.1 Introduction

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 coordinates of the auxiliary axes that are defined.

   The "C" axis will only be displayed when it is enabled (G15) and may be governed manually with the jog keys [C+] and [C-]. Being the X-C plane active, the coordinates shown correspond to the transformed coordinates; not to the machine coordinates.

   The actual spindle rpm (S symbol) or the actual rpm of the second spindle (S2 symbol).
5. The information shown in this window depends on the position of the left 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.

Here are all the possible cases.
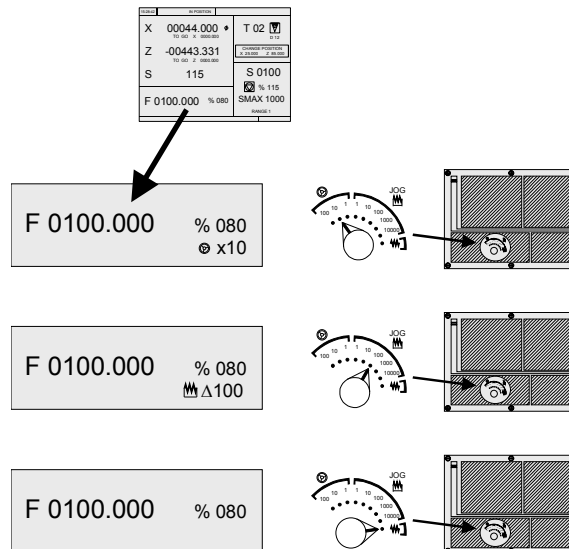
6. This window shows, in large characters, the selected tool number "T" and, in small characters, the "D" offset associated with the tool. If the tool number and the offset number are the same, the CNC will not show the "D" value. The window also shows a drawing of the location code (shape) associated 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.
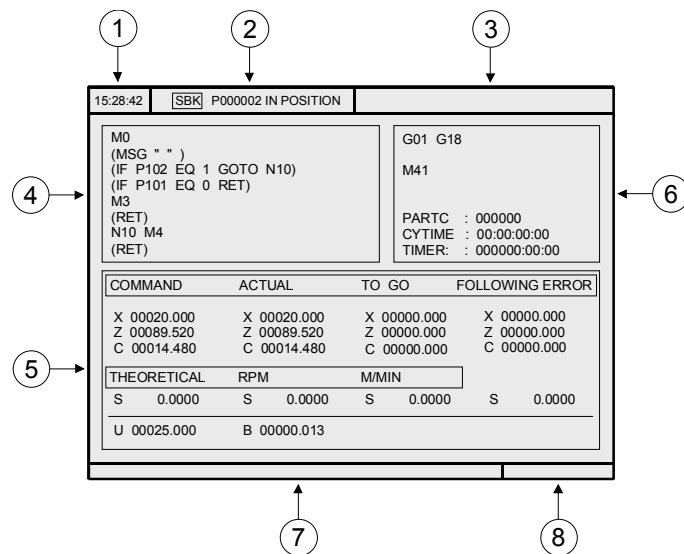
The window is only displayed when there are more than one active message.

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.

2.1.2 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, Z, C axes have the following fields:

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

The auxiliary axes only show the real current position of the axis.

Being the X-C plane active, the coordinates shown in the "Actual" field correspond to the transformed coordinates; not to the machine coordinates.

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" format.

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

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

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 [↑] [↓] [PAGE UP] [PAGE DOWN] keys are used to move around the messages. The [ESC] key is used to close the window.
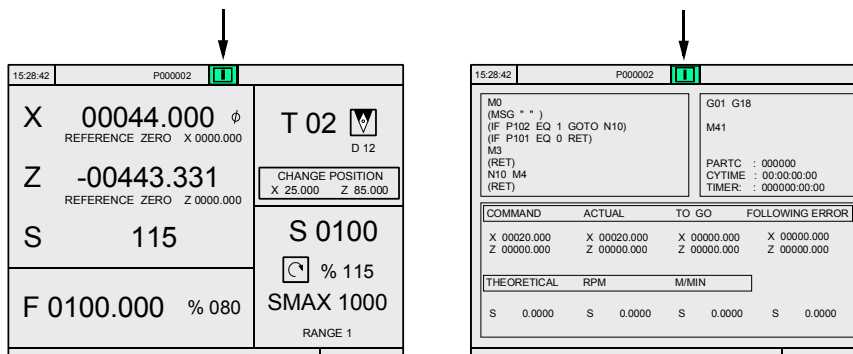
The window is only displayed when there are more than one active message.

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.

2.1.3   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.

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

·38·

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.

2.2.2   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.

2.2.3   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.

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

2.3 Machine reference (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].
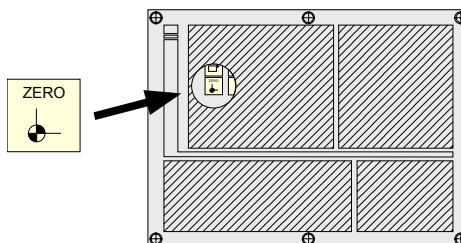
The CNC requests confirmation of the command (text 48 of program 999997). Press [START], the CNC will execute the home search subroutine defined by the OEM in general machine parameters P34 (REFPSUB).
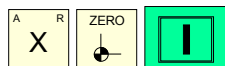
> **i** *After searching home this way, the CNC will maintain the part zero or zero offset active at the time. A home search subroutine (general machine parameter P34 other than 0) must be defined when using this method. Otherwise, the CNC will display the corresponding error.*

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).

X ZERO **I**   It homes the X axis.
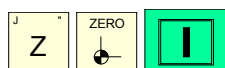
Z ZERO **I**   It homes the Z axis.

> **i** *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.*

2.4   Zero offset table

It is possible to manage the zero offset table from the conversational mode (G54.... G59, G159N7 ... 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	V
PLC	0.0000	0.0000	0.0000
G54	0.0000	0.0000	0.0000
G55	0.0000	0.0000	0.0000
G56	0.0000	0.0000	0.0000
G57	0.0000	0.0000	0.0000
ΔG58	0.0000	0.0000	0.0000
ΔG59	0.0000	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. Two origins may be active simultaneously, one absolute (G54 ... G57, G159N7 ... G159N20) and another incremental (G58-G59).

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 magnifying glass is placed on top of an offset (G54 ... G59, G159N7 ... G159N20), editing start on the first axis of that offset.

- Load the active zero offset into the table.

  Position the magnifying glass over the offset you wish to define (G54 ... G59, G159N7 ... G159N20) and click on 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.

  Position the magnifying glass over the offset that you wish to erase (G54 ... G59, G159N7 ... G159N20) and click on 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.

2.5   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.
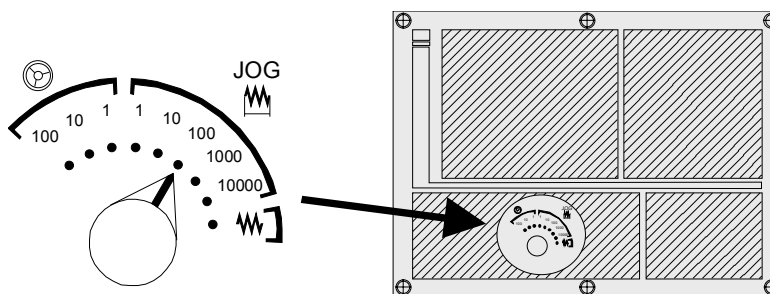- With a path handwheel, no axis is highlighted; but it is in path-jog.

2.5.1   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]

2.5.2   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
1000	1.000 mm or 0.1000 inches
10000	10.000 mm or 1.0000 inches

2.5.3 Continuous jog

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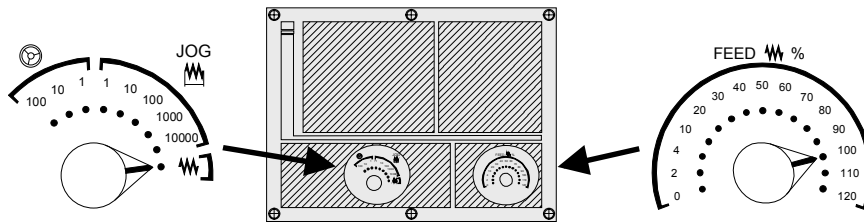
The continuous jog must be made one axis at a time. To do that, press the JOG keys for the direction of the axis to be jogged.

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.

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

% 115

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

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

F (mm/min) = F (mm/rev) x S (rpm) = 2 x 500 = 1000 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.*

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: [←] [→] [↑] [↓].

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

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	Deactivated	Only the axis and in the indicated direction
	Activated	Both axes in the indicated direction and along the indicated path
Incremental jog	Deactivated	Only the axis, the selected distance and in the indicated direction
	Activated	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

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

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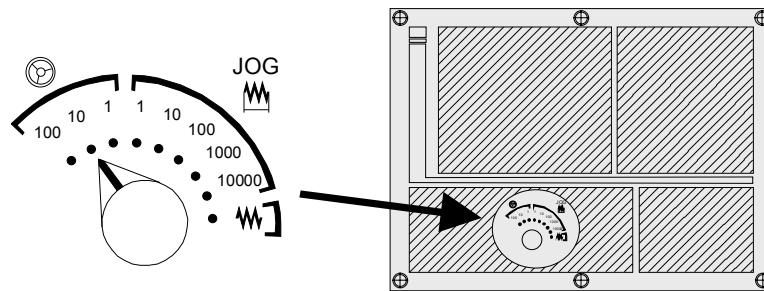
The positions available are 1, 10 and 100; they indicate the multiplying factor being applied besides the internal x4 to the feedback pulses supplied by the electronic handwheel.

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

> **i**
>
> *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.

2.5.6 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. Usually, this feature is turned on and off using an external push button or key configured for that purpose.*

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

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

FAGOR

FAGOR AUTOMATION

**CNC 8055
CNC 8055i**

·**TC**· O**PTION**
S**OFT: V02.2X**

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.

> **i** *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:
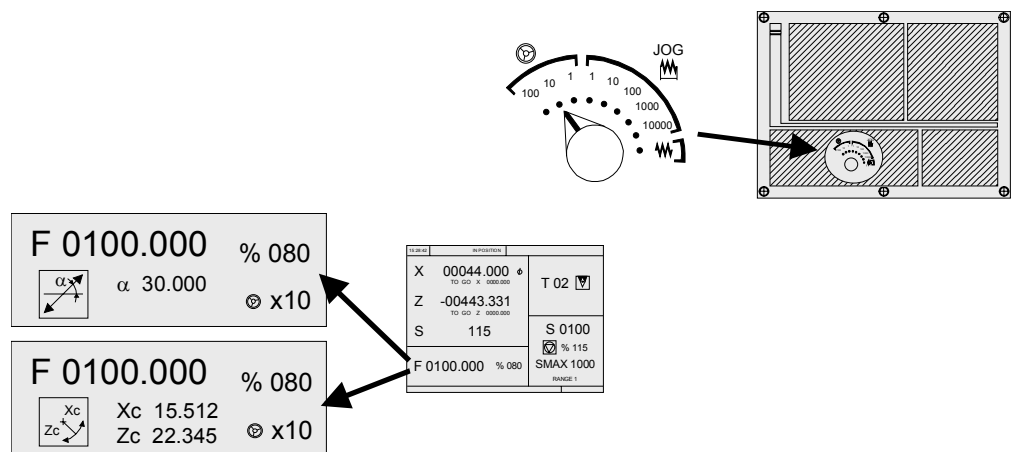
For a linear movement (top figure), the path angle must be defined and for an arc (bottom figure), the center coordinates must be indicated. To define these variables, press the [F] key and then one of these keys: [⬅] [➡] [⬆] [⬇].

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.
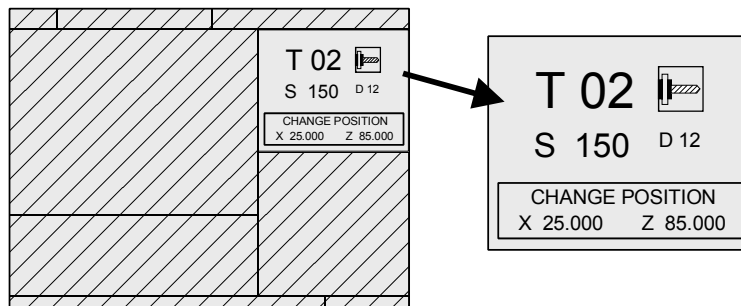
This window displays the following information:

- 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 [↑ ↓].

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.

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

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.).

2.6.2 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.

Arithmetic parameter P290.
Change position in X.
Arithmetic parameter P291.
Change position in Z.
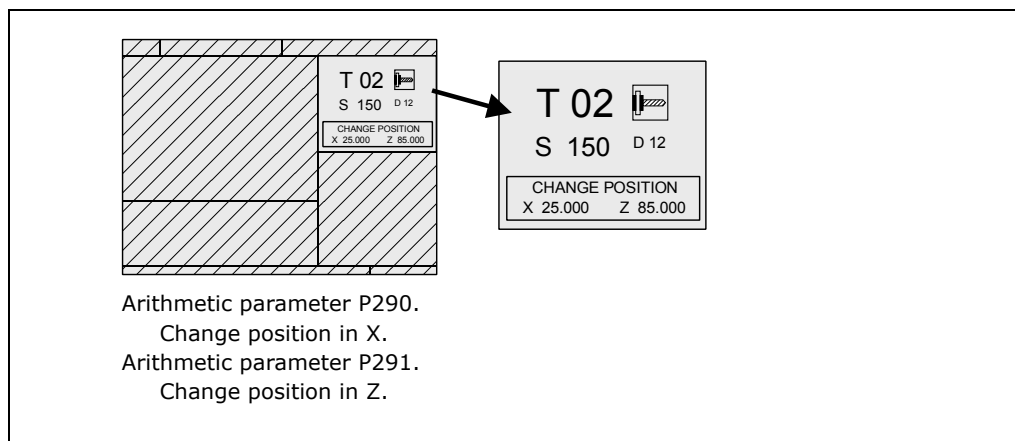
In subroutine 55 of the previous section, the line setting the movement to the tool change point must be modified:

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 [◄] [►] [▲] [▼] 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. The tools may be calibrated with or without using a probe.

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

The calibration mode can have several editing levels. The second level will only be available when using a table-top probe installed on the machine.

LEVEL
CYCLE

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 [LEVEL CYCLE] key or the [page up] and [page down] keys to scroll up and down through the various levels.

What can be done in tool calibration mode

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]).

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

·52·

2.7.1 Define the tool in the tool table (level 1)

When accessing this level, the CNC shows the following screen.
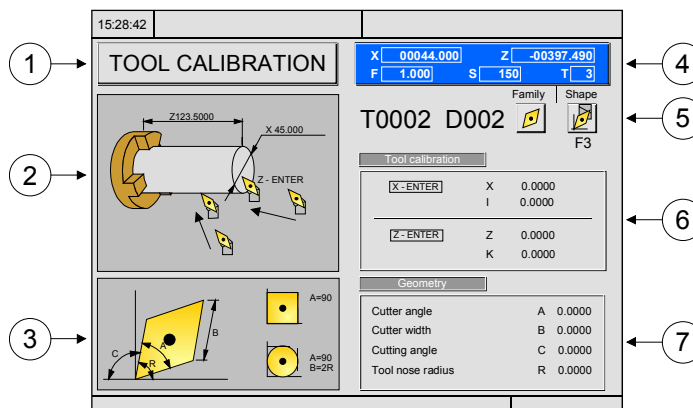
1. Indicating the selected work mode: "Tool calibration".

2. Graphic assistance for tool calibration.

3. Help graphics for defining tool geometry.

4. Current machine status.

   Real X Z coordinates, real axis feedrate F, real spindle speed S and currently selected tool T.

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.

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.

X	Tool's X dimension (in radius).
Z	Tool's Z dimension.
I	X wear offset (in diameter).
K	Z wear offset.
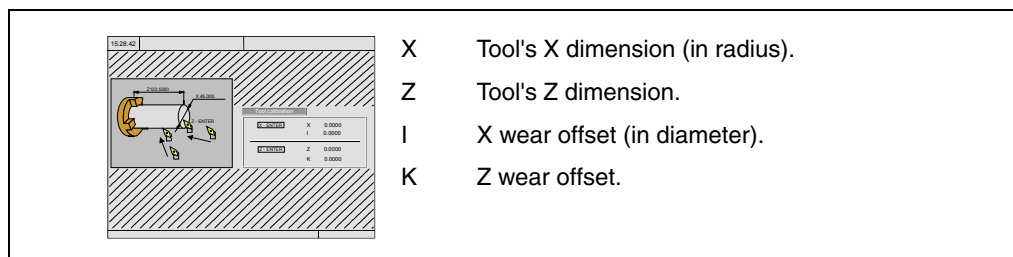
Even if the tool dimensions are known, it is recommended to measure it. See .

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].

2.

OPERATING IN JOG MODE
Tool calibration

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

·54·

Defining the tool type.

Place the cursor over the icon for tool type and press the two-color key. The available tool types are:
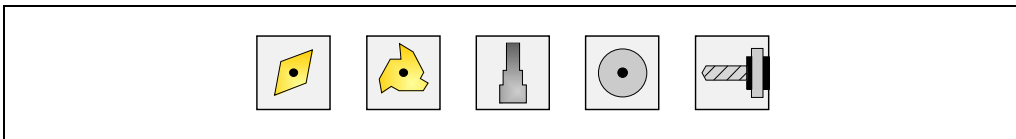
Define the location code of the tool.

Place the cursor over the icon for tool type and press the two-color key. The available tool types are:
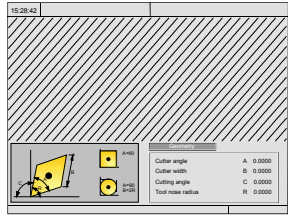
Define the rest of the data related to the tool.

A	Cutter angle.
B	Cutter width.
C	Cutting angle.
R	Tool radius.

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].

2.

OPERATING IN JOG MODE

Tool calibration

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

·55·

2.7.2 Manual tool calibration with/without a probe (level 1)

Before measuring the tool, it must be defined in the tool table. See *"2.7.1 Define the tool in the tool table (level 1)"* on page 53.

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.

Manual tool calibration with/without a probe

In the manual tool calibration cycle, it is possible to calibrate the tool using a master part or a probe. The type of calibration is defined with the following icon: Use the two-color key to select one of them.
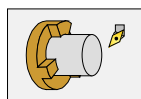
Tool calibration using a master part of known dimensions.
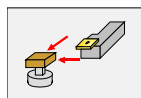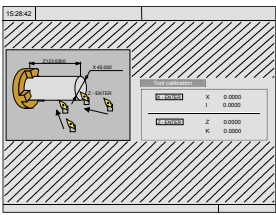
Tool calibration using a probe.

When using a probe for calibrating, one must define the approach distance "Δ", the approach feedrate "F" and the probe side to be probed. If "Δ" not defined, it will take this data from general machine parameter "PRBMOVE". Likewise, If "F" not defined, it will take this data from axis machine parameter "PRBFEED".

Once probing is completed, the screen updates the data.

Define the tool length or modify the length offsets

This window shows the dimensions assigned to the selected tool.

X	Tool's X dimension (in radius).
Z	Tool's Z dimension.
I	X wear offset (in diameter).
K	Z wear offset.

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.

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].

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

·57·

2.7.3   Tool calibration with a probe (level 2)

This calibration level requires the purchase of the right software options purchased and the use of a table-top probe.

Once the cycle has concluded, it updates the tool offset table with the length value X Z of the tool offset that is currently selected. The I and K values are updated to 0.

Defining the cycle data

The following data must be defined.

- Tool number (T) and tool offset (D) to be calibrated.
- Safety distance (Ds) for probe approach.
- Probing feedrate (F).

Probe position.

In this zone, one must indicate whether the cycle assumes the probe position defined in the machine parameters or the position defined in this zone. To select one of them, use the cursor to select the "Machine parameters / Programmable parameters" field and press the two-color key.

TOOL WEAR MEASUREMENT

In this cycle, besides calibrating a tool, it is also possible to measure tool wear.

Using the tool wear measuring operation, the user can define the maximum tool wear value. After several tool wear measuring probing operations, the wear will increase and when it exceeds the set maximum value, the tool will be rejected.

To perform this cycle it is necessary to have a table-top probe, installed in a fixed position on the machine and with its faces parallel to axes X, Y, Z.

When accessing this calibration level, the CNC shows the following information:
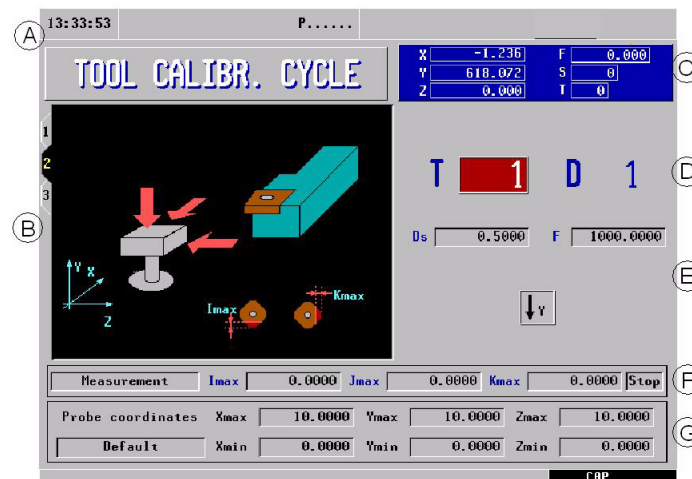
A. Indicating the selected work mode.

B. Graphic assistance for tool calibration.

C. Current machine status.

D. Tool number and associated offset.

E. Calibration data.

F. Type of operation and wear values.

G. Probe position.

This level may be saved as part of a part-program using the [P.PROG] key or executed using the [START] key.

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

Defining the cycle data

The following data must be defined. Not all the data will always be available; the cycle will show the necessary data according to the chosen operation.

- Safety distance (Ds) for probe approach.
- Probing feedrate (F).
- Icon to set the Y axis direction.
- Type of operation:

  The cycle allows doing a measurement or a calibration. To select the desired operation, place the cursor in the "Measurement / Calibration" field and press the two-color key. To take a measurement, define the following data.

  Imax   Maximum tool length wear measurement along the X axis.

  Kmax   Maximum tool length wear measurement along the Z axis.

  Jmax   Maximum tool length wear measurement along the Y axis.

  Stop / Chg Cycle behavior when exceeding the maximum wear permitted. Use the two-color key to select one of them.

  The "Stop" option interrupts the execution for the user to select another tool. With the "Chg" option, the cycle replaces the tool with another one of the same family.

  Measuring is only available when purchasing the software option: "Tool life monitoring".

- Probe position.

  In this zone, one must indicate whether the cycle assumes the probe position defined in the machine parameters or the position defined in this zone. To select one of them, use the cursor to select the "Machine parameters / Programmable parameters" field and press the two-color key.

  Machine parameters: The cycle assumes the probe position defined in the machine parameters.

  Programmed parameters: The cycle assumes the probe position defined in the cycle (Xmax, Xmin, Ymax, Ymin, Zmax, Zmin).

Actions after completing the tool wear measuring cycle

To activate the rejected tool, either because it has been replaced with another one or because it will be used to keep working, the following options will be offered:

1. Go into the tool table in ISO mode and delete the actual (real) life of that tool.

2. Go into the tool table in ISO mode and write the desired value for the actual (real) life of that tool.

   In this case, activating the tool requires the real life to be smaller than the nominal (rated) life value. Otherwise, the tool will appear as expired (worn out) (status = E).

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

2.7.4   Probe calibration (level 3)

This calibration level requires the purchase of the right software options purchased and the use of a table-top probe.

This cycle may be used to calibrate the sides of the table-top probe, installed in a fixed position of the machine whose sides parallel to the X and Z axes. The probe position must be defined in the corresponding machine parameters (PRB*MIN, PRB*MAX).

To execute the cycle, a master tool of known dimensions will be used whose values have been previously entered in the selected offset.

The data obtained in the calibration are updated directly in machine parameters PRB*MIN and PRB*MAX. To do this, program P99998 must be set as OEM.

Defining the cycle data

The following data must be defined.

- Tool number (T) and tool offset (D) used to define the dimensions of the master part.
- Safety distance (Ds) for probe approach.
- Probing feedrate (F).

Probe position.

In this zone, one must indicate whether the cycle assumes the probe position defined in the machine parameters or the position defined in this zone. To select one of them, use the cursor to select the "Machine parameters / Programmable parameters" field and press the two-color key.

2.

OPERATING IN JOG MODE
Tool calibration

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

·60·

2.7.5   Manual tool calibration without stopping the spindle

This feature may be used to manually calibrate the tool on a conversational lathe. It may be used to calibrate tools without having to go back with the tool after performing a turning or facing operation.

Tool calibration is done without having to stop the spindle at each pass. It saves the tool position in each touch and it applies the calibration by measuring the part only once at the end of the process.

Operation

Follow these steps to manually calibrate the tool:

1. After placing the part in the spindle and start the spindle, perform a facing operation.

2. Place the focus on "MEMO Z" and press the two-color key to save the current position (coordinate).

   Besides pressing the two-color key, pressing the [INS] or [-] key also memorizes the current position. Once the position has been saved, enable "APPLY Z".

3. Withdraw the tool along the Z axis.

4. Perform a turning operation, place the focus on "MEMO X" and press the two-color key to save the current position (coordinate).

   Besides pressing the two-color key, pressing the [INS] or [-] key also memorizes the current position.

5. Withdraw the tool along the X axis.

6. Stop the spindle and measure the resulting diameter of the turning operation just done.

   The memorized positions are kept until a new calibration is applied or until quitting the tool cycle. When quitting the cycle, a message warns that the saved data may get lost.

7. After filling the data out, the calibration may be applied on each axis (APPLY X or APPLY Z) or on all the axes whose position has been memorized (APPLY ALL) so it assumes the new calculated offsets. This is also possible with the Y axis.

   It is also possible to delete the memorized position by pressing the [CLEAR] key when the focus is on APPLY X, APPLY Y, APPLY Z or APPLY ALL. This way, a position may be corrected without having to exit the screen.

The following hotkeys may be used to access the data quickly:

• The X, Y, Z keys give access to the fields for the part measurements before machining and to the tool offsets for each axis.

• The M key gives access to the "MEMO" option used to memorize the positions.

• The A key gives access to the "APPLY" option used to apply the calibration.

Set bit 13 of general machine parameter CODISET (P147) = 0 to enable this calibration.

2.

OPERATING IN JOG MODE
Tool calibration

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·**TC**· **OPTION**
SOFT: V02.2X

2.8   Live tool

When a live tool has been selected, the standard screen of the TC mode offers the following information:
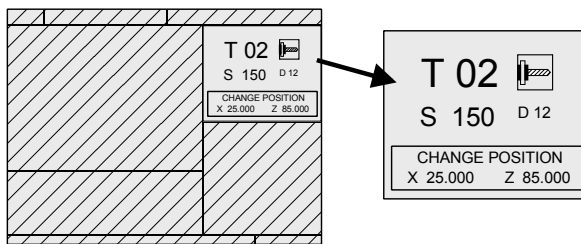
Proceed as follows to select the "S" rpm of the live tool:

1. Press the [T] key to select the "T" field.

2. Press the [S] key or the [↓] key to select the "S" rpm of the live tool.

3. Entering the value manually. Key in the desired value and press [ENTER].

The keys for the live tool are:

Turns clockwise.

Turns counterclockwise.

