CNC 8055 TC

Operating Manual

Ref. 0001 (in)

INDEX

3. OPERATING IN JOG MODE

4. OPERATING WITH OPERATIONS OR CYCLES

5. PROGRAM STORAGE

6. EXECUTION AND SIMULATION

APPENDIX

VERSION HISTORY (T)

(LATHE MODEL)

Date: **December 1999** ***Software Version:*** **4.0x**

FEATURE	AFFECTED MANUAL AND CHAPTERS	
Portuguese language	Installation manual	Chap. 3
Tangential control	Installation manual Programming manual	Chap. 9, Chap. 10, Appendix Chap. 6, Chap. 11, Appendix
PLC. user registers from R1 to R499	Installation manual Programming manual	Chap. 6, Chap. 7, Appendix Chap. 11
CNC status screen	Operating manual	Chap. 8
Hard Disk (HD)	Installation manual	Chap. 1, Chap. 3, Appendix
Diagnosis of the HD	Operating manual	Chap. 12,
Integrate the HD in an external PC network	Installation manual	Chap. 3
Consult directories, delete, rename and copy programs in the same or another device.	Operating manual Programming manual	Chap. 1, Chap. 7 Chap. 1
Execution and simulation from RAM, Memkey Card, HD