Stops turning.
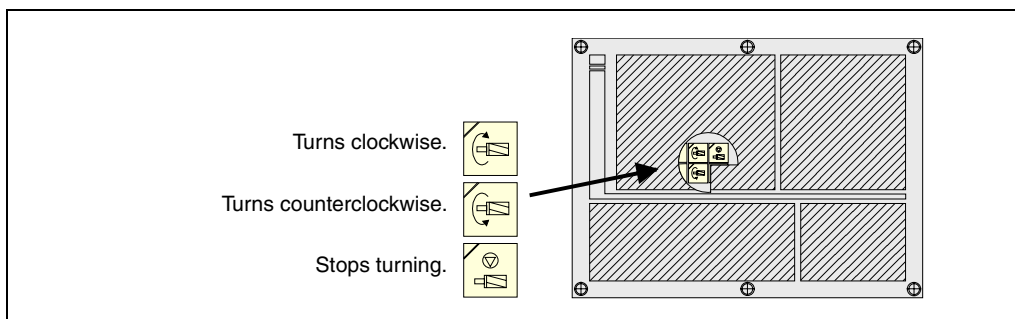
Considerations about the live tool

The following considerations must be borne in mind when the machine has a live tool:

- Set one of general parameters P0 to P9 with a value of 13.
- The location code (shape) of the live tool must be 10, 20 or 30.
- The PLC must manage the keys for the live tool.

Every time one of these keys is pressed, the CNC updates the corresponding register bit.

bit 7 of register 561 (B7 R561)

bit 3 of register 562 (B3 R562)

bit 5 of register 562 (B5 R562)

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

·**62**·

Example of a PLC program to manage the live tool

Here is an example of the portion of the PLC program that must manage the live tool:

( ) = CNCRD (TOOL, R101, M1)

> Assigns the number of the active tool to register R101.

= CNCRD (TOF R101, R102, M1)

> Loads register R102 with the location code of the active tool.

CPS R102 EQ 10 OR CPS R102 EQ 20 OR CPS R102 EQ 30 = M2

> If the active tool is a live tool (if its location code is 10, 20 or 30), it activates the M2 mark.

CUSTOM AND (DFU B7R561 OR DFD M2) = CNCEX1 (M45 S0, M1)

> If while the TC (CUSTOM=1) mode is selected, the "Stop live tool" key is pressed (DFU B7R561) or the live tool is de-selected (DFD M2).

> The PLC "tells" the CNC to execute block M45 S0 (stops the live tool).

CUSTOM AND M2 AND DFU B3R562 = CNCRD (LIVRPM, R117, M1) = CNCWR (R117, GUP100, M1)= CNCEX1 (M45 SP100, M1)

> If in TC (CUSTOM=1) mode, a live tool is selected (M2) and the "live tool clockwise" key is pressed (DFU B3R562).

> The PLC reads in R117 the rpm currently selected for the live tool (LIVRPM) and assigns them to general parameter P100.

> Finally, the PLC "tells" the CNC to execute block M45 SP100 (live tool clockwise at the selected rpm).

CUSTOM AND M2 AND DFU B5R562 = CNCRD (LIVRPM, R117, M1) = CNCWR (R117, GUP100, M1)= CNCEX1 (M45 S-P100, M1)

> If in TC (CUSTOM=1) mode, a live tool is selected (M2) and the "live tool counterclockwise" key is pressed (DFU B5R562).

> The PLC reads in R117 the rpm currently selected for the live tool (LIVRPM) and assigns them to general parameter P100.

> Finally, the PLC "tells" the CNC to execute block M45 S-P100 (live tool counterclockwise at the selected rpm).

2.

OPERATING IN JOG MODE
Live tool

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

2.9   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.
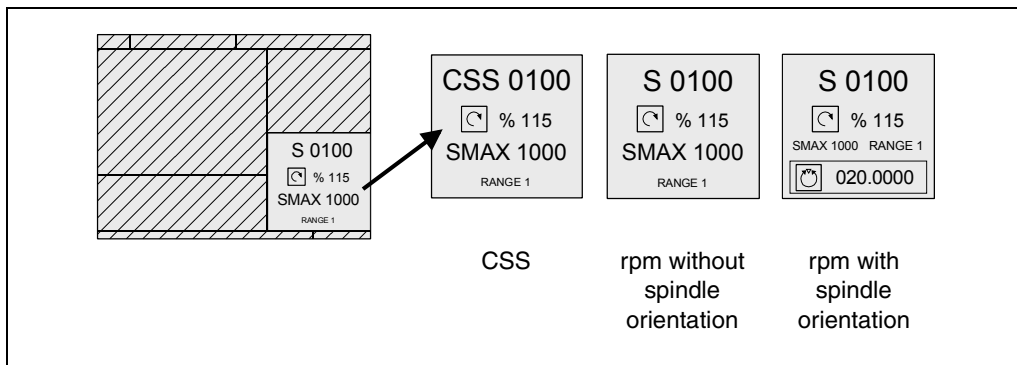
To switch from one mode to the other, press the key:
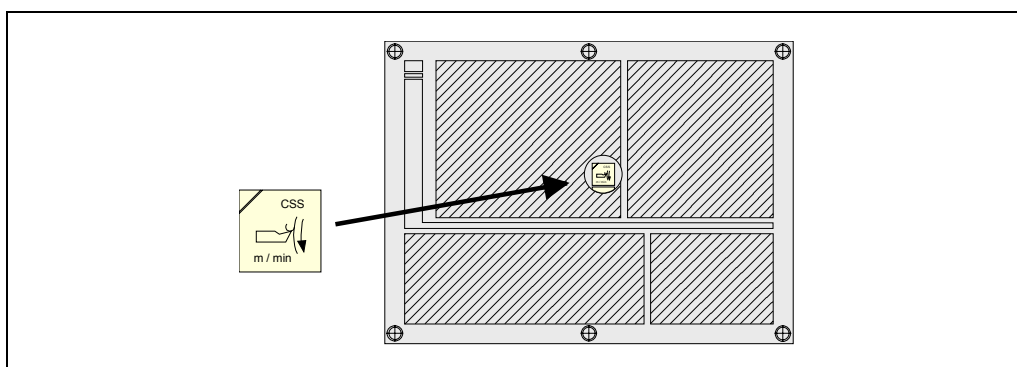
On CNC power-up and after the keystroke sequence [SHIFT] [RESET], the CNC selects the rpm mode. When working at constant surface speed (CSS), the light of the [CSS] key is on.

2.9.1   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.

	Spindle clockwise
	Spindle counterclockwise
	Outside rounding

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.

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

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

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

·66·

2.9.2   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.
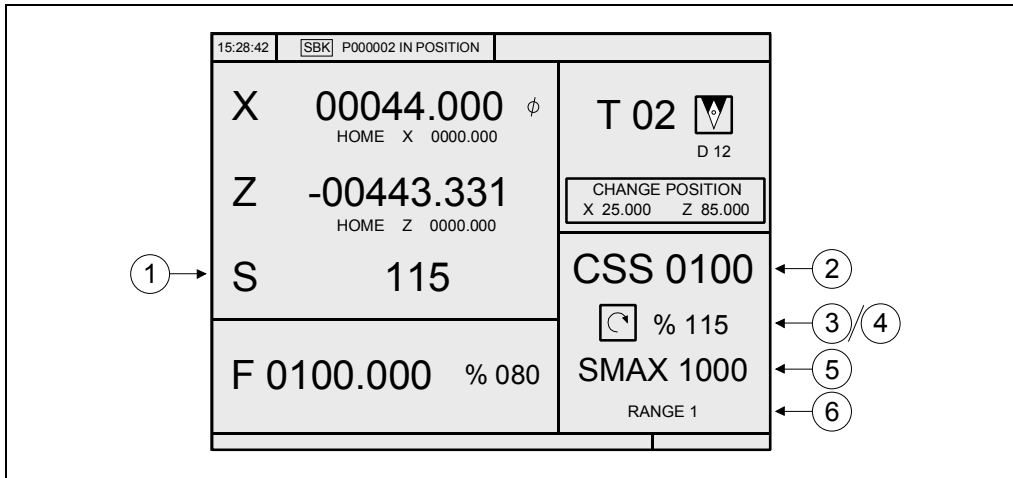
1. Real spindle speed in rpm.

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
⊽	Outside rounding

    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].

> **i** *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.*

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»:

Feedrate = F (mm/rev) x S (rev/min) = 2 x 500 = 1000 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.

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

·68·

2.9.3   Spindle orientation

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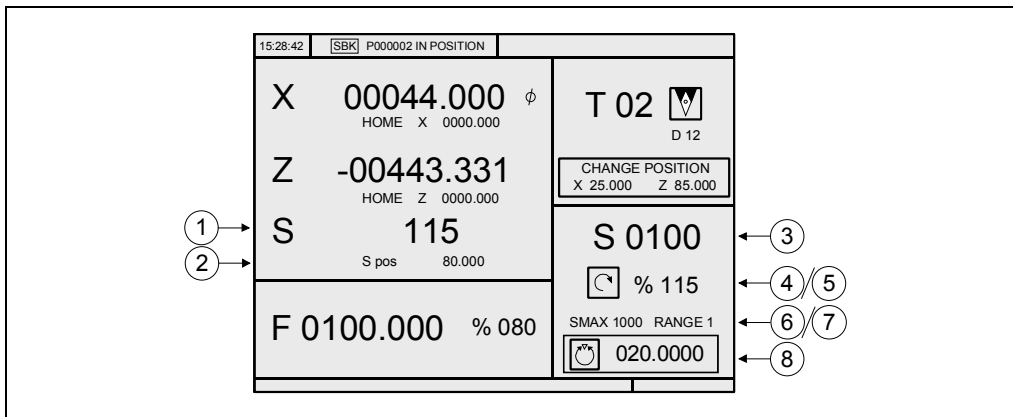
1. Real spindle speed in rpm.

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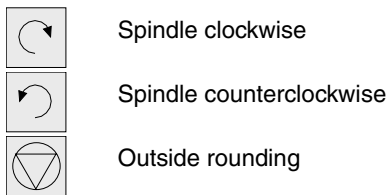
   When working in spindle orientation mode, it always shows the "spindle stopped" symbol.

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.
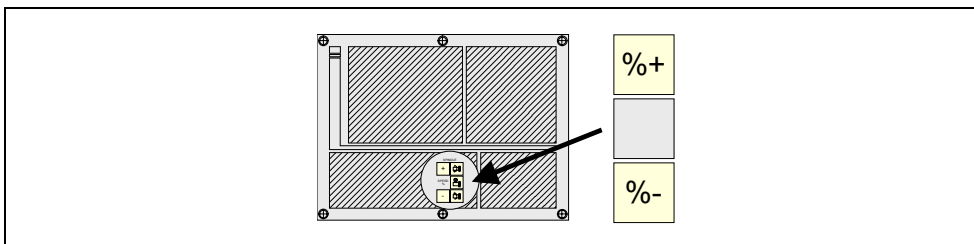
6. Maximum spindle speed in rpm.

   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.

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].

> **i** *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].

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.
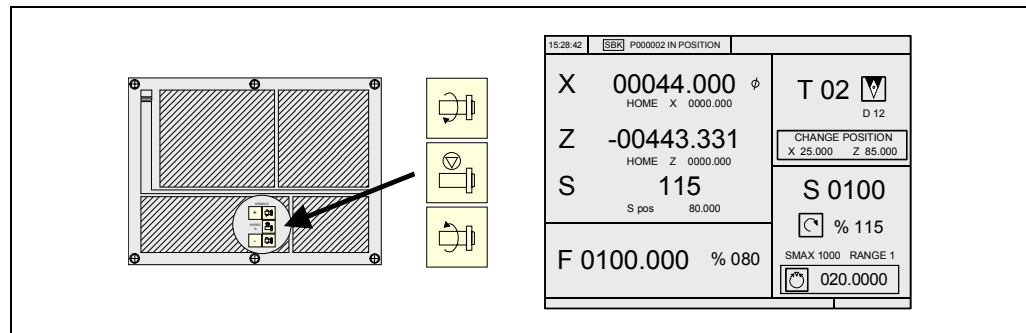
Spindle orientation mode

Press the key for spindle orientation to select this mode:
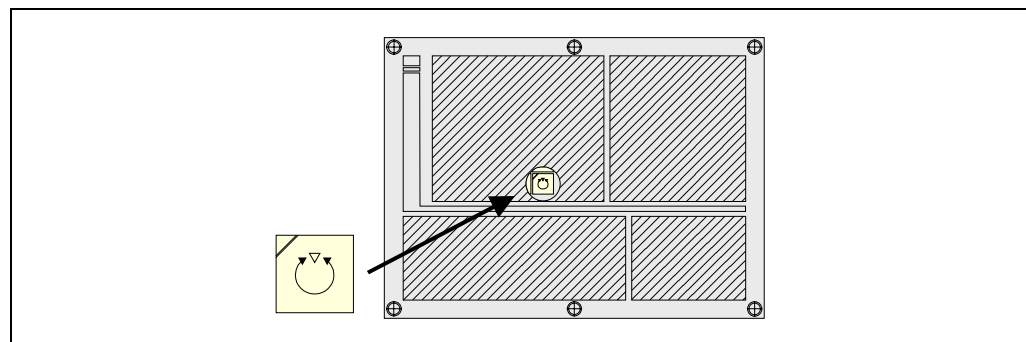
The spindle will stop (if it was turning), it then searches home and finally positions at the angular position indicated on the lower right side of the screen (at 20º in the upper figure).

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

2.10   Controlling the external devices

With this CNC, it is possible to activate and deactivate, via keyboard, up to 6 external devices, for example, the coolant.

The machine manufacturer must use the PLC program to activate and deactivate the devices. The CNC will inform the PLC about the status of each key. The corresponding register bit will be set to 1 when the key is pressed and 0 when it is not pressed.

The register bit for each key is:
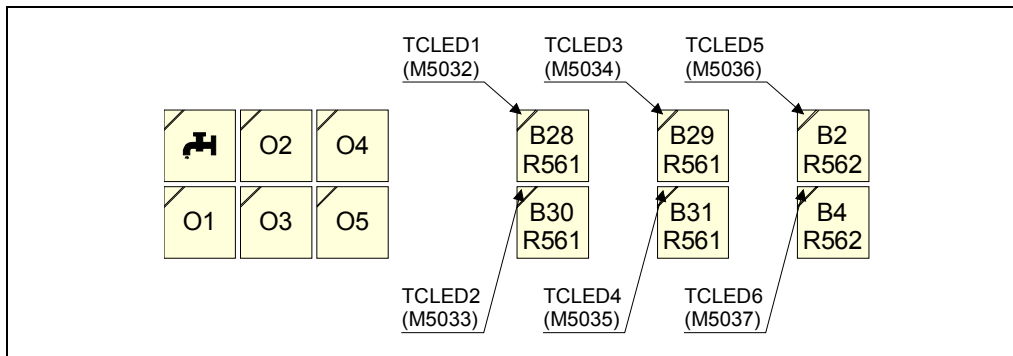
The status of the lamp of each key must be controlled by the machine manufacturer through the PLC program using the input variables TCLED* indicated in the figure.

Examples:

Coolant control:

DFU B28R561 = CPL TCLED1 = CPL O33

Tailstock control (O1). A number of conditions must be met for activating or deactivating the tailstock, such as spindle stopped, etc.

DFU B30R561 AND (Rest of conditions) = CPL TCLED2 = CPL O34

2.11   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.
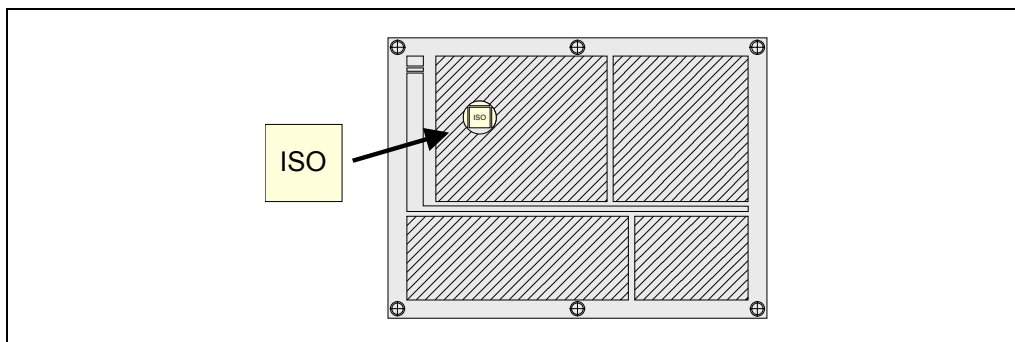
To access the MDI mode, the CNC must be in jog mode and the ISO key must be pressed. The CNC will show a window at the bottom of the standard (or special) screen.
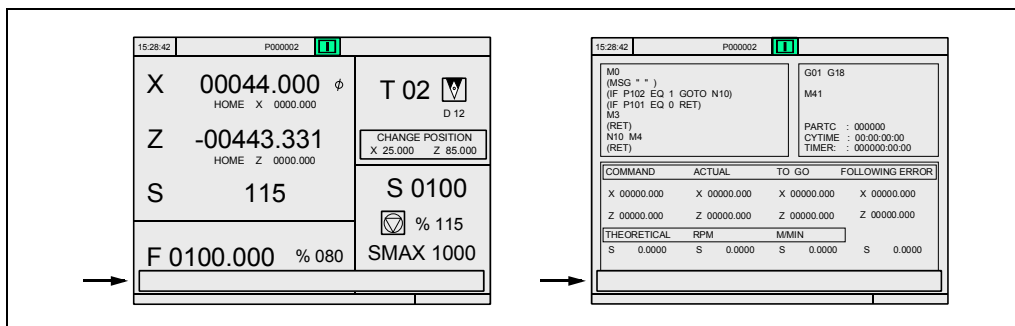
In this window, it is possible to edit a block in ISO code and then execute it, like in MDI in T mode.

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.

Considerations:

- 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.

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

2.

OPERATING IN JOG MODE

ISO management

2.

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

WORKING WITH OPERATIONS OR CYCLES

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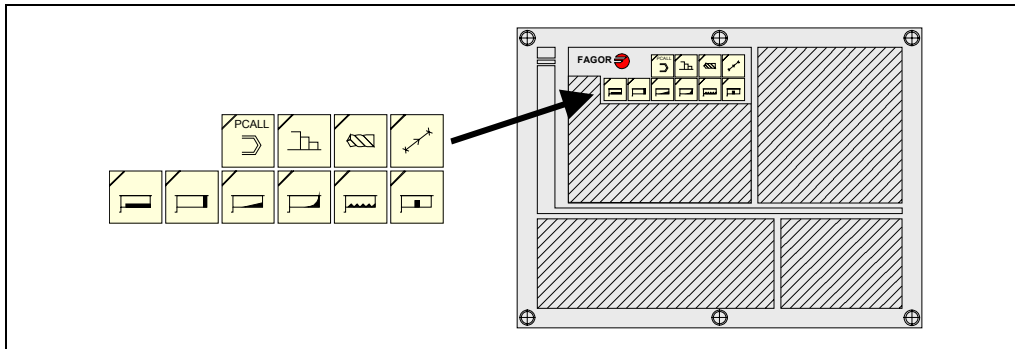
User cycles

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 pressing any other key, the CNC selects the corresponding standard machining cycle, changes the display and turns on the lamp of the key that has been pressed (indicating the selected type.

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

Positioning cycle.	Threading cycle.
Turning cycle.	Grooving cycle.
Facing cycle.	Drilling and tapping cycle.
Taper turning cycle.	Profiling cycle.
Rounding cycle.	

When the machining operation or cycle has several levels, [LEVEL CYCLE] 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 *"6 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 (the one with the lamp on) or the [ESC] key.

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

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.

3.1 Operation editing mode

Once the operation has been selected, the CNC shows a screen like the following:
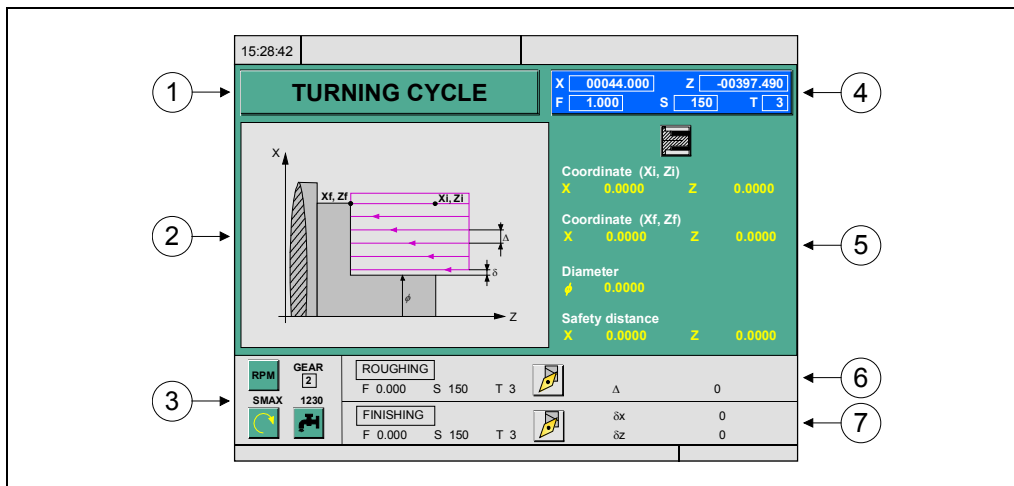
1. Name of the selected operation or work cycle.

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.
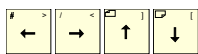
The CNC selects the previous one or the next one.

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.).

3.1.1   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 [↑][↓][←][→] keys.
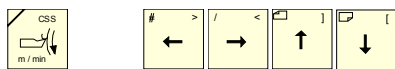
Once the data has been selected, press the [CSS] key or the two-color key to change the work mode.

Spindle range

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

In cycles using a live tool:

Icon for selecting the spindle gear to be used for machining cycles where a live tool is used. Its possible values are:

Value 0:   Gear corresponding to the S value

Value 1:   Gear 1

Value 2:   Gear 2

Value 3:   Gear 3

Value 4:   Gear 4

Maximum spindle speed (S) in rpm

Place the cursor over 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 two-color key to change the icon.
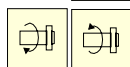
Start the spindle in the desired direction with the JOG keys. The CNC starts the spindle and assumes that turning direction as spindle turning data for the cycle.

Coolant

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

Place the cursor on this data and press the two-color key to change the icon.

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

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

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

3.

WORKING WITH OPERATIONS OR CYCLES
Operation editing mode

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

·78·

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 [TWO-COLOR] 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 [TWO-COLOR] key and press [ENTER]. When de-selecting the finishing, all its data will stay in gray.

Axis feedrate (F).

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

Spindle turning speed (S).

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

Machining tool (T).

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

Pocket machining direction.

Icon to set the machining direction of the roughing tool.

Machining direction.

Some cycles allow the machining direction to be selected (turning direction or facing direction).

Turning direction.   Facing direction.
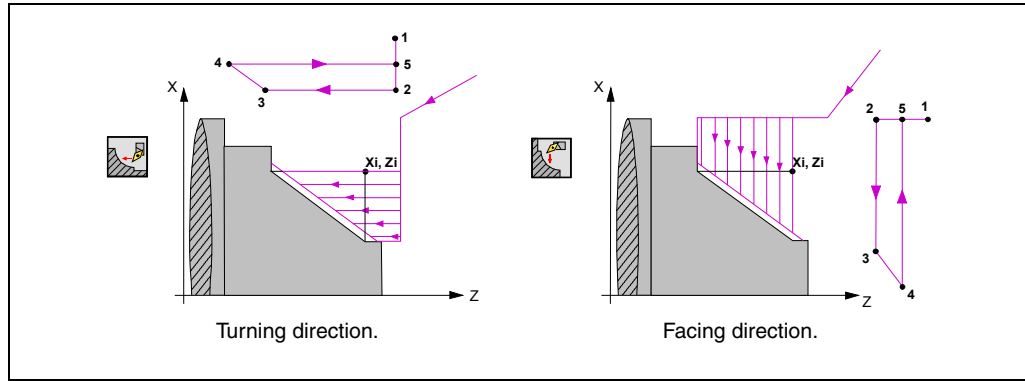
Place the cursor on this icon and press the two-color key. The icon changes and the help graphics are refreshed.

Roughing pass (⊿).

ENTER

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

Finishing stock (δ).

ENTER

The roughing pass is always defined in radius. Place the cursor over 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 0º, or greater than 90º, it issues the corresponding error message. If not programmed, it assumes 90º.

Auxiliary "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 as these are arranged on the list.

Select the corresponding window with the [←][→] keys Use the [↑][↓] keys to move around the window.

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

i   *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.
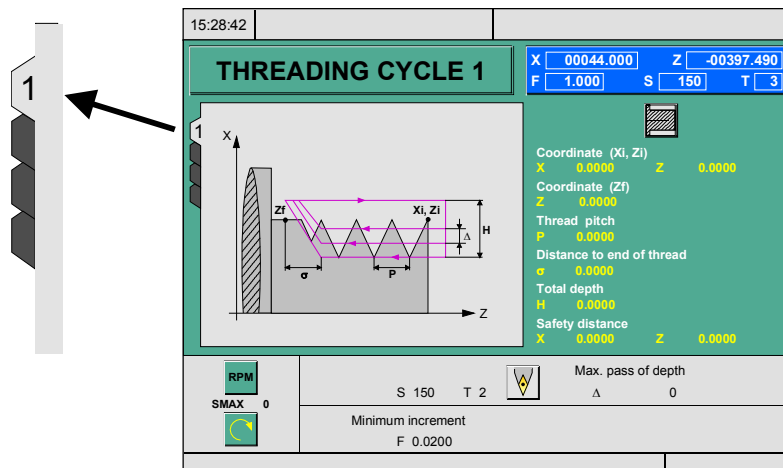
LEVEL CYCLE

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

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.

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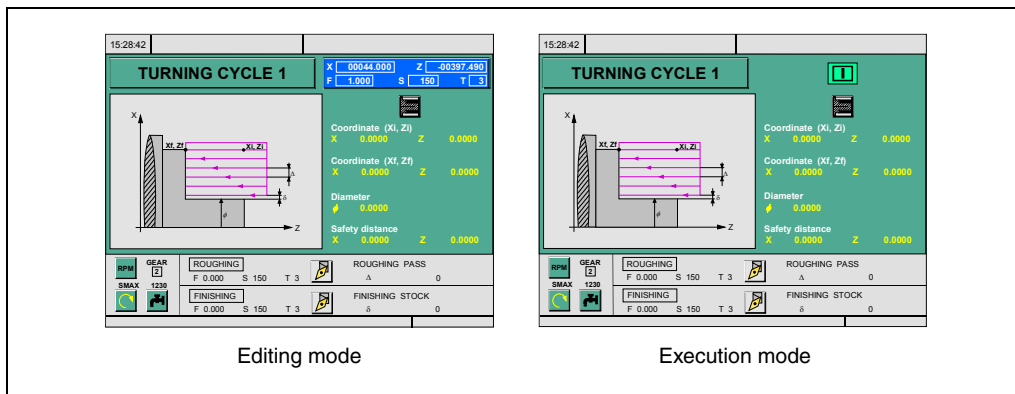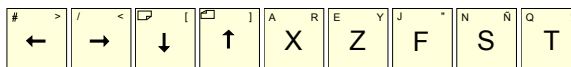
The operation or cycle may be simulated in either mode. To do that, press the [GRAPHICS] key.

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

For further information on simulating and executing cycles, see the chapter *"7 Execution and simulation"*.

FAGOR

FAGOR AUTOMATION

**CNC 8055
CNC 8055i**

·TC· OPTION
SOFT: V02.2X

·82·

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.

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

3.3   Positioning cycle

This key accesses the positioning operation.

This cycle may be defined in two ways:
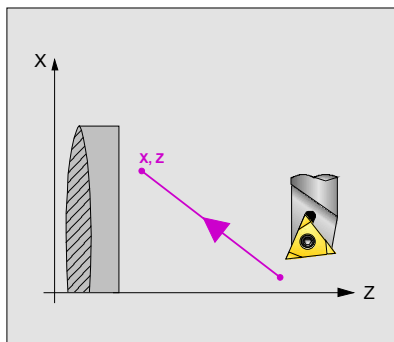
Level 1.

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 two-color key.

All two axes at the same time.

First X and then Z.

First Z and then X.
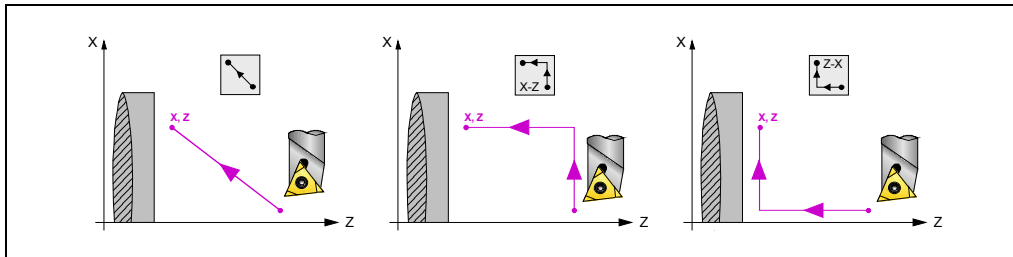
Type of feedrate.

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

Programmed feedrate.

Rapid feed.

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].

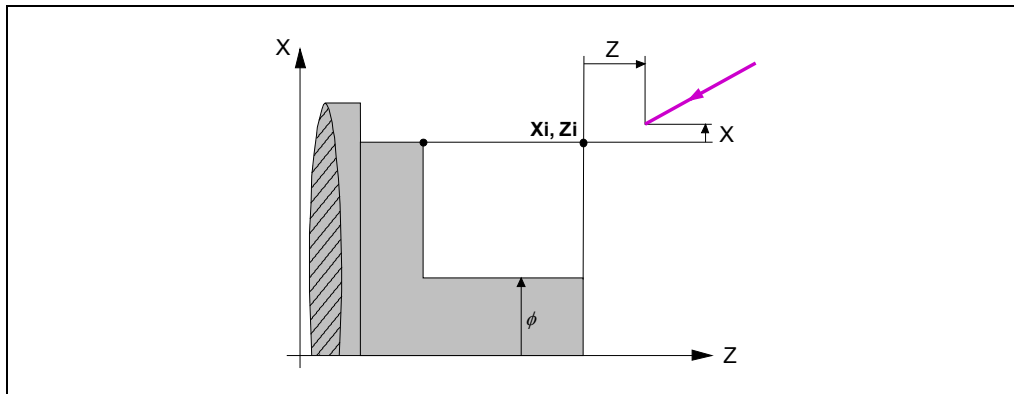   The top right window shows the tool position at all times.

Auxiliary "M" functions.

Auxiliary function "M" is the name given to the functions determined by the manufacturer which allow the different machine devices to be governed. 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 [←][→] keys. Use the [↑][↓] keys to move around the window. To delete a function, select it and press [CLEAR]

The functions will be executed in the same order as these are arranged on the list.

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

3.4   Turning cycle

This key accesses the turning cycle.

This cycle may be defined in several ways:

Turning levels 1 and 2

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.

Turning levels 3, 4 and 5

Level 3. Rectangular pocket on the cylindrical side of the part.

ZC plane

YZ plane

Level 4. Circular pocket on the cylindrical side of the part.

ZC plane

YZ plane

Level 5. ZC / YZ profile pocket.

3.

WORKING WITH OPERATIONS OR CYCLES

Turning cycle

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

·88·
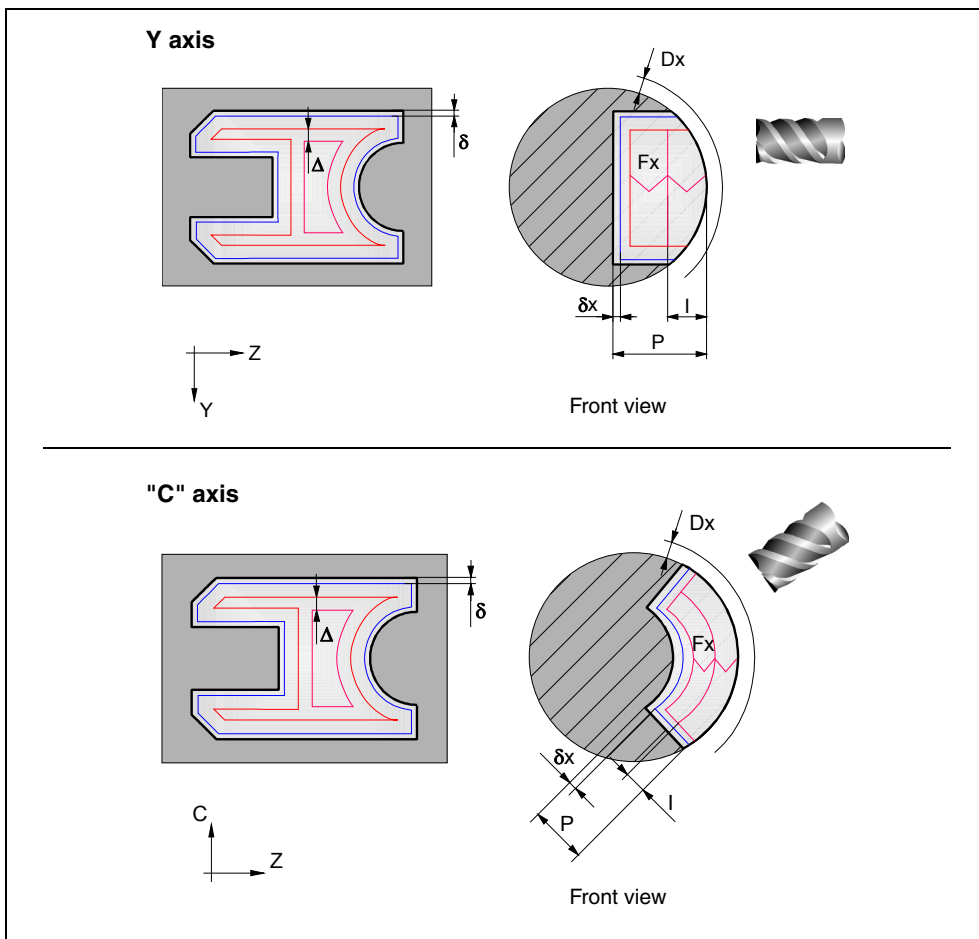
3.4.1   Data definition (levels 1 and 2)

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.

Coordinates of the first point (Xi, Zi) and of the last point (Xf, Zf).

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 over this data, key in the desired value and press [ENTER].

Safety distance.

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

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

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

Type of machining to be carried out on each corner.

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

| Square corner. | Rounded corner. | Chamfered corner. |

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

3.4.2 Data definition (levels 3, 4 and 5)

Level 3:

Icon for selecting the ZC or YZ plane.

Icon to select the position of the starting point.

Z,C / Z,Y: Coordinates of the starting point.

L, H: Pocket dimensions.

α**:** Inclination angle of the rectangular pocket.

W: Angular position of the spindle (in degrees) where the pocket will be milled in the YZ plane.

Icon for selecting the vertex type in the pocket corners:
- Normal vertix
- Rounded vertix
- Chamfered vertix

r / c: Value of the corner rounding or chamfering radius in the rectangular pocket.

Dx: Safety distance on the longitudinal axis (cylindrical side).

Dz: Safety distance on the longitudinal axis (face).

X: Part plane.

P: Total rectangular pocket depth. When programmed with a 0 value, it generates the corresponding error.

I: Penetration step (pass) when roughing:
- If programmed with a positive value, the actual step (pass) will be the one closest to this value so all passes will be identical.
- If programmed with a negative value, the actual pass will be the one programmed and the last pass will be adjusted to the final (remaining) depth.
- If not programmed, it assumes 0.

Fx: Penetration feedrate for roughing and finishing. If not programmed, it assumes 0.

Level 4:

Icon to select the ZC or YZ plane.

Zc, Cc / Center coordinates of the circular pocket.
Zc, Yc:

Rc: Circular pocket radius.

W: Angular position of the spindle (in degrees) where the pocket will be milled in the YZ plane.

Dx: Safety distance on the longitudinal axis (cylindrical side).

Dz: Safety distance on the longitudinal axis (face).

R / X:
- Cylinder radius when it is the ZC plane.
- X coordinate of the part surface when it is the ZY plane.

P: Total circular pocket depth. When programmed with a 0 value, it generates the corresponding error.

3.

WORKING WITH OPERATIONS OR CYCLES

Turning cycle

FAGOR ➔

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

I: Penetration step (pass) when roughing:

- If programmed with a positive value, the actual step (pass) will be the one closest to this value and all the passes will be identical.

- If programmed with a negative value, the actual pass will be the one programmed and the last pass will be adjusted to the final (remaining) depth.

- If not programmed, it assumes 0.

Fx: Penetration feedrate for roughing and finishing. If not programmed, it assumes F/2.

Level 5:

Icon for selecting the machining plane (ZC plane or YZ plane).

Profile program : Number of the part-program where the pocket is defined.

Dx: Safety distance on the longitudinal axis.

X: X coordinate of the part surface when it is the YZ plane.

R: Part radius when it is the ZC plane.

P: Total profile pocket depth. If not programmed or programmed with a 0 value, the CNC will display the corresponding error message.

I: Maximum roughing pass. If not programmed or if programmed with a 0 value, it assumes the value of 75% of the diameter of the roughing tool.

Fx: Machining feedrate on the X axis: if not programmed or programmed with a 0 value, the CNC assumes the roughing F value for the penetration in roughing and the finishing F for the penetration in finishing.

W: Angular position of the spindle (in degrees) where the profile will be machined in the YZ plane (in degrees).

Spindle turning direction
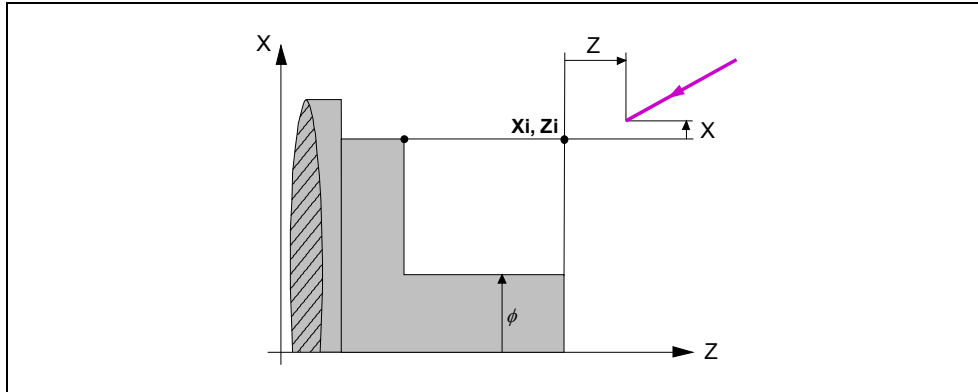
3.4.3   Basic operation (levels 1 and 2)

The machining steps in this cycle are as follows:

1. If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.

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

3. The tool approaches the starting point (Xi, Zi) in rapid, keeping the selected safety distance according to axes 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.
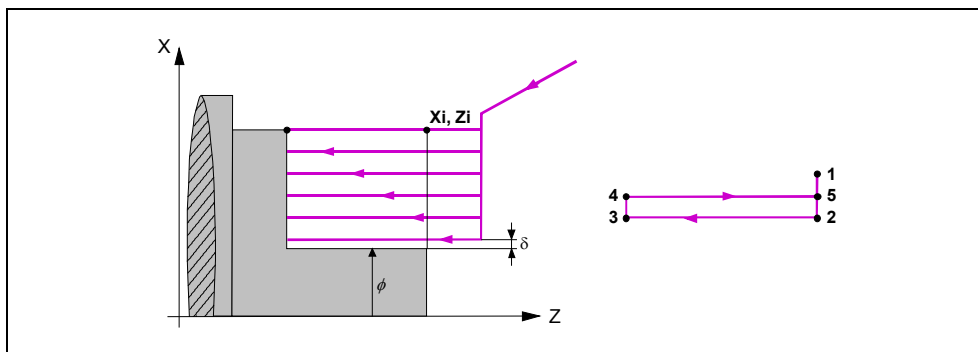
Each turning pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".

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).

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. This means that after approaching the finishing operation will be carried out.

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 Xi and Xf 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:

Facing levels 1 and 2
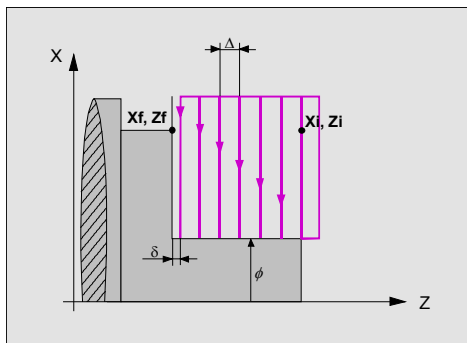
Level 1.

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.

Facing levels 3, 4 and 5
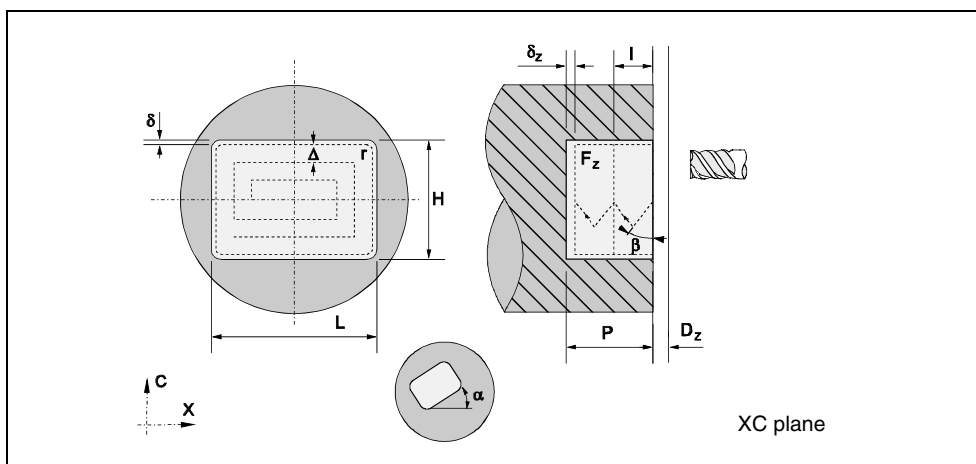
Level 3. Rectangular pocket on the face of the part.

XC plane
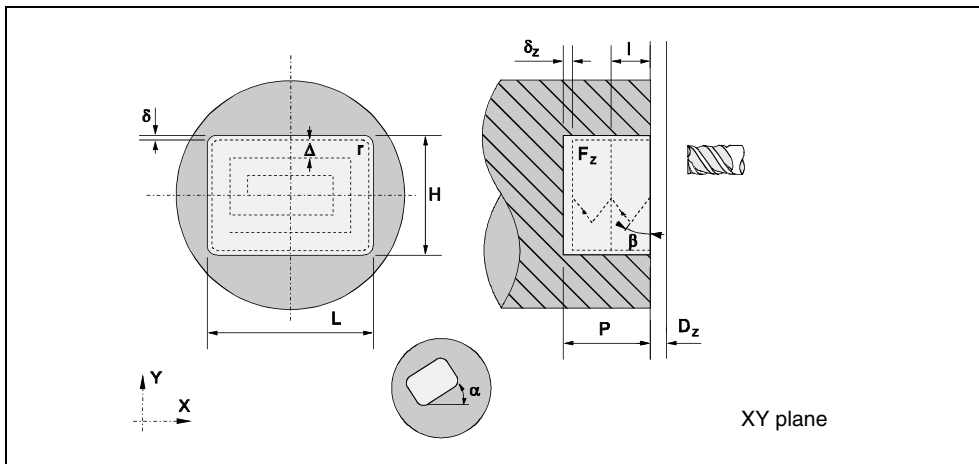
Level 4. Circular pocket on the face of the part.

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

·96·
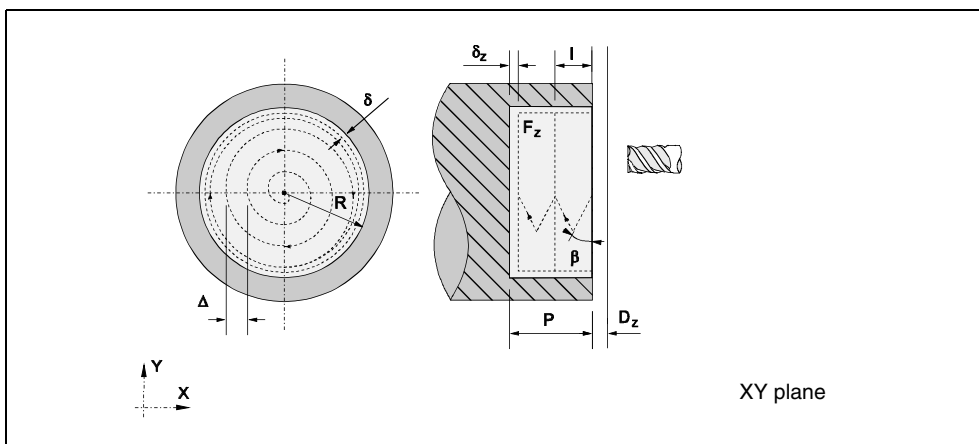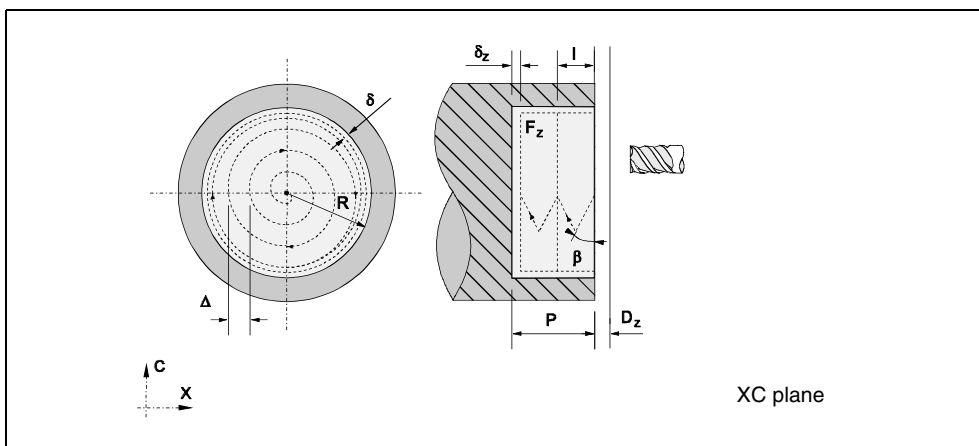
Level 5. XC / XY profile cycle.
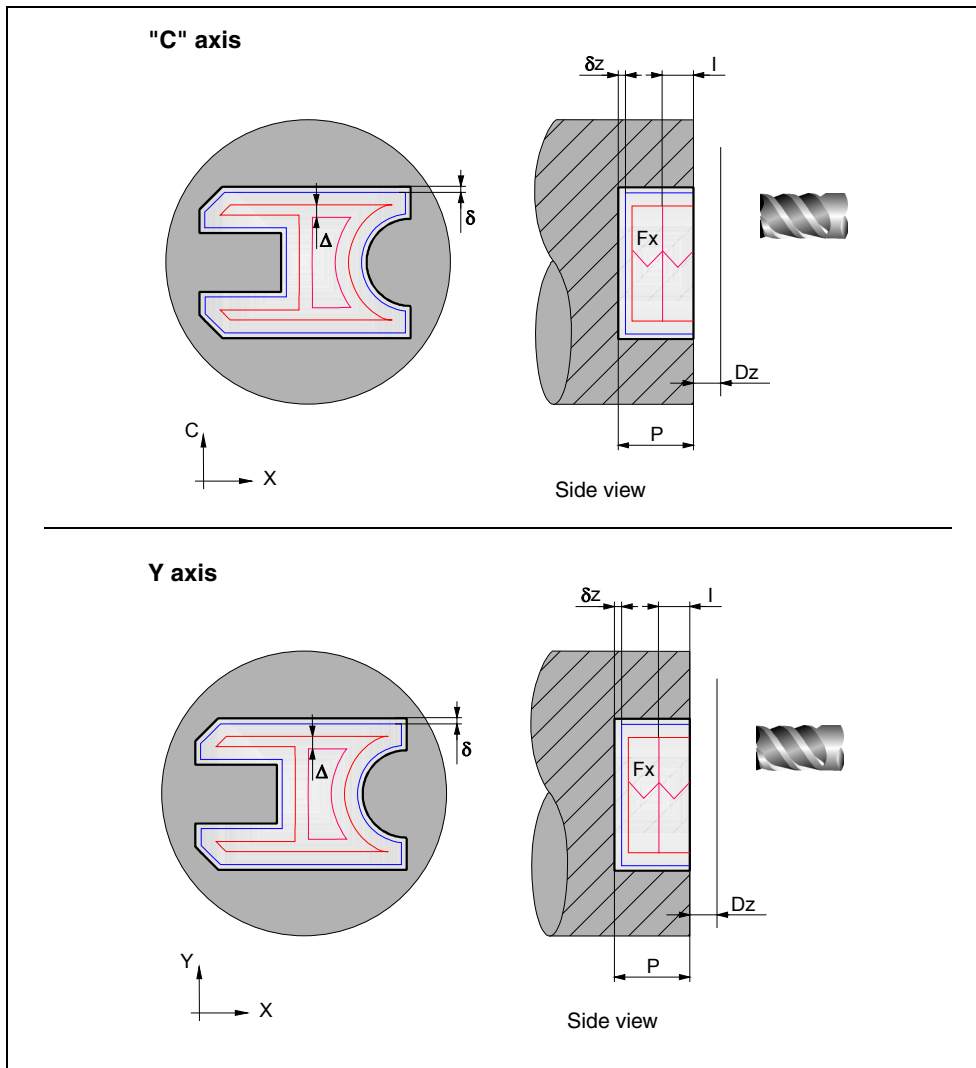
3.5.1   Data definition (levels 1 and 2)

Coordinates of the first point (Xi, Zi) and of the last point (Xf, Zf).

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 over this data, key in the desired value and press [ENTER].

Type of machining to be carried out on each corner.

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

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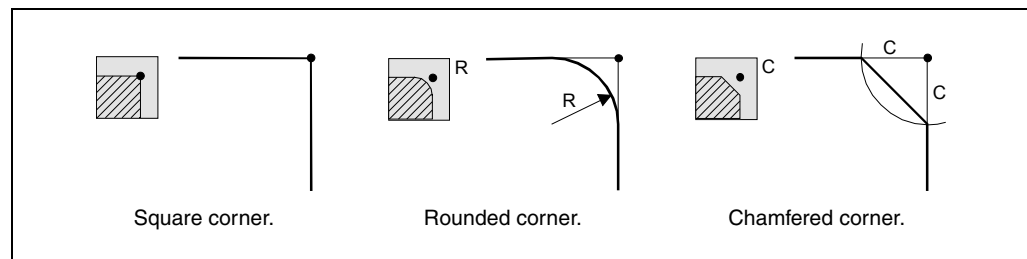
For a rounded corner, define the rounding radius (R); for a chamfer, define the distance from the theoretical corner to the chamfer point (C).

Safety distance.

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

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

The value of the safety distance on 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   Data definition (levels 3, 4 and 5)

Level 3:

    Icon for selecting the ZC or YZ plane.

    Icon to select the position of the starting point.

Z,C /   Coordinates of the starting point.
Z,Y:

L, H: Pocket dimensions.

α**:**   Inclination angle of the rectangular pocket.

W: Angular position of the spindle (in degrees) where the pocket will be milled in the YZ plane.

    Icon for selecting the vertex type in the pocket corners:

- Normal vertix
- Rounded vertix
- Chamfered vertix

r / c: Value of the corner rounding or chamfering radius in the rectangular pocket.

Dx: Safety distance on the longitudinal axis (cylindrical side).
Dz: Safety distance on the longitudinal axis (face).
X: Part plane.
P: Total rectangular pocket depth. When programmed with a 0 value, it generates the corresponding error.
I: Penetration step (pass) when roughing:

- If programmed with a positive value, the actual step (pass) will be the one closest to this value so all passes will be identical.
- If programmed with a negative value, the actual pass will be the one programmed and the last pass will be adjusted to the final (remaining) depth.
- If not programmed, it assumes 0.

Fx: Penetration feedrate for roughing and finishing. If not programmed, it assumes 0.

Level 4:

    Icon to select the ZC or YZ plane.

Zc, Cc /   Center coordinates of the circular pocket.
Zc, Yc:

Rc: Circular pocket radius.
W: Angular position of the spindle (in degrees) where the pocket will be milled in the YZ plane.
Dx: Safety distance on the longitudinal axis (cylindrical side).
Dz: Safety distance on the longitudinal axis (face).
R / X: • Cylinder radius when it is the ZC plane.

- X coordinate of the part surface when it is the ZY plane.

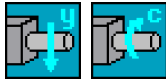**P:**   Total circular pocket depth. When programmed with a 0 value, it generates the corresponding error.

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

I: Penetration step (pass) when roughing:

- If programmed with a positive value, the actual step (pass) will be the one closest to this value and all the passes will be identical.

- If programmed with a negative value, the actual pass will be the one programmed and the last pass will be adjusted to the final (remaining) depth.

- If not programmed, it assumes 0.

Fx: Penetration feedrate for roughing and finishing. If not programmed, it assumes F/2.

Level 5:

Icon for selecting the machining plane (XC plane or XY plane).

Profile program : Number of the part-program where the pocket is defined.

Dz: Safety distance on the longitudinal axis.

Z: Z coordinate of the part surface.

P: Total profile pocket depth. If not programmed or programmed with a 0 value, the CNC will display the corresponding error message.

I: Maximum roughing pass. If not programmed or if programmed with a 0 value, it assumes the value of 75% of the diameter of the roughing tool.

Fz: Machining feedrate along the Z axis. If not programmed or programmed with a 0 value, the CNC assumes the roughing F value for the penetration in roughing and the finishing F for the penetration in finishing.

W: Angular position of the spindle (in degrees) where the profile will be machined in the XY plane (in degrees).

Spindle turning direction

3.5.3   Basic operation (levels 1 and 2)

The machining steps in this cycle are as follows:

1. If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.

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

3. The tool approaches the starting point (Xi, Zi) in rapid, keeping the selected safety distance according to axes X and Z.

4. Roughing operation, with successive facing passes, up to a distance from the selected final Z coordinate (Zf) 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.
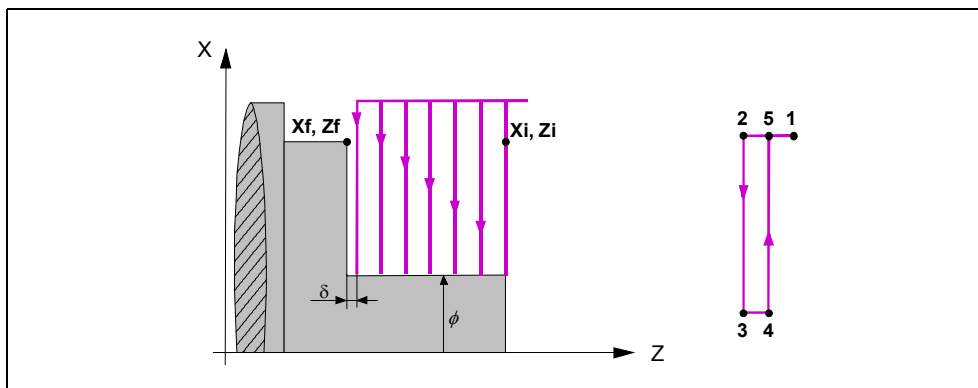
Each facing pass is done as shown in the figure, starting at point "1" and after going through points "3", "3" and "4", ending at point "5".

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).

FAGOR

FAGOR AUTOMATION

CNC 8055
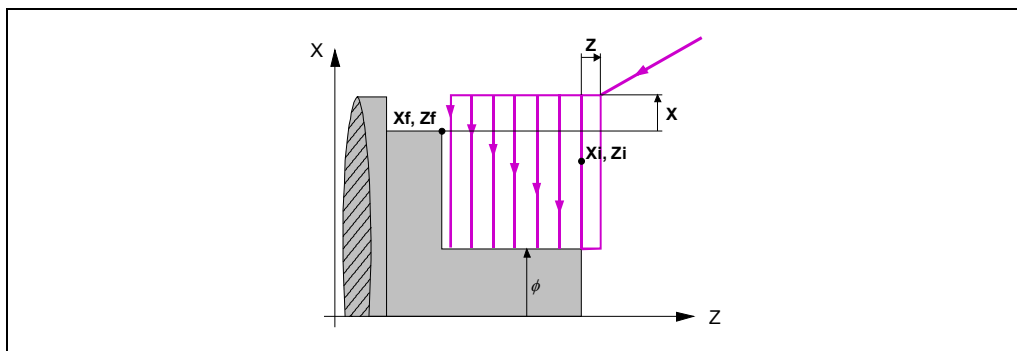**CNC 8055i**

·TC· OPTION
SOFT: V02.2X

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. This means that after approaching the finishing operation will be carried out.

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 diameter.

3.6　Taper turning cycle

This key accesses the taper turning cycles.

This cycle may be defined in two ways:
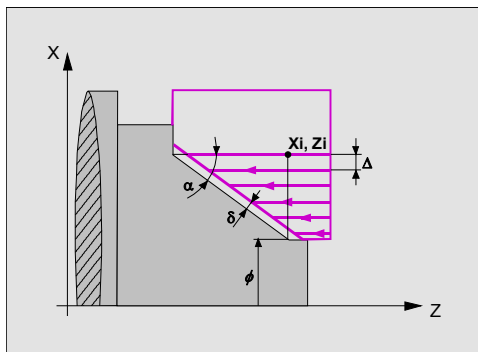
Level 1.

The following data must be defined:

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

Level 2.
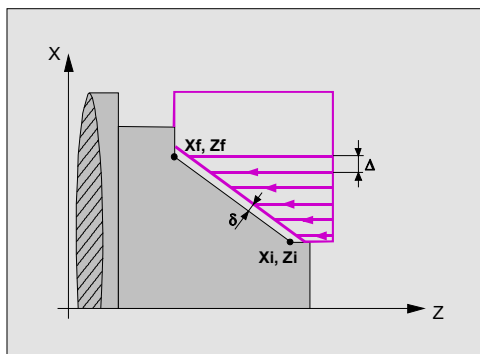
The following data must be defined:

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

Level 3.
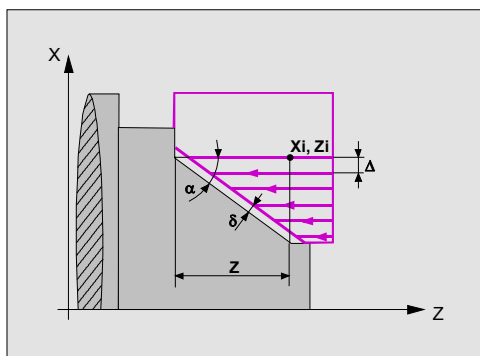
The following data must be defined:

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

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

3.6.1 Definition of data

Type of taper.

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

Coordinates of the theoretical corner, of the first point (Xi, Zi) and of the last point (Xf, Zf).

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 over this data, key in the desired value and press [ENTER].

Angle (α).

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

Chamfer (Z).

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

Safety distance.

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

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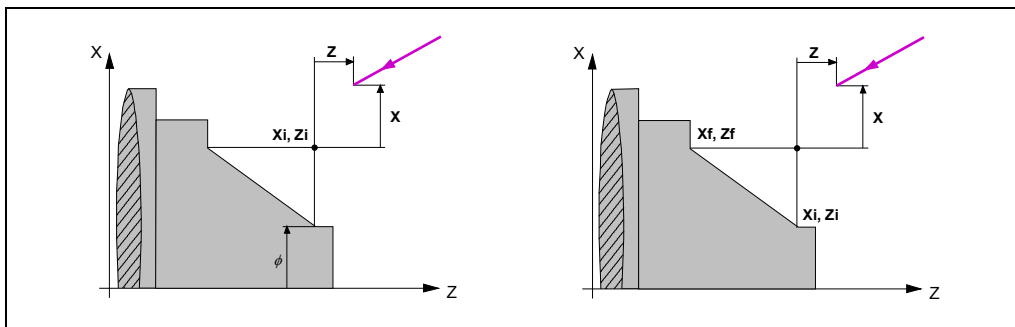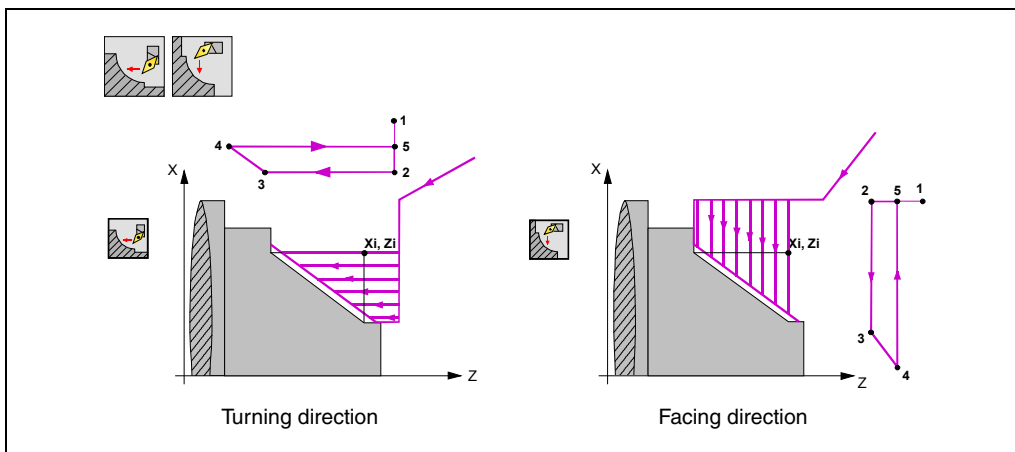

The value of the safety distance on X is always defined in radius.

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 two-color key.
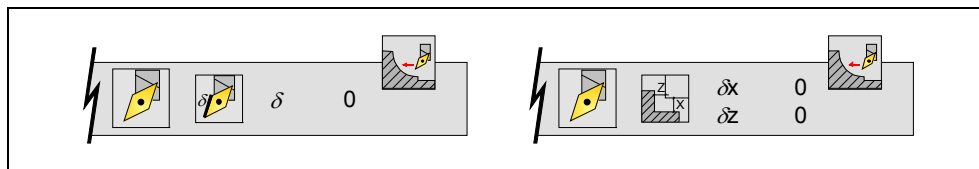

Turning direction   Facing direction

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

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 finishing stocks, place the cursor over this icon and press the two-color key. To define these residual stocks, place the cursor on the corresponding data, key in the desired value and press [ENTER].
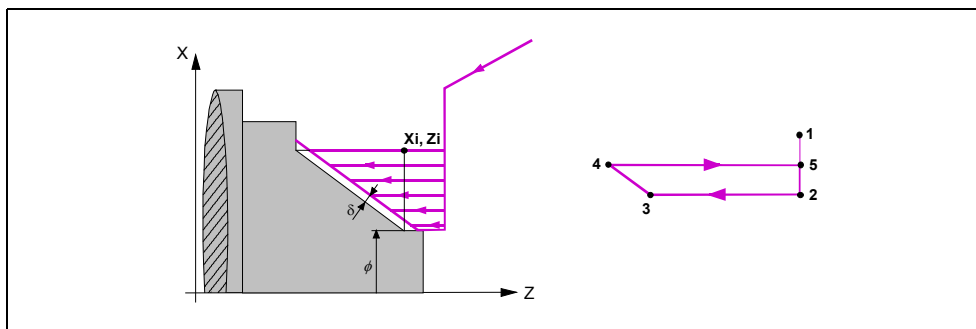
3.6.2 Basic operation

The machining steps in this cycle are as follows:

1. If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.

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

3. The tool approaches the theoretical corner in rapid, keeping the safety distance selected according to axes 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.
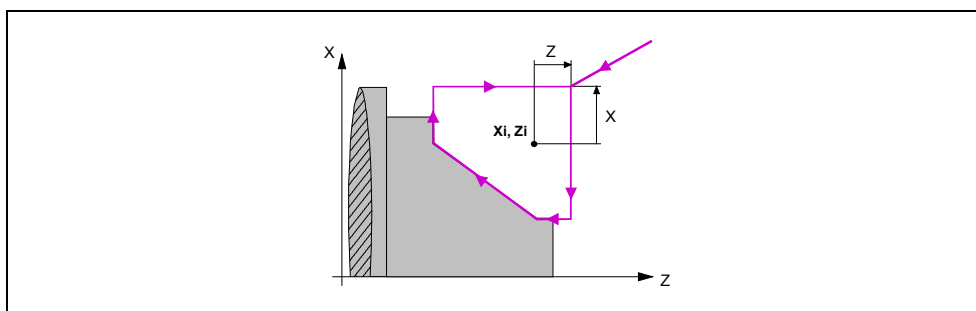

   Each machining pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".

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).

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. This means that after approaching the finishing operation will be carried out.

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.
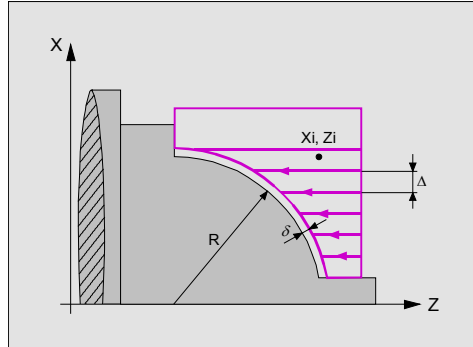
3.7   Rounding cycle

This key accesses the rounding cycles.

This cycle may be defined in two ways:

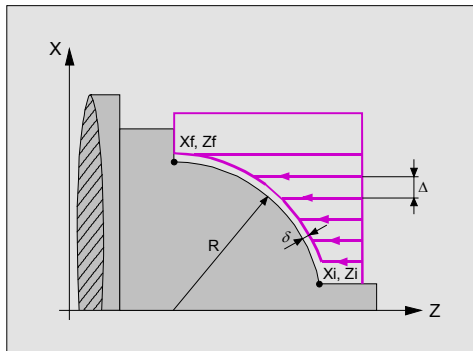*Level 1.*

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

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

3.7.1   Geometry definition

Type of rounding.

To select the type of rounding, place the cursor over this icon and press the two-color 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 two-color key.
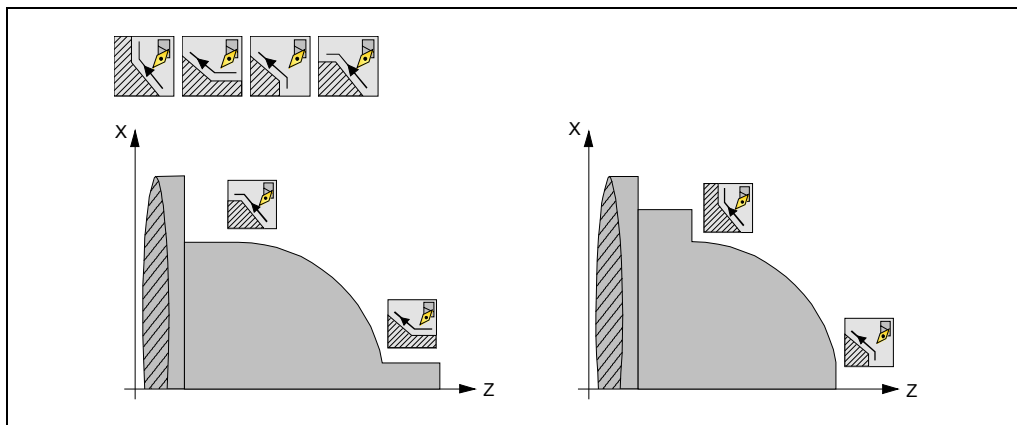
Convex rounding / concave rounding.

Whenever one of these is changed the CNC modifies the icon and displays the relevant geometry 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 two-color 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.
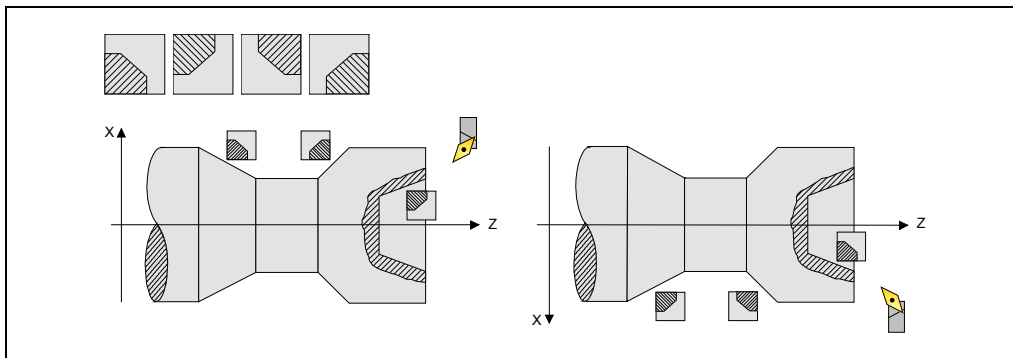
Coordinates of the theoretical corner or coordinates of the first point (Xi, Zi) and of the last point (Xf, Zf).

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].

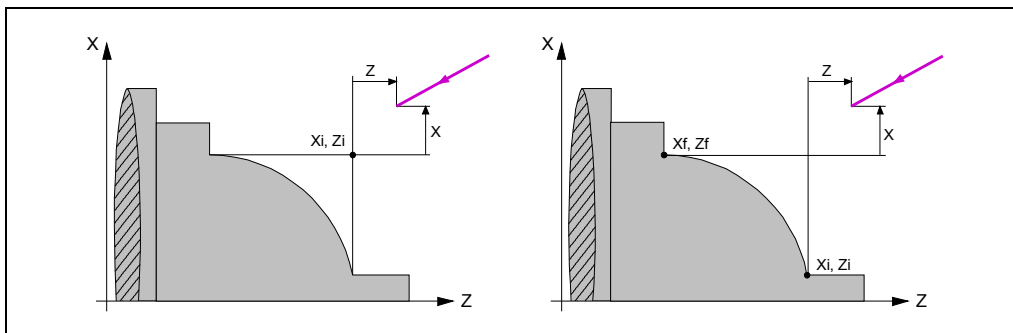 The top right window shows the tool position at all times.

Rounding radius (R).

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

Safety distance.

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

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

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

FAGOR

FAGOR AUTOMATION

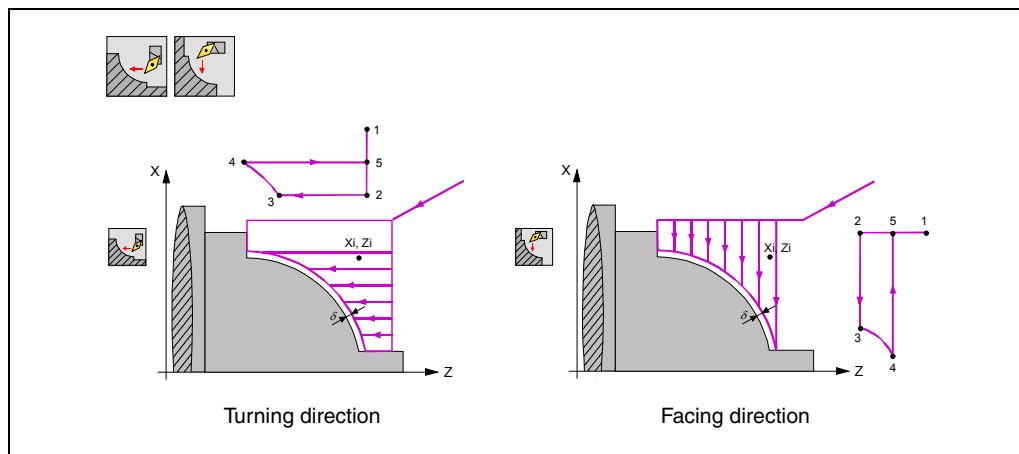**CNC 8055**
CNC 8055i

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 two-color key.
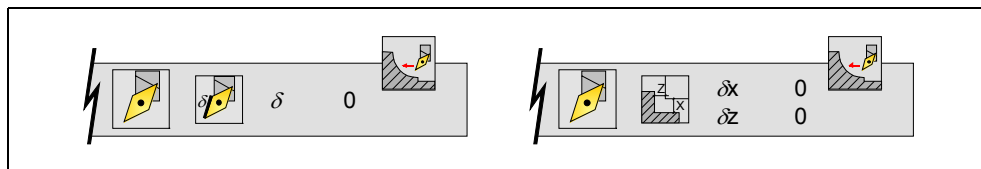

| Turning direction | Facing direction |

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 finishing stocks, place the cursor over this icon and press the two-color key. To define these residual stocks, place the cursor on the corresponding data, key in the desired value and press [ENTER].

FAGOR

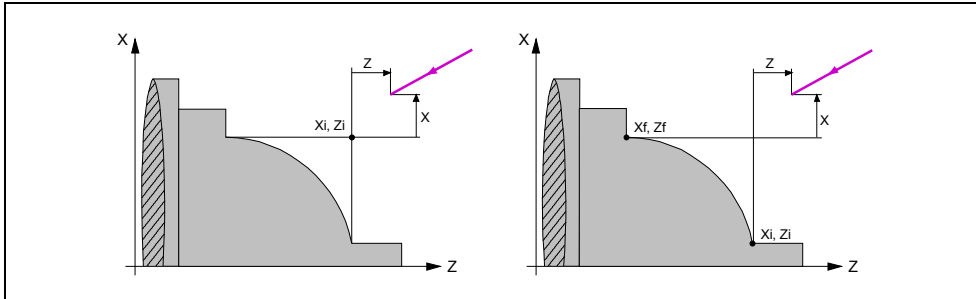FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2x
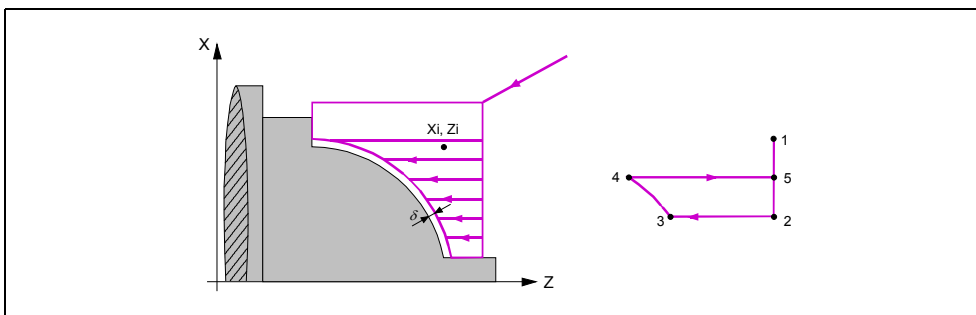
·112·

3.7.2 Basic operation

The machining steps in this cycle are as follows:

1. If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.

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

3. The tool approaches the theoretical corner in rapid, keeping the safety distance selected according to axes 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.
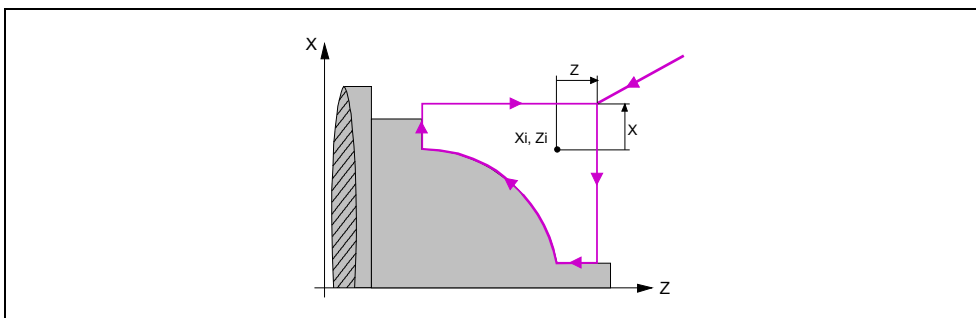

    Each machining pass is done as shown in the figure, starting at point "1" and after going through points "2", "3" and "4", ending at point "5".

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).

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. This means that after approaching the finishing operation will be carried out.

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.
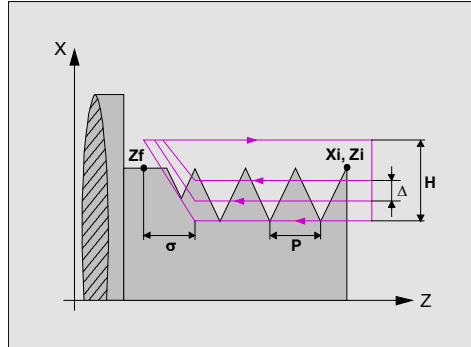
3.8   Threading cycle

This key accesses the threading cycles.

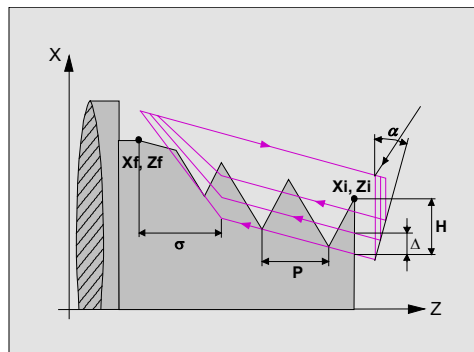This cycle may be defined in several ways:

Level 1. Longitudinal threading.

The following data must be defined:
- Coordinates of the starting point.
- Z coordinate of the last point.
- The thread pitch.
- Thread type (variable pitch or fixed 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 threading.

The following data must be defined:
- Coordinates of the starting point.
- Coordinates of the last point.
- The thread pitch.
- Thread type (variable pitch or fixed pitch).
- The distance to the end of the thread.
- The total depth.
- The spindle angular position.
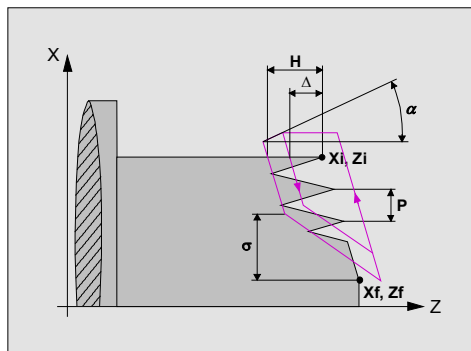- The distance to the end of the thread.

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·**TC**· OPTION
SOFT: V02.2X

Level 3. Face threading.

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.

Level 4. Thread repair.

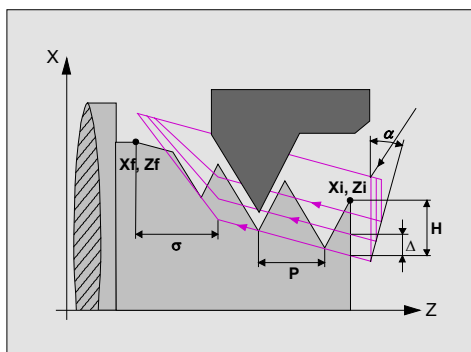Available when setting spindle machine parameter "M19TYPE (P43)=1".

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.
- Z coordinate of the valley (root).
- The spindle angular position at the valley (root).
- The distance to the end of the thread.

Level 5. Threads with several entries.

Available when setting spindle machine parameter "M19TYPE (P43)=1".
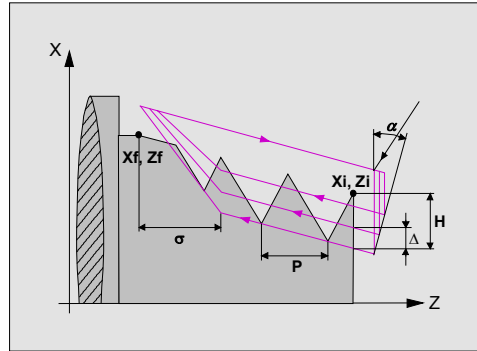
The following data must be defined:

- Coordinates of the starting point.
- Coordinates of the last point.
- The thread pitch.
- Thread type (variable pitch or fixed pitch).
- The distance to the end of the thread.
- The total depth.
- Z coordinate of the valley (root).
- The spindle angular position at the valley (root).
- The number of thread entries.
- The distance to the end of the thread.

3.8.1 Geometry definition

Type of tapping.

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

Inside threading.

External Threading.

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

Coordinates of the first point (Xi, Zi) and of the last point (Xf, Zf).

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.

Instead of defining the end point of the thread, the thread can also be programmed by defining the angle and the Z increment. This option is valid for all threads and thread repairs, except for face threads. To enable this option, set the general machine parameter COCYF6 (P153)=1.

Standard threads.

Up to 6 types of standard threads may be selected. See *"3.8.2 Standard threads"* on page 124.

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

Number of entries (N).

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 set along the taper of the thread or along its associated axis. In either case, parameter "P" is be used, but with a different sign.
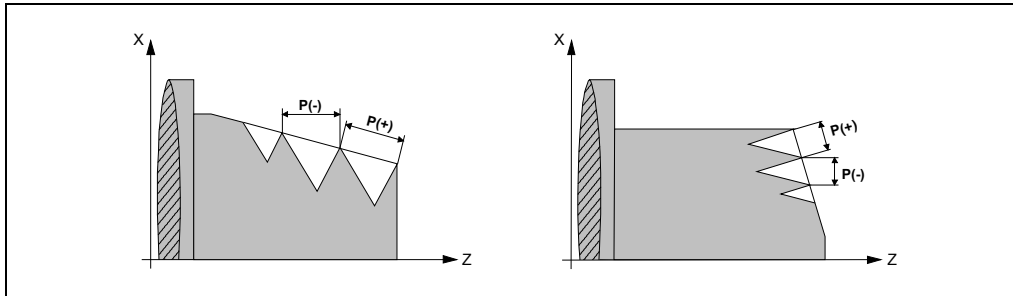
- "P" with a positive sign to program the pitch is programmed along the inclination of the thread.
- "P" with a negative sign to program the pitch along the associated axis.

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

Thread type

Icon to select the type of thread to make (with constant pitch or with variable pitch)

### Increment/decrement of thread pitch (△P)

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

> *It must be borne in mind that is 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. In this withdrawal movement threading is continued. Place the cursor over this data, key in the desired value and press [ENTER].
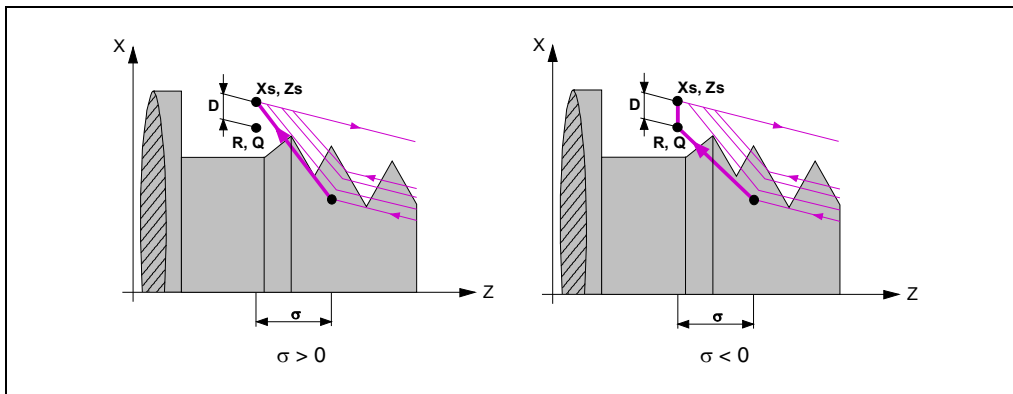
- If programmed with a positive value, the tool exits the thread without going through the final point (Xf, Zf).
- If programmed with a negative value, the tool exits the thread going through the final point (Xf, Zf).
- If not programmed, a value of 0 is assumed (blind thread).

  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.

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

To change one of these values, place the cursor on the corresponding data, key in the desired value and press [ENTER].
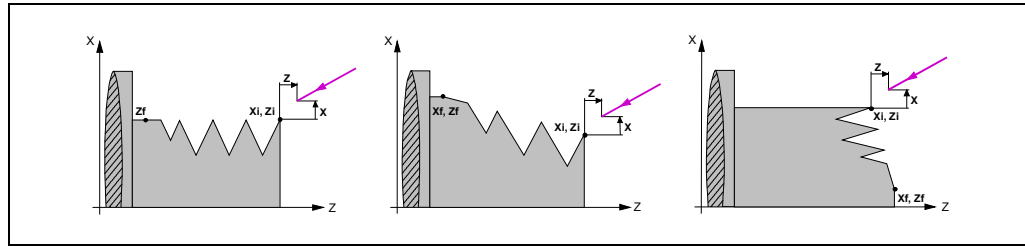
The value of the safety distance on X is always defined in radius.

Spindle angular position.

In the thread repair cycle, this parameter indicates the angular position of the spindle at the root (valley) and must be programmed together with the "Z coordinate of the valley (root)" as described later on.

On the rest of the levels, programming this data is optional and it indicates the angular position of the spindle or angle with respect to the reference mark where the threading must begin. It permits making multiple-entry threads without bringing the starting point back.

To select whether the entry angle must be defined or not, place the cursor over this icon and press the two-color key.

Threading without programming the entry angle.

W 0.0000 Threading programming the entry angle.

Depth of the successive threading passes (∆).

The ∆ 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 two-color key.

The depth of each pass will depend on the number of the corresponding pass. The penetrations are ∆, ∆√2, ∆√3, ∆√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 ∆.
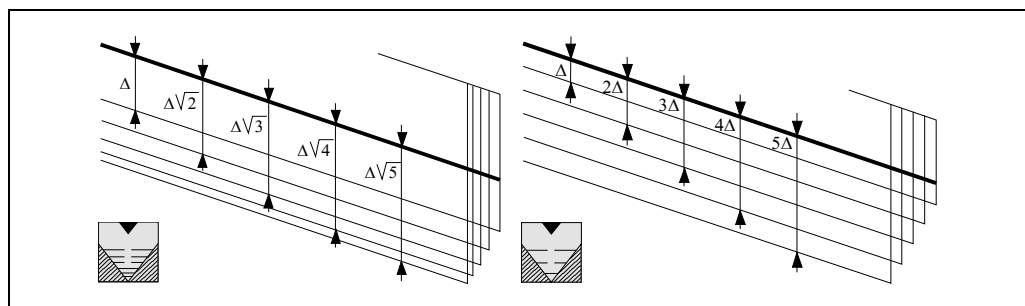
For the level 1 threading cycle, the depth of each pass depends on the corresponding pass number (∆, ∆√2, ∆√3, ∆√4, etc.).

Type of tool penetration.

To select the type of tool penetration, place the cursor over the icon and press the two-color 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.

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.

Repairing part of a thread

Within the thread repair cycle, it is possible to do a partial thread repair.

This icon defines the partial thread repair. The rest of the data to be entered are the same as for the standard thread repair. The entry to the thread (start) is calculated automatically.

Enter the starting and end point of the segment to be machined in their corresponding boxes. Partial thread repair works for both outside and inside threads.

To enable this option, set bit 11 of g.m.p. CODISET (P147) =1.

Multi-start (entry) thread repair

Within the thread repair cycle, it is possible to do a multi-entry thread repair.

To program a multi-start thread repair operation, indicate the number of thread starts (entries) to be repaired in the new field [N]. The CNC will calculate and run all the passes to complete the repair of all the threads.

To enable this option, set bit 12 of general parameter CODISET (P147) = 1.

Face thread repair cycle

Within the thread repair cycle, it is possible to do a multi-entry face thread repair.

To repair face threads, use the front thread definition icon.

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

·121·

Penetration increment (step) in thread repair

In any type of thread repair, the "h" parameter increases the depth of the repair. Only positive "h" values are admitted.

This increase in the thread repair depth is enabled with bit 14 of general machine parameter COCYF6 (P153) =1, which is disabled by default.

Symmetrical tools:

For symmetrical tools, if parameter "h" is programmed, it will enlarge the thread while keeping the entry flank. The tool data CUTA and NOSEA must be entered correctly as well as the entry (start) angle α.
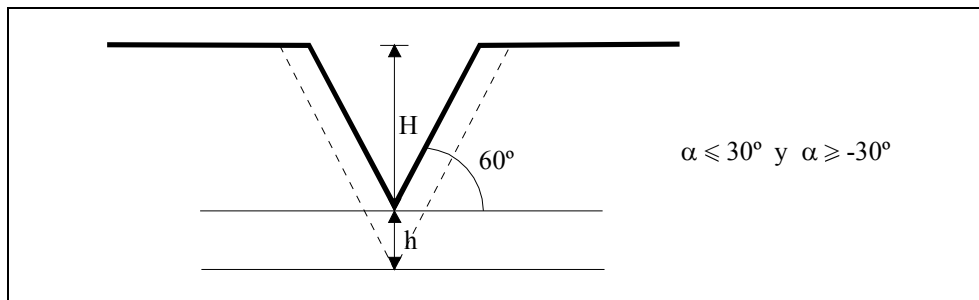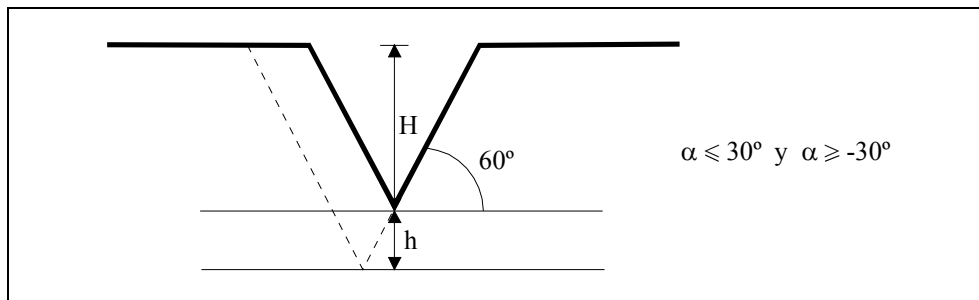
CUTA=90:

$\alpha \leqslant 30^\circ$ y $\alpha \geqslant -30^\circ$

CUTA=60:

$\alpha \leqslant 30^\circ$ y $\alpha \geqslant -30^\circ$

Asymmetrical cutters with two tips:

In the case of asymmetrical tools, the entry angle must always be respected.
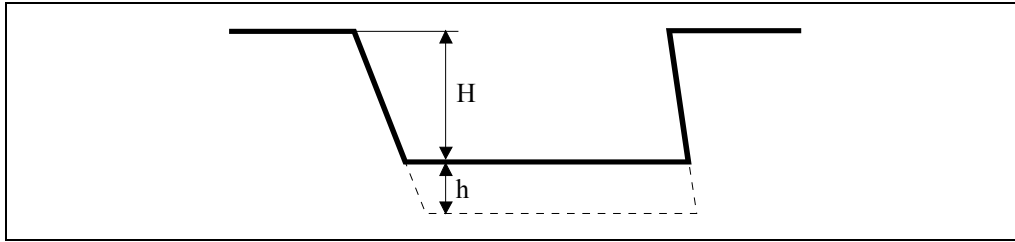
The tool data CUTA and NOSEA must be entered correctly according to the calibrated tip.

10º   *3º*   CUTA=100   NOSEA=80   Punta calibrada $\alpha \leqslant -3º \ y \ \alpha \geqslant -10º$

10º   *3º*   CUTA=93   NOSEA=87   Punta calibrada $\alpha \leqslant -3º \ y \ \alpha \geqslant -10º$
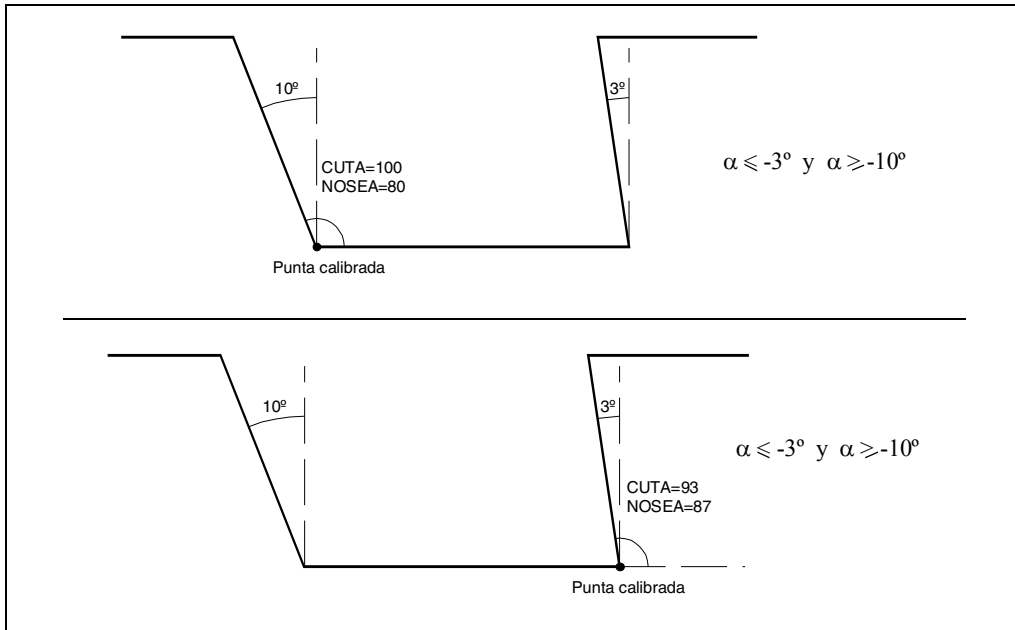
Reversal of the starting and final point of the face thread repair

This feature is only used on face threads; i.e. in threading level 3 and in face thread repair level 4.

Set bit 13 of general parameter COCYF6 (P153) = 1 to enable this option.

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)	Normal pitch Whitworth thread.
B.S.F.	Fine pitch Whitworth thread.
U.N.C.	Normal pitch unified American thread.
U.N.F.	Fine pitch American thread.
API	Special threads for pipes of the oil sector.

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

Diameter		Step		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

3.

WORKING WITH OPERATIONS OR CYCLES
Threading cycle

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

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

Diameter		Step		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

Normal pitch Whitworth thread: B.S.W. (W)

Thread			Step			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:

Pitch in mm = 25.4 / number of edges

Pitch in inches = 1 / number of edges

Depth in internal threads = 0.6403 x Pitch

Depth in external threads = 0.6403 x Pitch

3.

WORKING WITH OPERATIONS OR CYCLES
Threading cycle

FAGOR ∕

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

Fine pitch Whitworth thread: B.S.F

Thread			Step			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:

Pitch in mm = 25.4 / number of edges

Pitch in inches = 1 / number of edges

Depth in internal threads = 0.6403 x Pitch

Depth in external threads = 0.6403 x Pitch

Normal pitch unified American thread: UNC (NC,USS)

Thread			Step			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 millimeters or inches. For example, to define an American thread with a 1/4 pitch, enter 6.3500 mm or 0.2500 inches.

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

Pitch in mm = 25.4 / number of edges

Pitch in inches = 1 / number of edges

Depth in internal threads = 0.5413 x Pitch

Depth in external threads = 0.6134 x Pitch

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

Fine pitch American thread: UNF (NF,SAE)

Thread			Step			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:

Pitch in mm = 25.4 / number of edges

Pitch in inches = 1 / number of edges

Depth in internal threads = 0.5413 x Pitch

Depth in external threads = 0.6134 x Pitch

API compliant thread

The window for standard threads includes the API standard thread; a special standard for pipes of the oil sector. Once this standard is selected, a new window will appear for configuring the thread according to that standard.

These standards are not applicable to face threading, for level 4, selecting face threading disables all standards.

The following table shows the values for the API standard:

		TPI (Thread per inch)	TPF (taper per foot)	Depth (H)	Pitch (-P negative)
API 5	CR 08	8	¾	1.810	-3.175
	CR 10	10	¾	1.412	-2.54
	CB 05	5	¾	1.575	-5.08
API 7	V38 4-2	4	2	3.095	-6.35
	V38 4-3	4	3	3.083	-6.35
	V40 5-3	5	3	2.993	-5.08
	V50 4-2	4	2	3.741	-6.35
	V50 4-3	4	3	3.754	-6.35

3.8.3 Basic operation. Longitudinal threading

The machining steps in this cycle are as follows:

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 with the speed selected and in the direction stated. Depending on the spindle turning direction, the thread will be clockwise or counterclockwise.

3. The tool approaches the theoretical corner in rapid, keeping the safety distance selected according to axes X and Z.
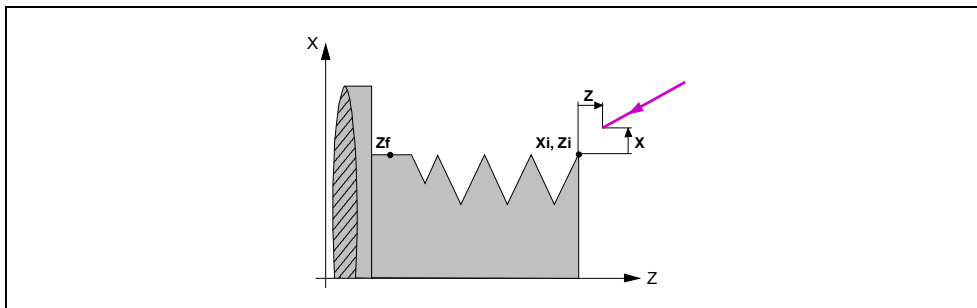
4. The threading is done with radial penetration and in successive passes until the total depth is reached. The depth of each pass depends on the corresponding pass number Δ, $\Delta\sqrt{2}$, $\Delta\sqrt{3}$, $\Delta\sqrt{4}$, etc.
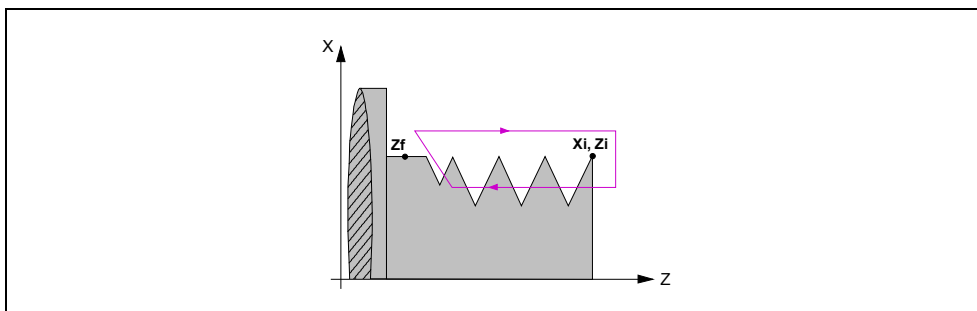
Each threading step is carried out as follows:

– 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.

– Fast backwards 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).
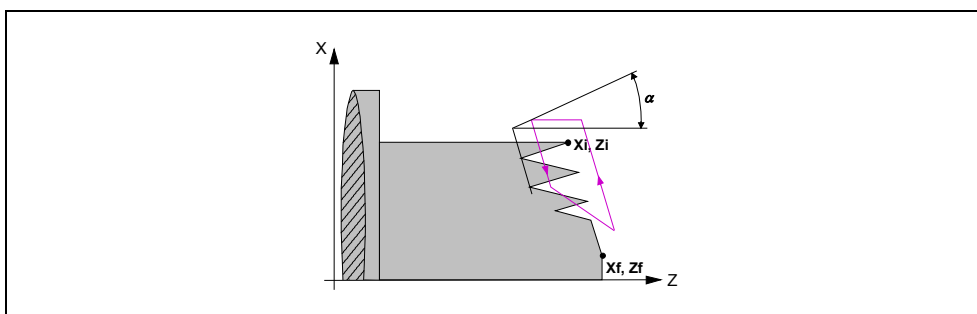
3.8.4 Basic operation. Taper threading

The machining steps in this cycle are as follows:

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 with the speed selected and in the direction stated. Depending on the spindle turning direction, the thread will be clockwise or counterclockwise.

3. The tool approaches the theoretical corner in rapid, keeping the safety distance selected according to axes 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.

   − Fast backwards 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).

3.8.5 Basic operation. Face threading

The machining steps in this cycle are as follows:

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 with the speed selected and in the direction stated. Depending on the spindle turning direction, the thread will be clockwise or counterclockwise.

3. The tool approaches the theoretical corner in rapid, keeping the safety distance selected according to axes 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.

 – Fast backwards 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).

3.8.6   Basic operation. Thread repair

Cycle definition

Define the dimensions of the thread like at the rest of the levels and the coordinates of one of the roots.

> To define the coordinates of the root, the CNC must know the position of the spindle. The spindle must be oriented in order for the CNC to know the spindle position. On CNC power-up, it's enough to do this once.

With the spindle stopped, take the tool to be used for thread repair to one of the roots of the thread. Once there, take these 2 values:

- Z coordinate of the valley (root).

  Place the focus on the data and press [RECALL] [ENTER].

- Spindle angular position at the valley (root).

  Place the focus on the data and press [RECALL] [ENTER].

The CNC assumes these 2 data needed for thread repair.

Machining steps

The machining steps for this cycle are identical to those for taper threading described earlier. The CNC will make a new thread over the existing one, maintaining the roots and inclinations of the current thread.
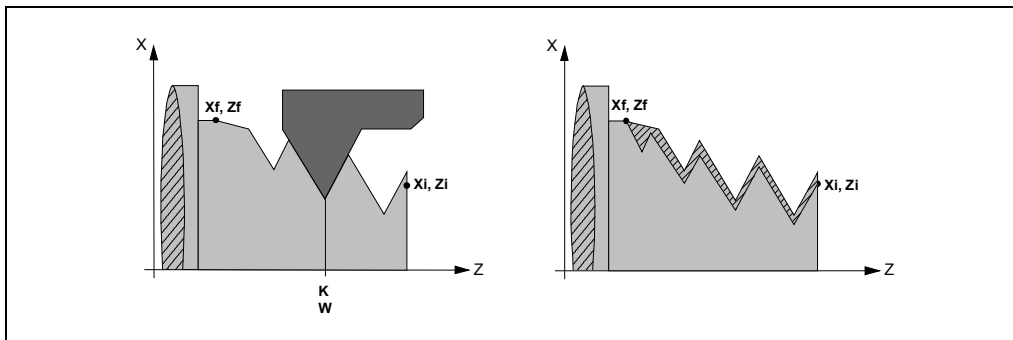
To do a thread repair, proceed as follows:

1. The spindle must have been oriented (M19) at least once since the CNC was turned on.

2. Take the Z coordinate values (teach-in) and the angular position of the spindle at the root of the thread, parameters K W, while the tool is positioned at one of the roots of the thread to be repaired.

3. Define the cycle for thread repair.

4. Execute (run) the cycle.

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.
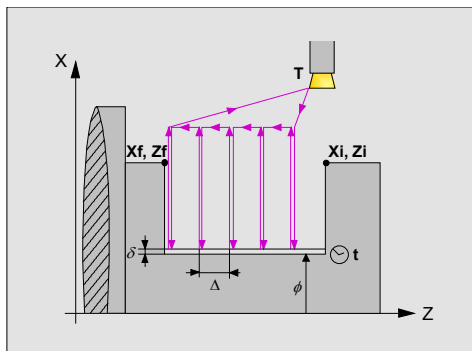
The following data must be defined:

- 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.
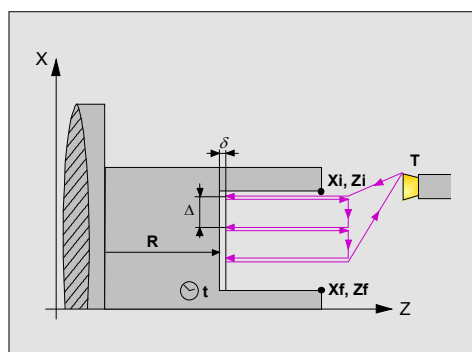
The following data must be defined:

- 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.
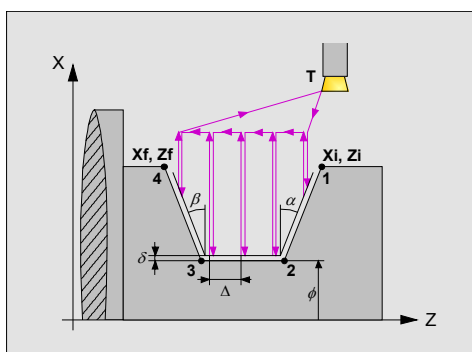
The following data must be defined:

- 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.
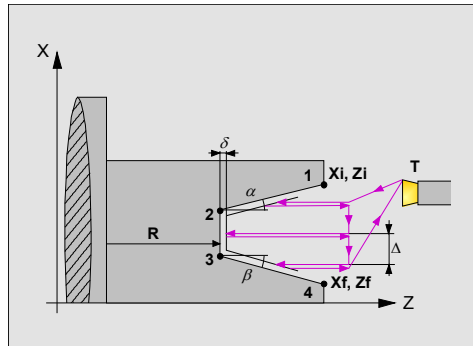
The following data must be defined:

- 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.
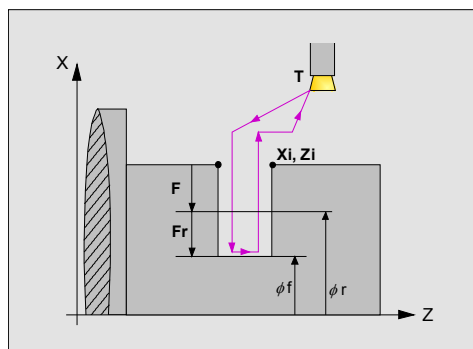
The following data must be defined:

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

3.

WORKING WITH OPERATIONS OR CYCLES
Grooving cycle

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

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. The left-hand corner of the cutter is calibrated.
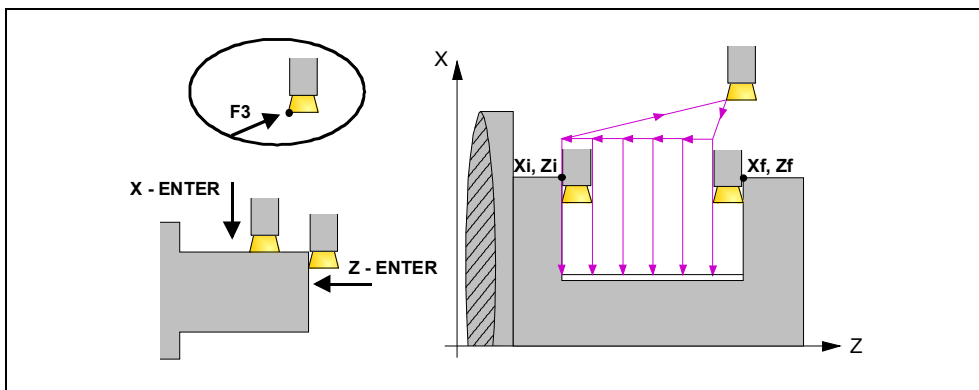
• Location code F1. The right-hand corner of the cutter is calibrated.
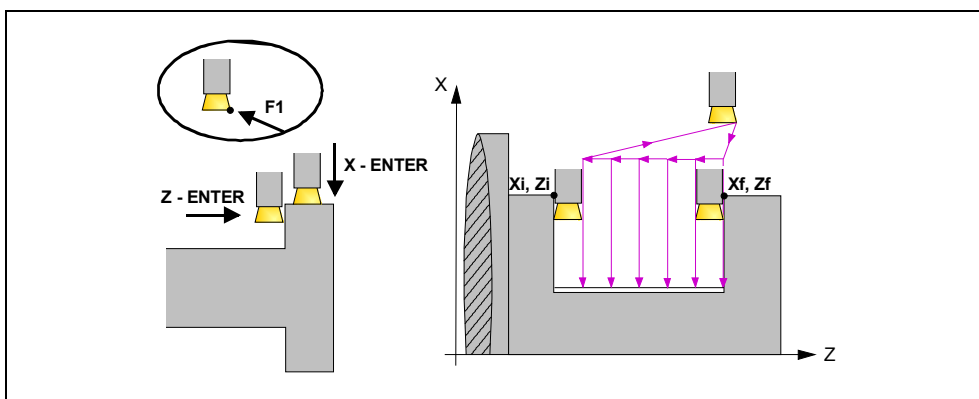
• Location code F2. The CNC only calibrates along the X axis and it assumes the center of the cutter as the calibrated point.
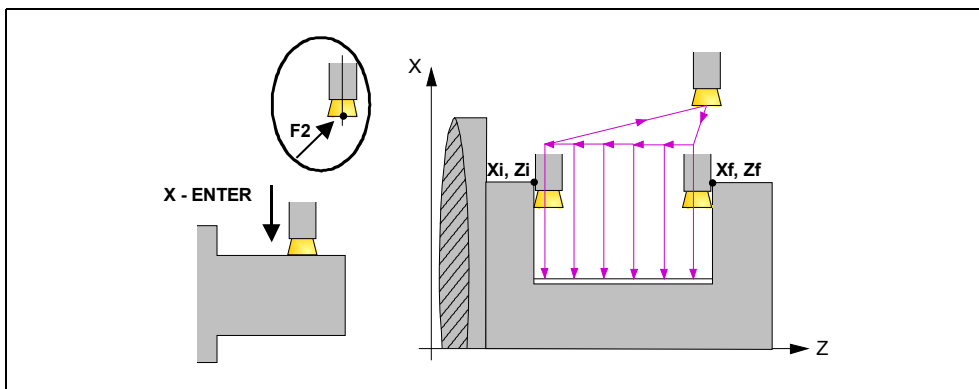
3.9.2 Geometry definition

Type of grooving.

To select the type of grooving, place the cursor over this icon and press the two-color 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 (Xi, Zi) and of the last point (Xf, Zf).

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 (Φr) and final diameter (Φf).

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].
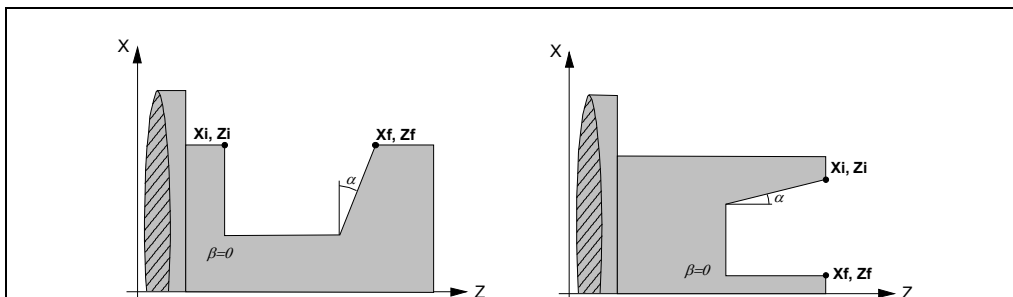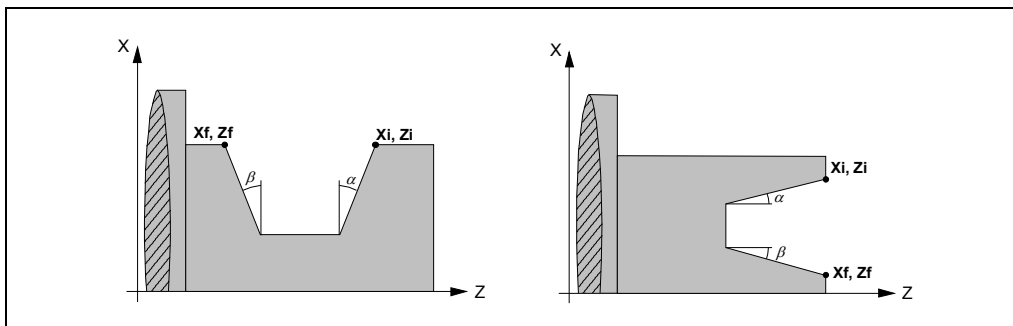
The next example shows grooves with $\alpha=20^\circ$ and $\beta=0^\circ$.

Type of machining to be carried out on each corner.

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

Square corner.

R   Rounded corner.

C   Chamfered corner.

These data must be defined when grooving with incline walls. The type of machining must be defined for all four corners of the groove.
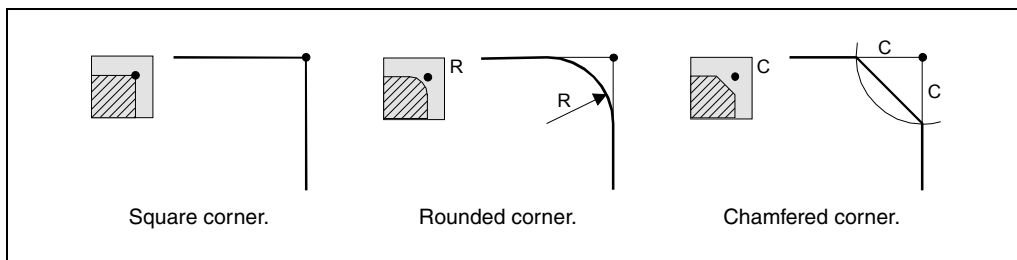
| Square corner. | Rounded corner. | Chamfered corner. |

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.

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

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

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

Type of machining for the roughing pass.

To select the type of grooving, place the cursor over the desired icon and press the two-color 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 a 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.
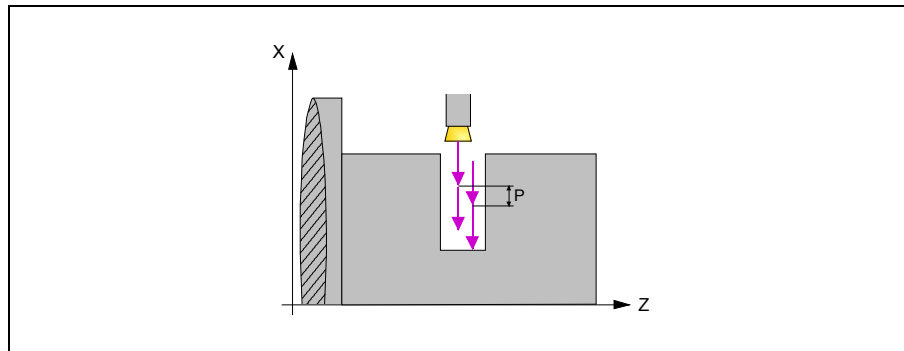
In the first pass, it machines in zig-zag with penetration according to parameter P.
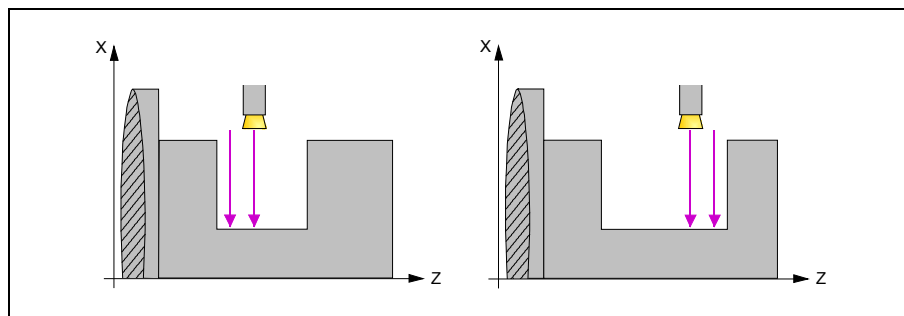
It runs the rest of the passes machining down to the bottom. In order to take care of the tool, first machine the center area towards the origin with one side of the cutter (left figure) and then from the center to the opposite side with the other side of the cutter (right figure).

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

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

Type of machining for the finishing pass.

This data must be defined when grooving with incline walls. To select the type of machining, place the cursor over this icon and press the two-color key.
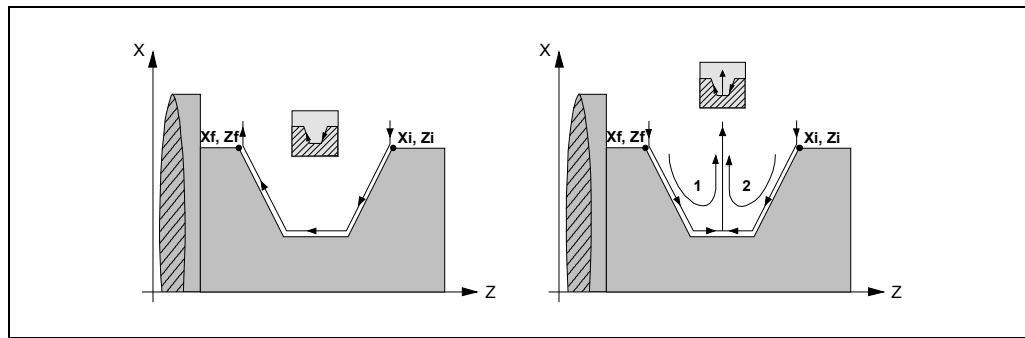
Groove repetition.

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 initial groove is tapered, Xi different from Xf, that taper is maintained for the rest of the grooves.
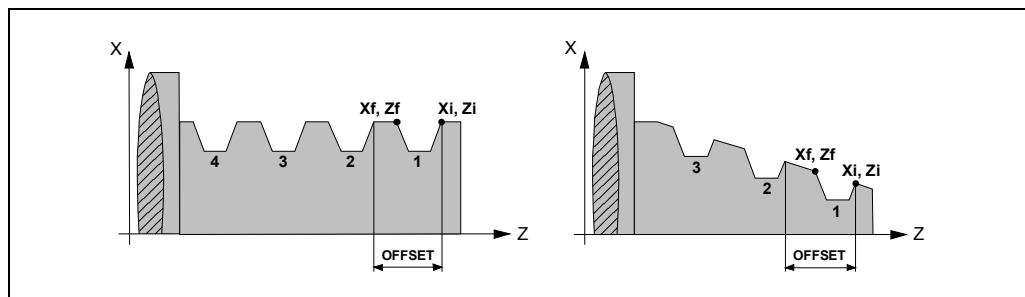
If the number of grooves is defined with a value of 0 or 1, it only makes one groove.

Intermediate diameter ($Φr$) and feedrate (Fr).

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

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

·142·

3.9.3 Basic operation. Grooving

The machining steps in this cycle are as follows:

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 with the speed selected and in the direction stated.

3. The tool approaches the starting point (Xi, Zi) in rapid, keeping the selected safety distance according to axes X and Z.
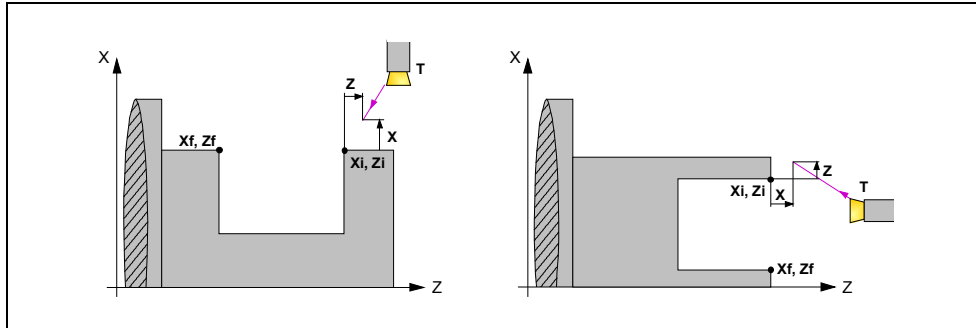
4. Roughing operation, with successive grooving passes, up to a distance from the final depth equal to the finishing stock.

   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. This means that after approaching the finishing operation will be carried out.

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.

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

·144·

3.9.4   Basic operation. Cut off

The machining steps in this cycle are as follows:

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 with the speed selected and in the direction stated.

3. The tool approaches the starting point (Xi, Zi) in rapid, keeping the selected safety distance according to axes 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).

3.10   Drilling and tapping cycles

This key accesses the drilling and tapping cycles.

There may be up to 5 cycles available depending on the type of machine and how the CNC machine parameters have been set:

- Drilling cycle.
- Multiple drilling cycle.
- Tapping cycle.
- Multiple threading cycle.
- Multiple slot milling cycle.

The drilling and tapping cycles are always available. The multiple drilling, multiple threading and multiple slot milling cycles are available when the machine has a live tool and spindle orientation.

Level 1. Drilling cycle.
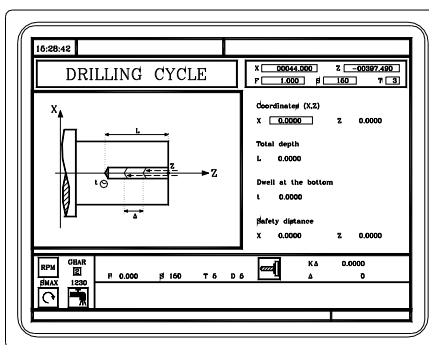
The following data must be defined:

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

Level 2. Tapping cycle.
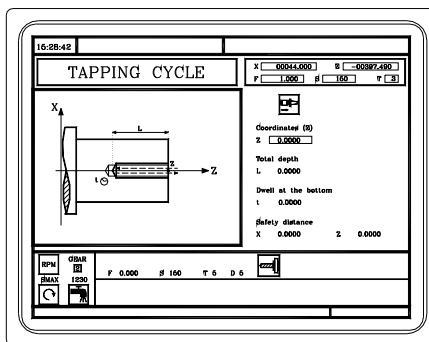
The following data must be defined:

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

Level 3. Multiple drilling cycle.
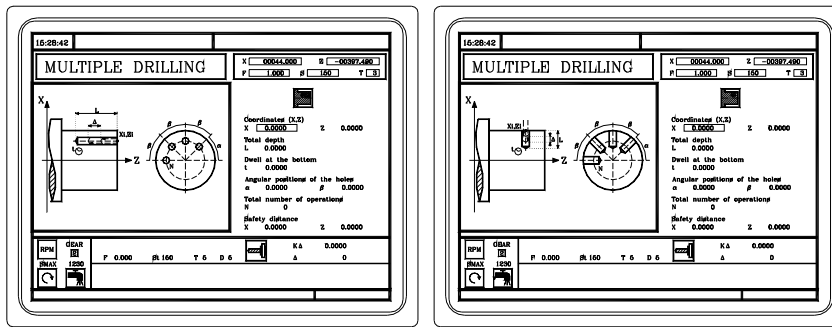
Multiple drilling can be done on the cylindrical side of the part or on its face. The following data must be defined:

- The coordinates of the starting point.
- The total depth.
- The dwell at the bottom.
- The angular position of the holes.
- The total number of holes.

Level 4. Multiple threading cycle.
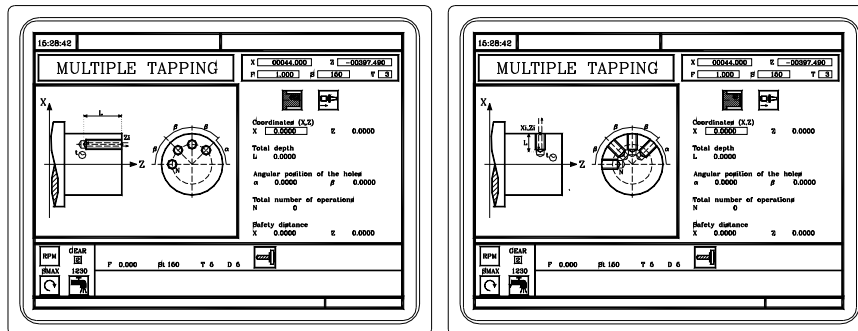
Multiple threading can be done on the cylindrical side of the part or on its face. The following data must be defined:

- The coordinates of the starting point.
- The total depth.
- The angular position of the threads.
- The total number of threads.

Level 5. Multiple slot milling cycle.
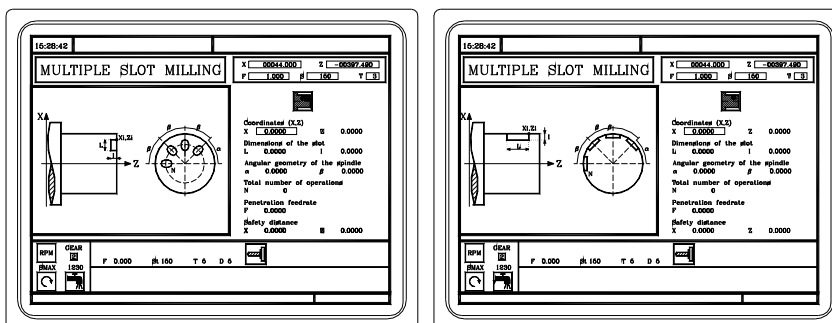
Multiple slots may be milled out on the side of the part or on its face. The following data must be defined:

- The coordinates of the starting point.
- The dimensions of the slot.
- The angular position of the slots.
- The total number of slots.

3.10.1   Geometry definition

Machining on the face or side of the part.

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

Machining on the face of the part.

Machining on the side of the part.

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

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.

Tapping must always be axial, at the rotation center (X0). For drilling, although usually done at the rotation center, it is possible to define X with a value other than X0.

Total depth (L).

Place the cursor over 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).

Define the waiting time in seconds, after each deepening, until the withdrawal movement starts. To define it, place the cursor on this data, key in the desired value and press [ENTER].

Angular position of the machining operations (α, β).

The α data indicates the angular position of the first machining operation and the β data the angular step between machining operations. To define them, place the cursor on the corresponding data, key in the desired value and press [ENTER].
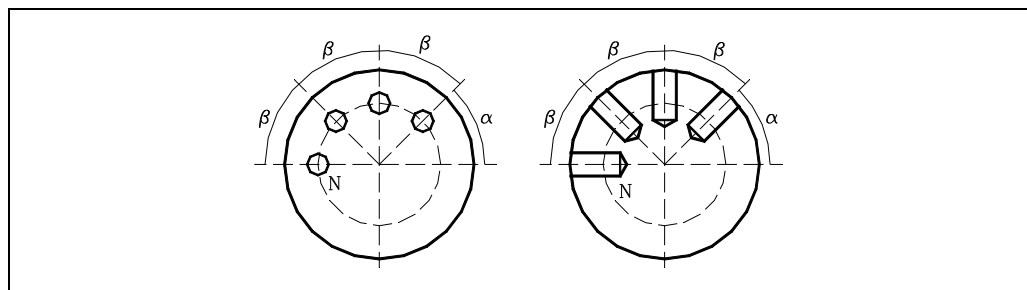
Number of operations (N).

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

Dimensions of the slot (L, I).

"L" indicates the length of the slot and "I" its depth. To define them, place the cursor on the corresponding data, key in the desired value and press [ENTER].
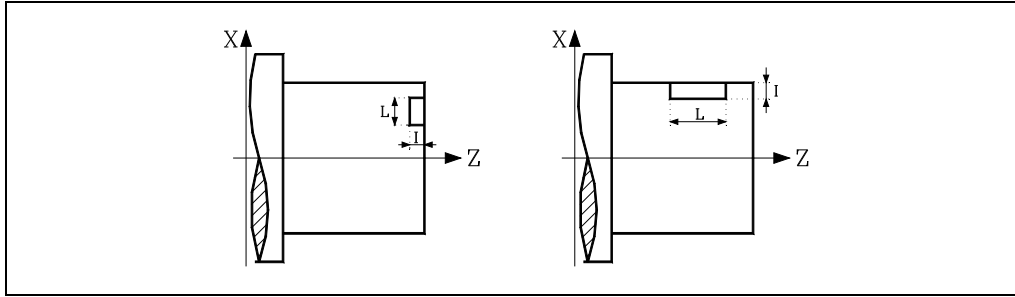
Penetrating feedrate (F).

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

Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the 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].
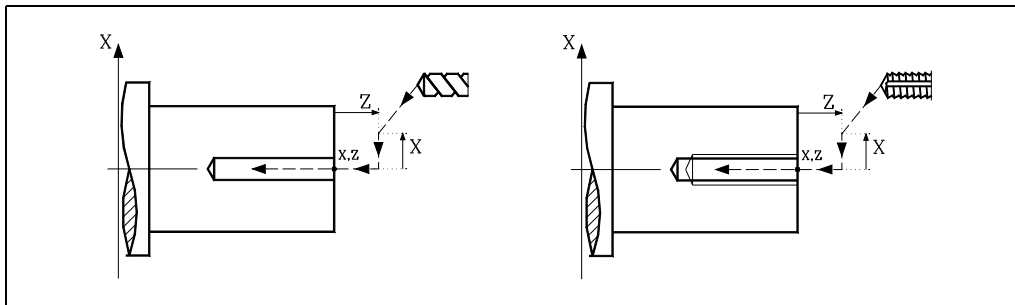
The value of the safety distance on X is always defined in radius.

3.10.2 Drilling cycle. Basic operation

The machining steps in this cycle are as follows:

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 with the speed selected and in the direction stated.

3. The tool approaches the starting point (Xi, Zi) in rapid, keeping the selected safety distance according to axes X and Z.

4. Drilling loop. The steps are repeated, penetrating each time the amount indicated by K∆ and ∆, up to the depth L. ∆ defines the drilling peck and K∆ the reduction factor of that peck.

   – The first drilling peck will be ∆, the second one "K∆ ∆", The third one "K∆ (K∆ ∆)" and so on; in other words, from the second peck on, the new peck will be the product of the K∆ factor times the previous peck.

   – After every drill pass, the tool moves back an "H" distance. Then it quickly approaches 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.

   – Fast backwards to the approach point.

5. Dwell t at the bottom of the hole.

6. Fast backwards 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).

3.10.3   Tapping cycle. Basic operation

The machining steps in this cycle are as follows:

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 with the speed selected and in the direction stated.

3. The tool approaches the starting point (Xi, Zi) in rapid, keeping the selected safety distance according to axes X and Z.
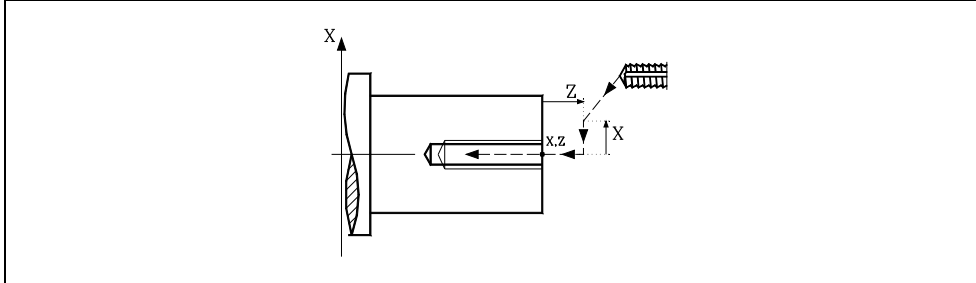
4. Tapping of the part in work feedrate F, until the depth L is reached.

Tapping (with a clutch).

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

5. The spindle reverses 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.
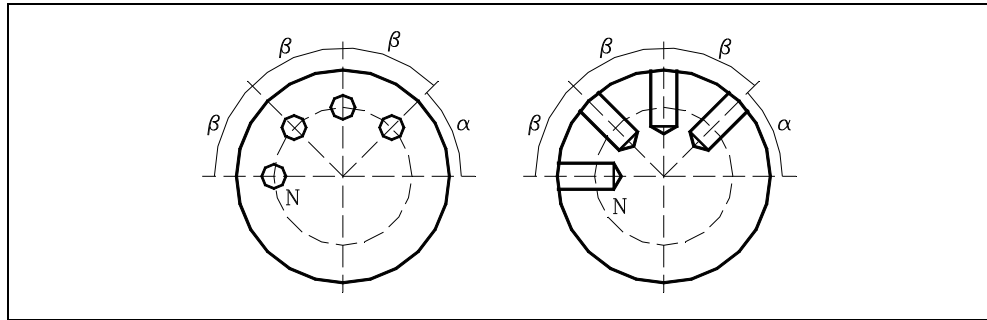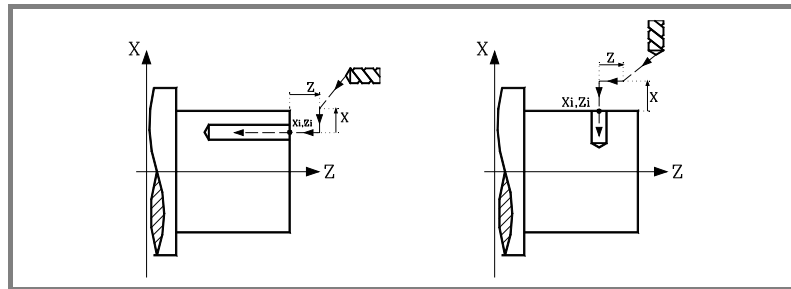
3.10.4   Multiple drilling cycle. Basic operation

The machining steps in this cycle are as follows:

1. If the spindle is working in open loop (RPM or CSS mode) the CNC stops the spindle and performs a home search on the spindle (Io).

2. If the operation 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.

3. The CNC starts the live tool at the selected speed and in the indicated direction.

4. Orients the spindle at the angular position for the first hole (indicated by α).

5. The tool approaches in rapid up to the starting point, keeping the safety distance selected according to axes X and Z.

6. Drilling loop. The steps are repeated, penetrating each time the amount indicated by KΔ and Δ, up to the depth L. Δ defines the drilling peck and KΔ the reduction factor of that peck.

   − The first drilling peck will be Δ, the second one "KΔ Δ", The third one "KΔ (KΔ Δ)" and so on; in other words, from the second peck on, the new peck will be the product of the KΔ factor times the previous peck.

   − After every drill pass, the tool moves back an "H" distance. Then it quickly approaches 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.

   − Fast backwards to the approach point.

7. Dwell t at the bottom of the hole.

8. Fast backwards to the approach point.

9. Depending on the value assigned to parameter N (number of holes), the CNC repeats the following steps N times.

   − The spindle moves to the next drilling point (angular increment β).

   − Repeats the drilling movements indicated in steps 6, 7 and 8.

10. 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.

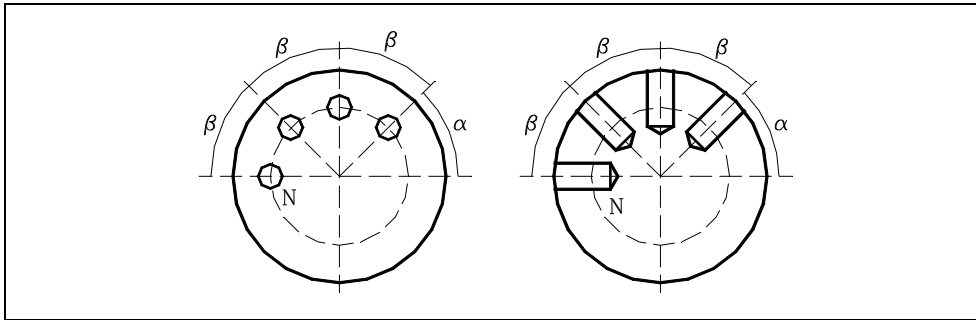11. The CNC stops the spindle, but it keeps the machining conditions set; tool (T), feedrate of the axes (F) and spindle speed (S).

3.10.5   Multiple threading cycle. Basic operation

The machining steps in this cycle are as follows:

1. If the spindle is working in open loop (RPM or CSS mode) the CNC stops the spindle and performs a home search on the spindle (Io).

2. If the operation 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.

3. The CNC starts the live tool at the selected speed and in the indicated direction.

4. Orients the spindle at the angular position for the first hole (indicated by α).

5. The tool approaches in rapid up to the starting point, keeping the safety distance selected according to axes X and Z.
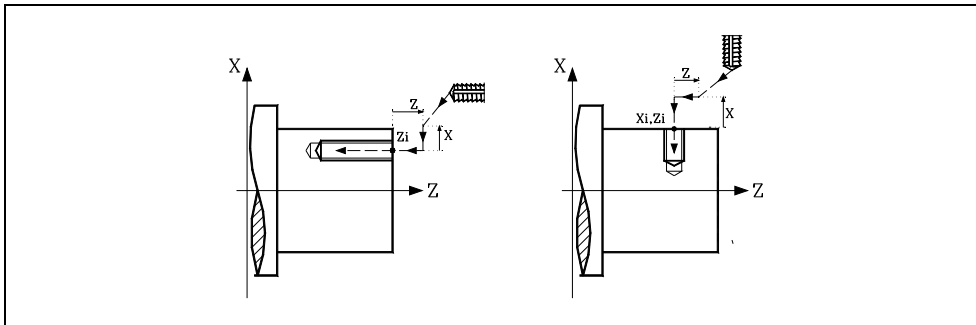
6. Tapping of the part in work feedrate F, until the depth L is reached.

    Tapping (with a clutch).

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

7. Reversal of the turning direction of the live tool.

8. Withdrawal in work feedrate to the approach point.

9. Depending on the value assigned to parameter N (number of holes), the CNC repeats the following steps N times.

  – The spindle moves to the next drilling point (angular increment β).

  – Repeats the drilling movements indicated in steps 6, 7 and 8.

10. 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.

11. 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.

3.10.6 Multiple slot milling cycle. Basic operation

The machining steps in this cycle are as follows:

1. If the spindle is working in open loop (RPM or CSS mode) the CNC stops the spindle and performs a home search on the spindle (Io).

2. If the operation 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.

3. The CNC starts the live tool at the selected speed and in the indicated direction.

4. Orients the spindle at the angular position for the first hole (indicated by α).

5. The tool approaches in rapid up to the starting point, keeping the safety distance selected according to axes X and Z.
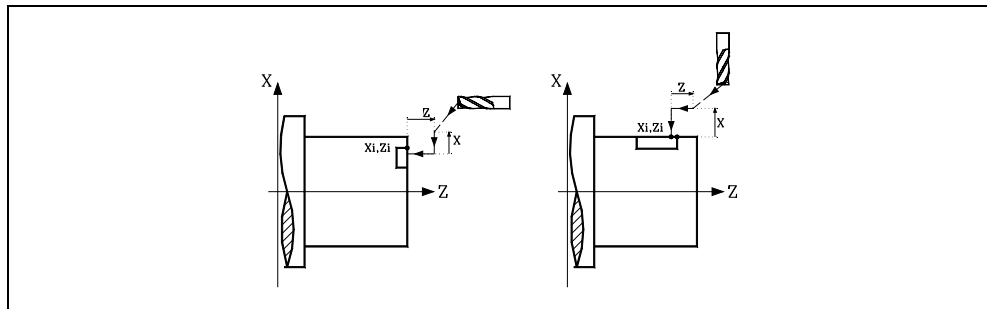
6. Milling of the slot by following these steps:
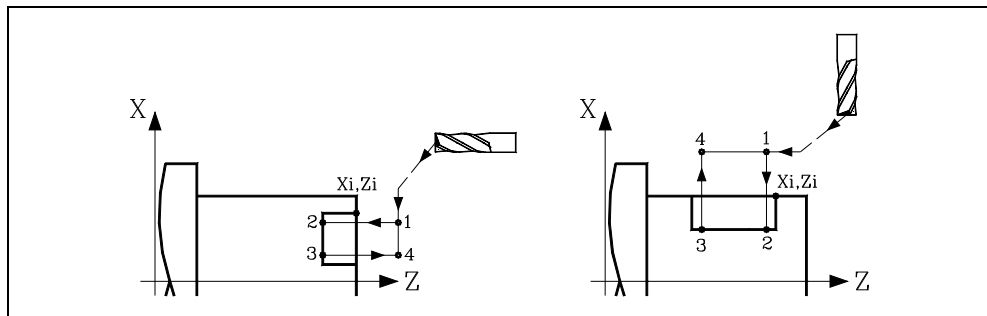
 – Penetration at programmed feedrate F to the bottom of the slot (section 1-2).

 – It mills the slot by moving the X or Z axis (accordingly) at the programmed "F" feedrate (section 2-3).

 – Withdrawal to the approach point (sections 3-4 and 4-1).

7. Depending on the value assigned to parameter N (number of holes), the CNC repeats the following steps N times.

 – The spindle moves to the next drilling point (angular increment β).

 – Repeats the drilling movements indicated at the step 6.

8. 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.

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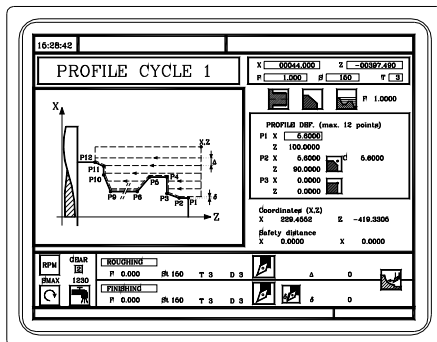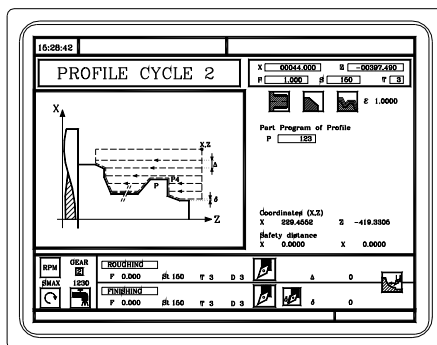
3.11   Profiling cycle

This key accesses the profiling cycles.

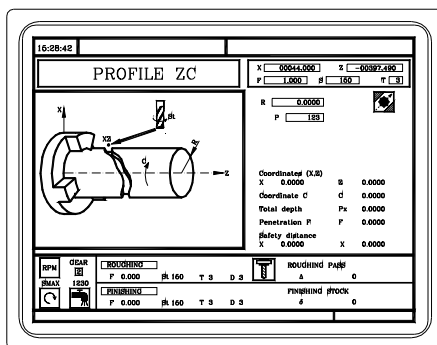This cycle may be defined in several ways.

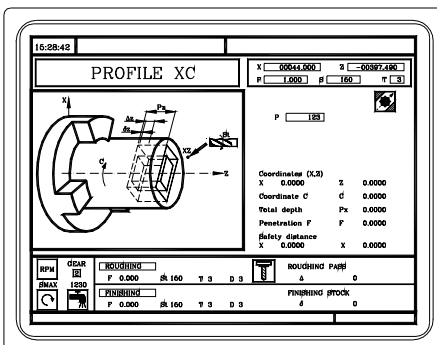### *Level 1. Defining all the points of the profile.*

Level 2. Using a part-program which contains the profile.

Level 3. ZC profile. Available when having a "C" axis.

Level 4. XC profile. Available when having a "C" axis.
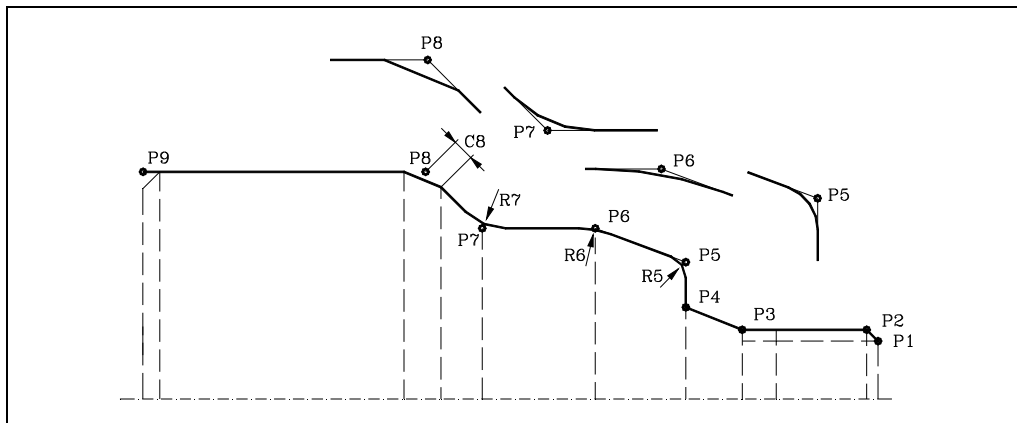
FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

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 profile starting point. The remaining points must be correlative.

Use the [⬇] [⬆] keys to select and exit the window that contains the profile defining points and the [⬅] [➡] 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 does not take into account the type of machining of the last profile point.

- 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 two-color 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.

At all the intermediate points of the profile the type of machining required to be done on the corner must be defined. To select the type of corner, place the cursor over this icon and press the two-color key.
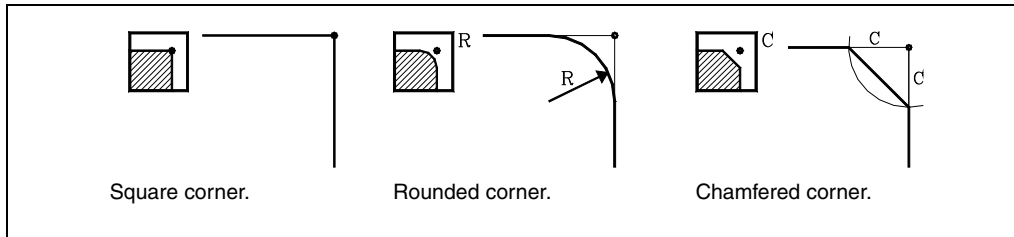
Square corner.　　　　　　Rounded corner.　　　　　　Chamfered corner.

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

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

3.11.2   Levels 2, 3 and 4. 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).

After editing the profile, the CNC requests the comment to be associated with the "Profile program" that has been 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 [P.PROG]. 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.

Modify an already 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:

- Add new items at the end of the present profile.
- Modifying the data of any element.
- Modify or include chamfers, rounded corners, 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.
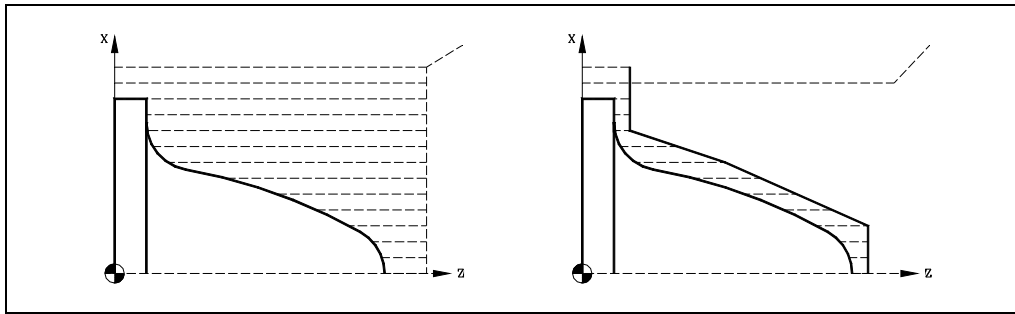
When the part profile is known, it is recommended to define both profiles, that of the rough part and the desired final profile. The machining is faster since only the stock between both profiles is removed.

To define both profiles, proceed as follows:

1. Access the profile editor.
2. Edit the desired final profile.
3. Press the softkey for a new profile.
4. Edit the rough part profile.
5. Exit the profile editor saving the profile.

Remember that the final profile must be defined first and then the rough profile.

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 two-color key.

Inside profile.
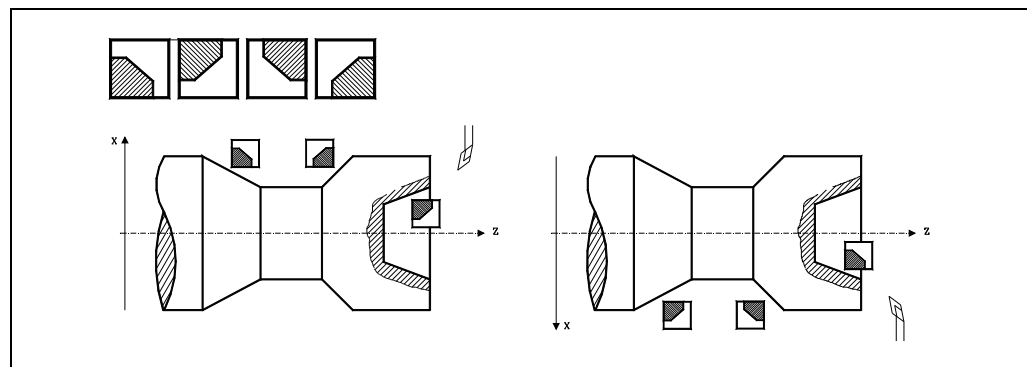
Outside profile.

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 two-color key.

Type 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 two-color key.

F 1.0000   Paraxial machining.

ε 1.0000   Machining following the profile

When changing the type of machining, the CNC changes the icon and shows the corresponding help screen.
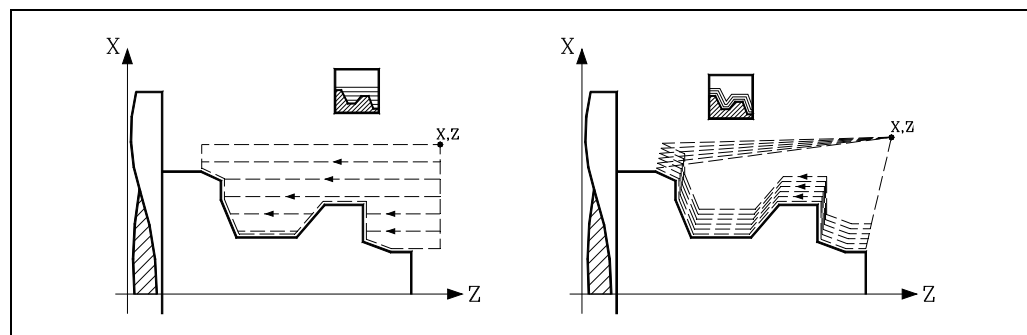
In paraxial machining you must define the feedrate (F) for the tool to penetrate into the valleys. The machining feedrate will the one indicated in the roughing and finishing windows.

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

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].

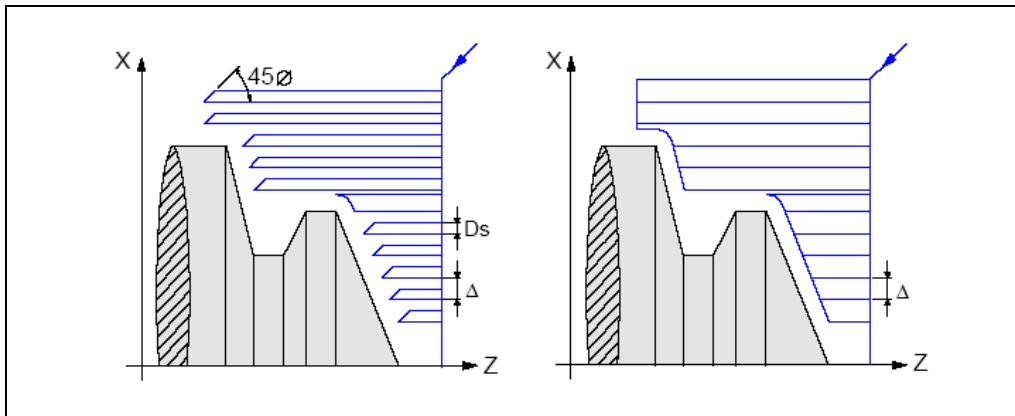 The top right window shows the tool position at all times.

Safety distance.

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

To change one of these values, place the cursor on the corresponding data, key in the desired value and press [ENTER].
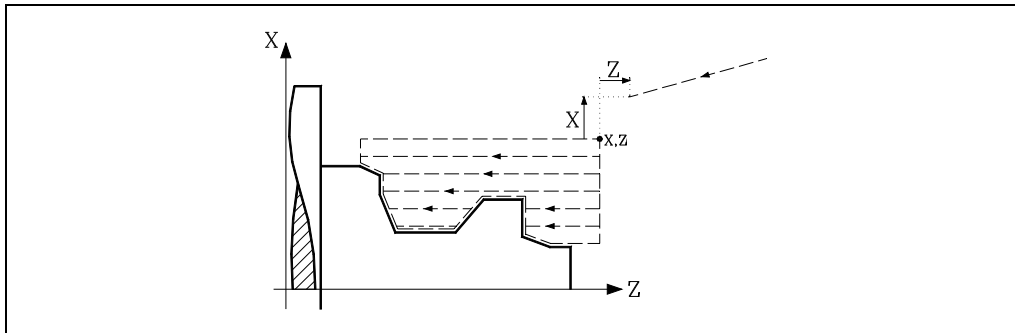
Withdrawal distance at 45º (Ds).

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 45º (left figure).

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 two-color key.
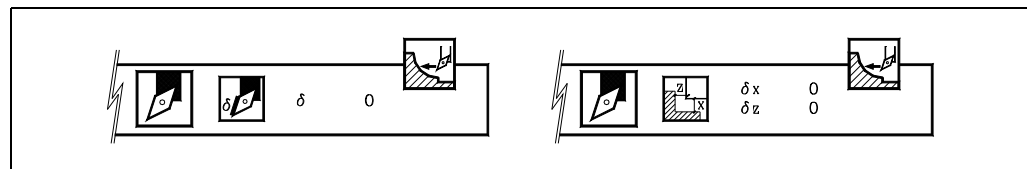

| Turning direction | Facing direction |

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 finishing stocks, place the cursor over this icon and press the two-color key. To define these residual stocks, place the cursor on the corresponding data, key in the desired value and press [ENTER].

Roughing by segments in inside cycles

Two problems may occur when roughing inside profiles in the entire profile:

- It could be hard to remove the chips if the profile is very deep.

- Since the tool is long to reach the bottom of the profile, it could shake while machining.

With roughing by segments, there will be more room to remove the chips and shorter and a more robust tools may be chosen to avoid vibrations.

In profiling cycles 1 and 2, roughing by segments on the Z axis is only possible inside and machining penetration along the Z axis. A distance must be programmed in inside profiling, pattern repeat and paraxial movements. The cycle adjusts the number of passes according to the programmed distance and to the pass defined with the parameter "Dp".

The finishing pass is the same for all segments. If parameter Dp=0, there are no passes by segments.

To enable this option, set the general machine parameter COCYPROF (P157)=1.

FAGOR ●

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2x

·162·

3.11.5 Definition of geometry at levels 3 and 4. XC, ZC profiles

Milling with or without tool radius compensation.

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

Without compensation.

With lefthand tool radius compensation.

With righthand tool radius compensation.

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

Radius.

It indicates the outside radius of the part. Place the cursor over this data, key in the desired value and press [ENTER].

Spindle starting angle (W).

Indicates the initial angle of the spindle in C axis cycles. Position the cursor on this data, enter the desired value and press the [ENTER] key.

Total depth (Px).

The total depth of the thread is programmed with a positive value and in radius (ZC profile). To define that value, place the cursor on this data, key in the desired value and press [ENTER].

Penetrating feedrate (F).

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

Safety distance.

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

To change one of these values, place the cursor on the corresponding data, key in the desired value and press [ENTER].
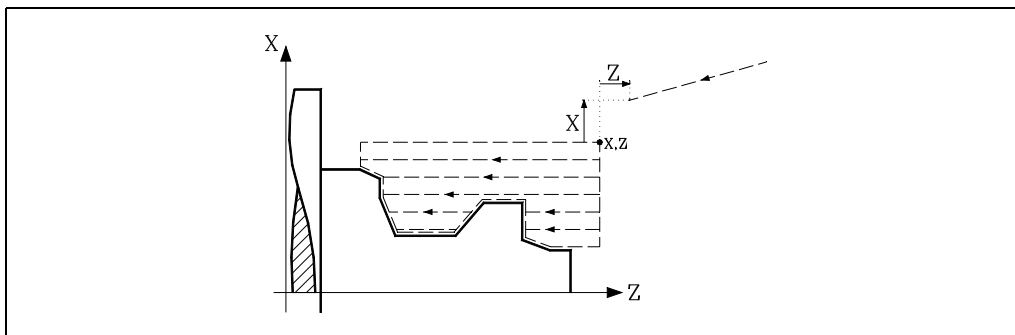
Finishing stock at the bottom (δx, δz).

The finishing stock at the bottom may be defined when working with tool radius compensation. To define these residual stocks, place the cursor on the corresponding data, key in the desired value and press [ENTER].

Side stock (δ) and number of passes (N) to remove it.

The side stock may be defined when working with tool radius compensation. To define these residual stocks, place the cursor on the corresponding data, key in the desired value and press [ENTER].

3.11.6 Basic operation at levels 1 and 2. ZX profile

The machining steps in this cycle are as follows:

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 with the speed selected and in the direction stated.

3. The tool approaches in rapid up to the starting point, (X, Z) keeping the safety distance selected as per axes 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).

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. This means that after approaching the finishing operation will be carried out.

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).

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

·164·

3.11.7   Basic operation at levels 3 and 4. XC, ZC profiles

The machining steps in this cycle are as follows:

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 tool approaches in rapid up to the starting point, (X, Z) keeping the safety distance selected as per axes X and Z.

3. The spindle orients to the indicated "C" position.

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

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.

   Finishing of the bottom of the profile removing the bottom excess stock (δx or δz). Then finishing of the side wall, removing the excess stock in successive passes.

   The finishing of the part is carried out with the machining conditions set for finishing; feedrate of the axes (F), speed of the live tool.

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.

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

3.11.8 Example. Level 1

Geometry definition.

Outside profile.

Type of machining.

Work quadrant.

Profile definition.

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

3.11.9   Examples. Level 2

Geometry definition.

	Outside profile.
	Type of machining.
	Work quadrant.

Profile definition.

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

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

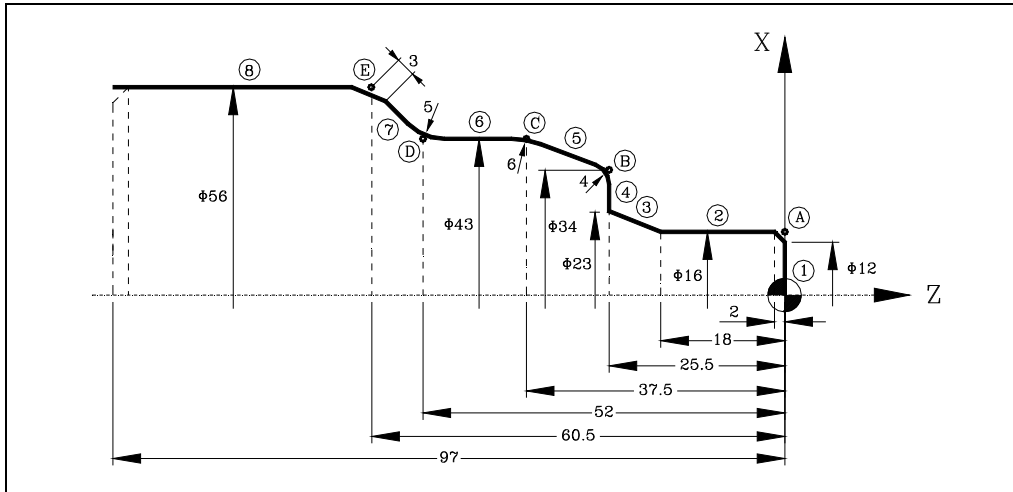*Spindle.*

RPM

Geometry definition.

Outside profile.

Type of machining.

Work quadrant.

Profile definition.

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

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·**TC**· **O**PTION
SOFT: **V02.2**X

Roughing.

F 1.000   S 1000   T 3 Δ 2

Finishing.

F 0.800   S 1000   T 3 δ 0.25
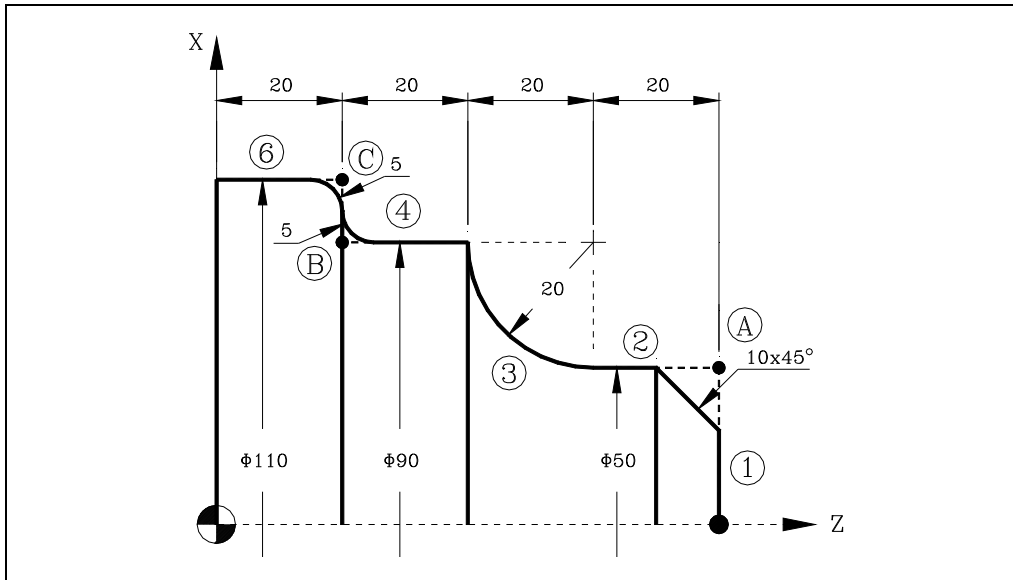
Spindle.

RPM

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

·170·

Geometry definition.

 Outside profile.

 Type of machining.

 Work quadrant.

Profile definition.

Abscissa and ordinate of the starting point.				Z = 170	X = 0
Section 1	Cntrclock- wise arc	Zc = 140	Xc = 0	R = 30.	
Section 2	Cntrclock- wise arc			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 line	Z = 20	X = 220		Tangent = Yes
The CNC displays the possible options between sections 3-4. Select the correct one.					
Section 5	Straight line	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

Geometry definition.

Outside profile.

Type of machining.

Work quadrant.

Profile definition.

Abscissa and ordinate of the starting point.				Z = 170	X = 0
Section 1	Cntrclock- wise arc	Zc = 140	Xc = 0	R = 30.	
Section 2	Cntrclock- wise arc			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 line	Z = 20	X = 220		Tangent = Yes
The CNC displays the possible options between sections 3-4. Select the correct one.					
Section 5	Straight line	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 line	Z = 180	X = 60
Section 7	Straight line	Z = 90	X = 140
Section 8	Straight line	Z = 30	X = 180
Section 9	Straight line	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

Geometry definition.

Outside profile.

Type of machining.

Work quadrant.

Profile definition.

Abscissa and ordinate of the starting point.		Z = 180	X = 0	
Section 1	Cntrclock- wise arc	Zc = 150	Xc = 0	R = 30.
Section 2	Straight line	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 line	Angle = 160		Tangent = Yes
Section 5	Clockwise arc	Z = 30	Zc = 45	R = 15. Tangent = Yes
		X = 80	Xc = 80	
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 line	Z = 30	X = 100	
Section 7	Straight line	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.

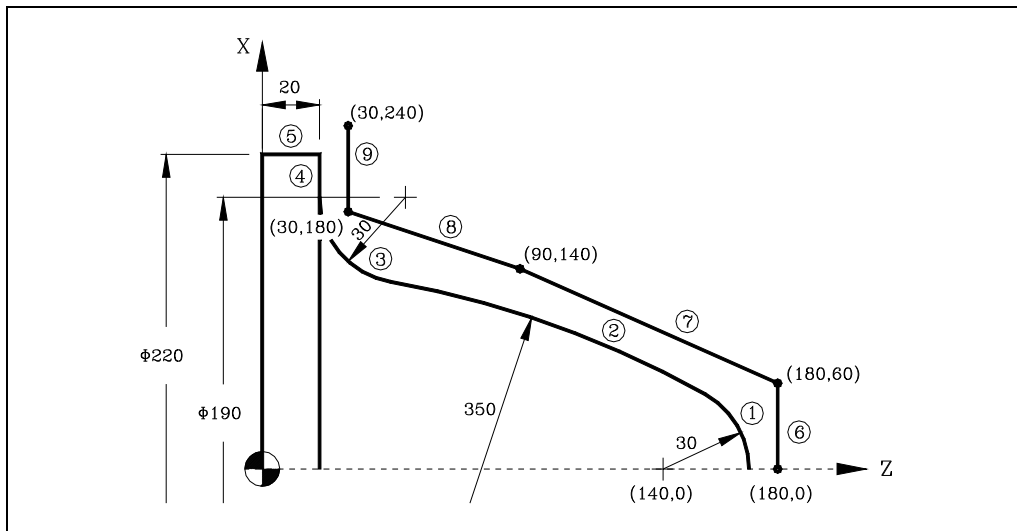F 0.800   S 1000   T 3 δ 0.25

Spindle.

RPM

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·**TC· OPTION**
SOFT: V02.2x

·**174**·
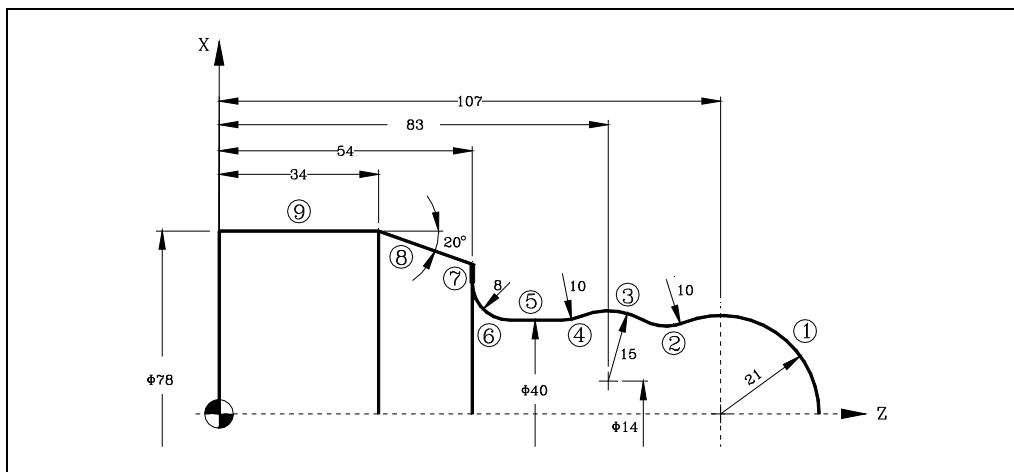
Geometry definition.

 Outside profile.

 Type of machining.

 Work quadrant.

Profile definition.

Abscissa and ordinate of the starting point.			Z = 128	X = 0
Section 1	Cntrclock- wise arc	Zc = 107	Xc = 0	R = 21.
Section 2	Clockwise arc	R = 10.		Tangent = Yes
Section 3	Cntrclock- wise 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 line	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 line	Z = 54	Angle = 90	Tangent = Yes
Section 8	Straight line	Z = 34	X = 78	Angle = 160
Section 9	Straight line	Z = 0	X = 78	

Coordinates (X, Z).

X 85.0000   Z 135.0000

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

Y AXIS

Without the Y axis option, the Y axis will only be available without cycles, calibration or Y axis compensation.

Both for programming and displaying, the Y axis format will always be determined as radius, not as diameter.

4.1   Profiling cycles with Y axis

Profile cycles in YZ and XY may be accessed with their relevant icons from either of the two levels of profile cycles ZC and XC respectively.

XY profiling cycle	YZ profiling cycle
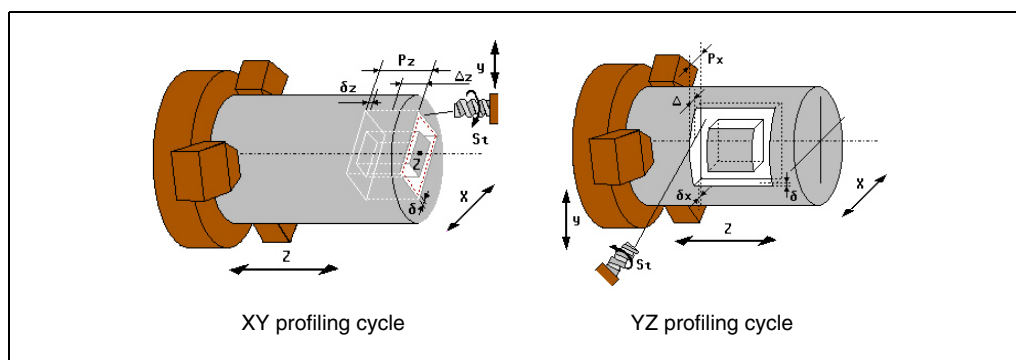
 This icon selects the planes of the Y axis (YZ, XY). Selecting a plane of the Y axis changes the graphics on the screen.

The parameters of the cycles in the Y axis planes are equivalent to those in the C plane (ZC, XC), except for the radius (R) of the ZC profile that is not used in the YZ plane and it uses the X coordinate, instead.

A data is added in the two cycles that use the Y axis to indicate the starting angle of the spindle (W).

4.2   Graphics: XY and ZY plane selection

The XY and ZY planes may be selected using the relevant softkeys shown in the <graphics> window similar to that of the C axis planes, XC and ZC.

4.3   Tool calibration

There are three tool calibration levels in conversational TC mode managing the Y axis:

Level 1: Manual tool calibration and with a probe.

Selecting the manual tool calibration using a known part shows, with Y axis, the following screen:
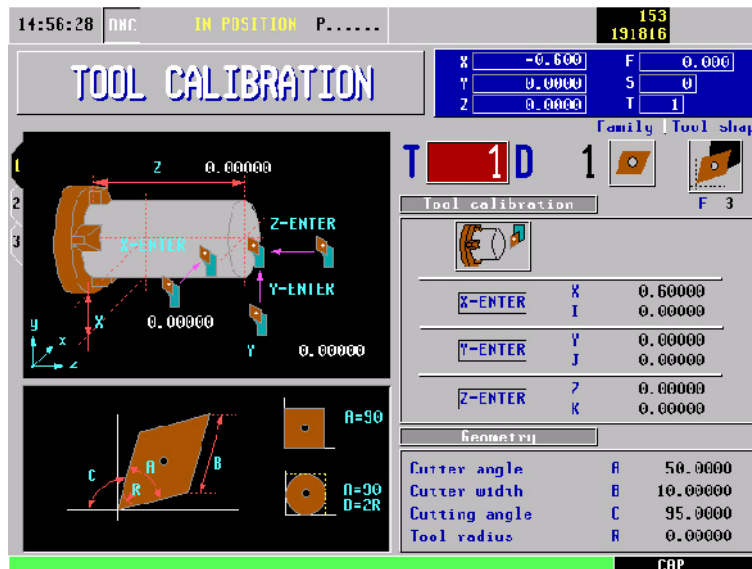
Observe that the <tool calibration> area of this window shows the label "Y-ENTER" and the "Y" and "J" fields.

To make the measurement, once the tool has been moved manually until touching the part and [ENTER] is pressed inside the Y field, the CNC calculates the new length and assigns it to the field of the active tool offset. Parameter J is set to 0.

Selecting the manual tool calibration using a probe shows, with Y axis, the following screen:
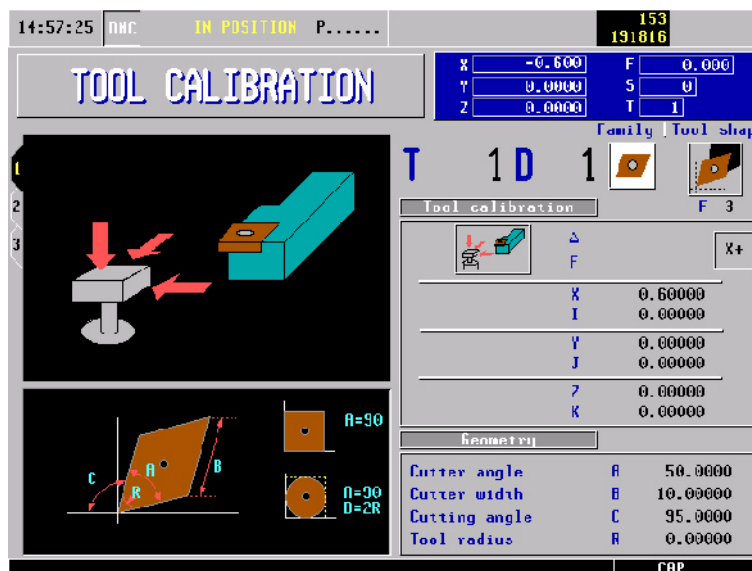
When having a Y axis, just like with the rest of the axes, it is possible to enter the approach distance "Δ", the approach feedrate "F" and the side of the probe being probed "Y+" or "Y-". If no value is entered for "Δ", it takes the value of g.m.p. PRBMODE. Likewise, if no value is entered for "F, it takes the value of a.m.p. PRBFEED.

Observe that the <tool calibration> area of this window shows the "Y" and "J" fields and the icon "Y+", "Y-".

When done probing, the CNC calculates the new length that is updated in the Y field on the screen. Parameter J is set to 0.

Level 2: Tool calibration with probing cycles.

Selecting the tool calibration using probing cycles shows, with Y axis, the following screen:
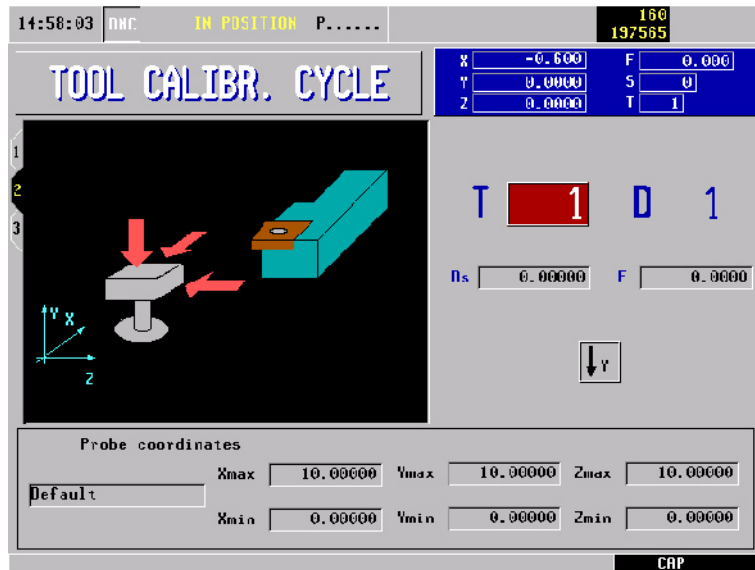
The < ↓Y > icon turns on or off the display of the probe's Y coordinates at the bottom of the screen. When selecting "display the Y axis", the calibration is also carried out on this axes. The Y axis coordinates will be displayed by default.

Level 3: Probe calibration.

Selecting the probe calibration shows, with Y axis, the following screen:
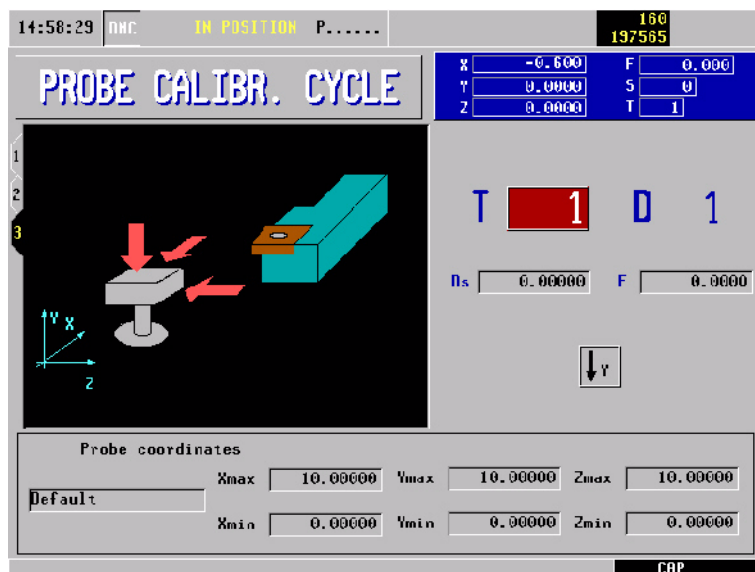
The < ↓Y > icon turns on or off the display of the probe's Y coordinates at the bottom of the screen. The Y axis coordinates will be displayed by default.

4.

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

OPERATING IN ISO MODE

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 *"2.11 ISO management"* on page 72.

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.

| LEVEL CYCLE | To change levels, use the [LEVEL CYCLE] key or the [page up] and [page down] keys to scroll up and down through the various levels. |
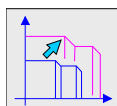
ISO   Block editing   Scaling factor
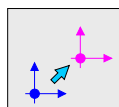
Zero offsets and presets   Work zones
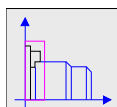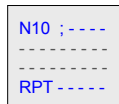
N10 ; - - - -   Insert labels and repetitions   Mirror image
RPT - - - - -   from label to label
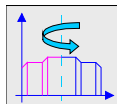
Simulating, executing or saving ISO cycles

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 [GRAPHICS] key.

- To execute the blocks, press the [START] key.

- To save the edited blocks, press the [P.PROG] key.

It is possible to combine ISO-coded blocks with standard and/or user machining cycles to create part-programs. See chapter *"6 Saving programs"*.

5.1   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   ENTER

G0 Z100   ENTER

G1 X30 F0.1   ENTER

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.

5.2   Programming assistance

5.2.1   Zero offsets and presets
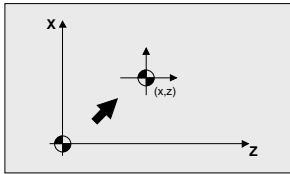
The icon may be used to select the following options:

- 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 you do not wish to preset any of them, leave its field empty (blank).

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

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

5.2.2   Work zones
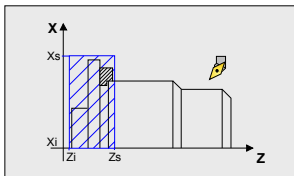
The icons may be used to defined the following options:

- 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.

5.2.3   Insert labels and repetitions from label to label
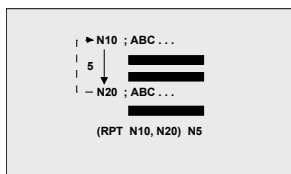
This cycle permits editing labels and repetition blocks between two labels. It is selected with an icon.

- Edit labels. There is a field to enter the block number an 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.

5.2.4 Mirror image
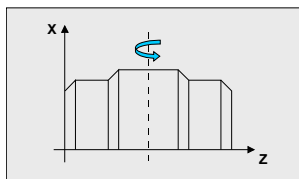
The icons may be used to select the following options:

- 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 on which mirror image is applied. This icon is not shown when selecting mirror image cancellation.

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

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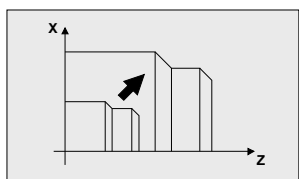
An icon is used to indicate whether to cancel the existing scaling factor or activate a new one. In the latter case, a numeric field is displayed to define the value of the scaling factor.

The cycle internally generates a block with function G72.

SAVING PROGRAMS

6

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

6.1 List of saved programs

P.PROG

Press [P.PROG] 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
```
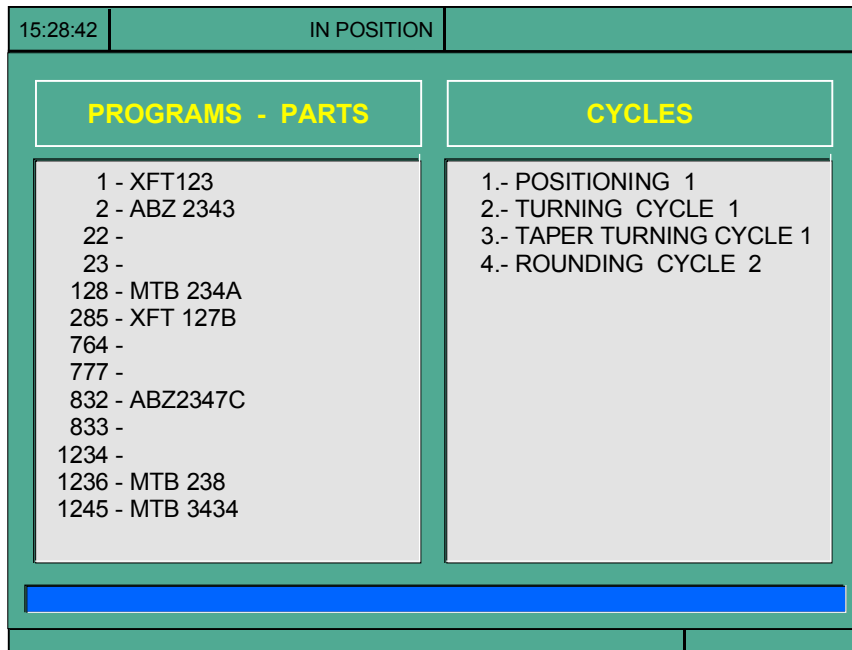
Press one of these keys to exit the directory or list of part-programs:

[ESC] [ISO] [▭] [▭] [▭] [▭] [▭] [▭]
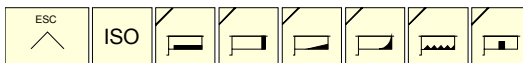
List of part-programs.

The left side shows the list of part-programs that are saved in the CNC's memory. Use the [⬆][⬇] keys to move the pointer over the list of programs. Use the [SHIFT][⬆] y [SHIFT][⬇] 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.

6.2   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 [⬆][⬇] keys. The right column shows the cycles that make up the part.

> ENTER
>
> When pressing [ENTER] or one of the [⬅][➡] keys, the pointer switches to the right column. Now, the [⬆][⬇] keys may be used to move the pointer over the blocks or cycles that make up the part.

> **i**
>
> *So, use the* [⬆][⬇] *keys to move up and down on each column and* [⬅][➡] *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.

6.2.1   Seeing one of the operations in detail

> RECALL
>
> 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 *"7 Execution and simulation"*.
- Execute the operation. See chapter *"7 Execution and simulation"*.
- To modify the operation.
- Save the operation. Replacing the previous one or inserting it as a new one.

6.3   Edit a new part-program

To edit a new program, proceed as follows:

P.PROG

1. Press [P.PROG] 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 [P.PROG] 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.

ENTER

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.

ESC

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.

6.4   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 [P.PROG] 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 cycle 2	Taper cycle 2
	Turning cycle
Rounding cycle 2	Rounding cycle 2
Taper cycle 1	Taper 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].

6.5   Delete a new part program

To delete a part-program, proceed as follows:

1. Press [P.PROG] to access the list of part-programs stored.

2. Select, with the pointer, the part-program to be deleted from the left column.

CLEAR

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.

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2x

6.6   Copying a part-program into another one

To copy a part-program into another one, proceed as follows:

1. Press [P.PROG] 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 [P.PROG] 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.

6.7   Modify a part-program

To modify a part-program, proceed as follows:

1. Press [P.PROG] 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.

6.7.1   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.

6.7.2   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 [P.PROG] 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].

6.7.3   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 two-color 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].

Example:

Current program	Desired program
Positioning cycle 2	Positioning cycle 2
Facing cycle	Taper cycle 2
Taper cycle 2	Turning cycle
Turning cycle	Rounding cycle 2
Rounding cycle 2	Taper cycle 1
Taper 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].

6.7.4   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 [P.PROG] to access the list of part-programs stored.

    The CNC shows the pointer on the same operation. To select another position, use the [⬆][⬇] 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

i   *It is possible to select an existing operation, modify it and then insert it in another position and even in another part-program.*

6.

SAVING PROGRAMS
Modify a part-program

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2x

·194·

6.8   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:
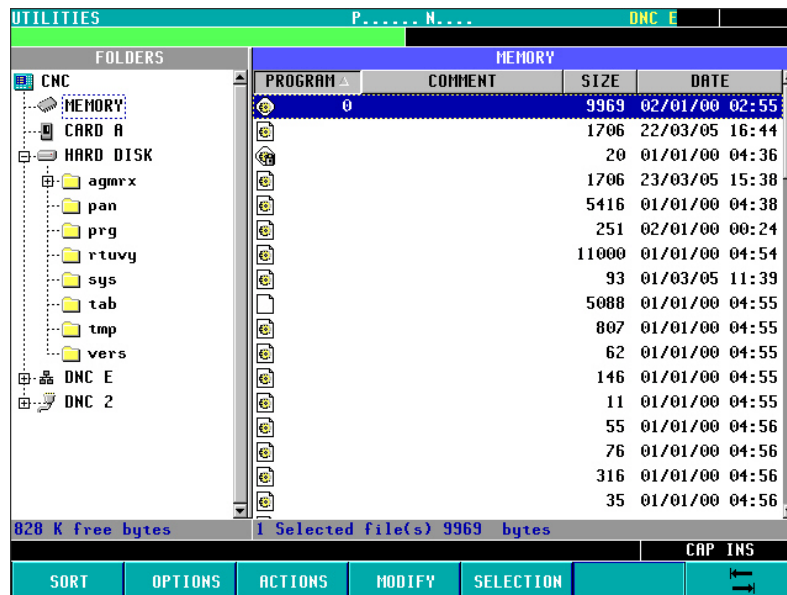
Once in the explorer, it will be possible to select any program of the Ram memory or hard disk (KeyCF) to edit it or execute it. When selecting the hard disk (KeyCF), the CNC will select the PRG folder automatically because it is the only folder where programs may be executed.

> *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.*

6.

FAGOR ⋑

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

EXECUTION AND SIMULATION

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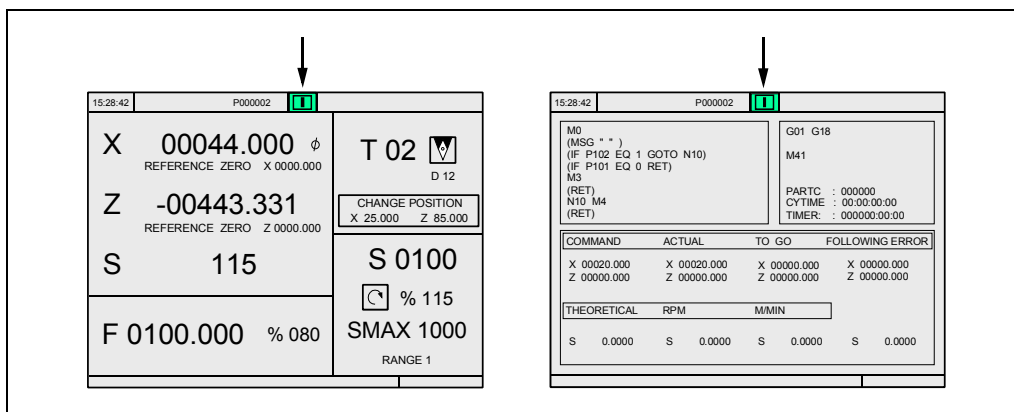
When the top center window shows the part-program selected next to the green "start" symbol, the CNC acts as follows:

- 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.

7.1   Simulating or executing an operation or cycle

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

Editing mode   Execution mode
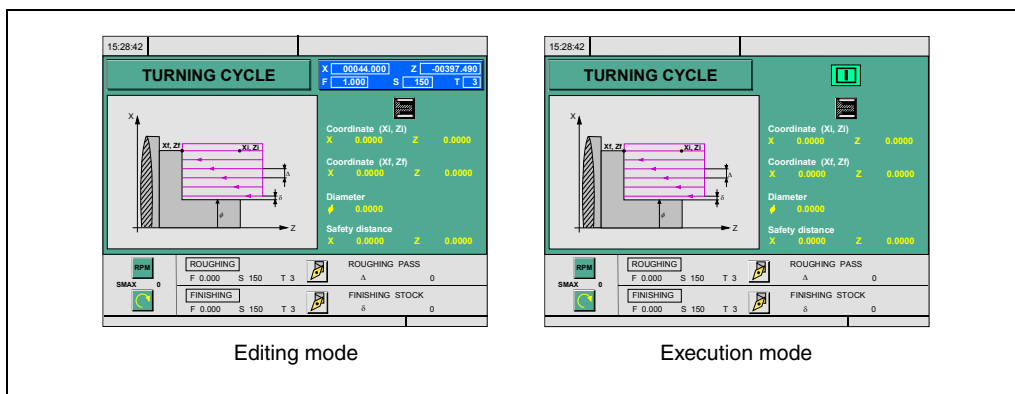
Simulation

GRAPHICS

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.

7.2   Simulating or executing a part-program

Proceed as follows to simulate or execute a part-program:

1. Press [P.PROG] to access the list of part-programs stored.

2. Select on the left column the program to be simulated or executed.

Press [GRAPHICS] 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.

7.2.1   Simulating or executing a portion of a part-program

Proceed as follows to simulate or execute a portion of a part-program:

1. Press [P.PROG] 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 [GRAPHICS] 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.

7.

EXECUTION AND SIMULATION
Simulating or executing a part-program

7.3   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 [P.PROG] 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 [GRAPHICS] to simulate the operation and [START] to execute it.

7.4   Execution mode

When pressing [START] to execute an operation or part-program, the CNC shows the standard screen of the TC mode.

```
15:28:42                    P000002    ▣

X      00044.000  ⌀          T 02  ▼
       REFERENCE  ZERO   X 0000.000         D 12

Z     -00443.331            CHANGE POSITION
       REFERENCE  ZERO   Z 0000.000         X 25.000    Z 85.000

S           115             S 0100

                            ⟳  % 115
F 0100.000    % 080         SMAX 1000
                                 RANGE 1
```

When pressing the two-color key, the CNC shows the special screen of the TC mode.

```
15:28:42                    P000002    ▣

M0                                G01  G18
(MSG " " )
(IF  P102  EQ  1  GOTO  N10)       M41
(IF  P101  EQ  0  RET)
M3
(RET)                             PARTC  : 000000
N10  M4                           CYTIME : 00:00:00:00
(RET)                             TIMER:  : 000000:00:00

COMMAND        ACTUAL        TO GO         FOLLOWING ERROR

X 00020.000    X 00020.000   X 00000.000   X 00000.000
Z 00000.000    Z 00000.000   Z 00000.000   Z 00000.000

THEORETICAL    RPM           M/MIN

S    0.0000    S    0.0000   S    0.0000   S    0.0000
```

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

While executing an operation or a part, the [GRAPHICS] key may be pressed to access the graphics display mode.

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2x

7.4.1   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.
*   Select another tool.
*   To modify the tool values.
*   To resume 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.

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.

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

·202·

7.5   Graphic representation

GRAPHICS

When the [GRAPHICS] key is pressed, the CNC displays the graphic representation page of the T model. To exit the graphic representation mode, press the [GRAPHICS] key or the [ESC] key.

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 [⬆][⬇] 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 [⬅][➡][⬆][⬇] 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.

7.

EXECUTION AND SIMULATION
Graphic representation

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

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 s [←][→] 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.

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 "GRAPHICS" key. This can be done in a whole program on the PPROG screen or in any of the particular TC or MC cycles.

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.

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).

7.

EXECUTION AND SIMULATION
Graphic representation

FAGOR ⬤

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·**TC· OPTION**
SOFT: V02.2X

7.

EXECUTION AND SIMULATION

Graphic representation

FAGOR

FAGOR AUTOMATION

CNC 8055
CNC 8055i

·TC· OPTION
SOFT: V02.2X

7.

7.

FAGOR

FAGOR AUTOMATION

AENOR
ER
Empresa
Registrada
ER-073/1994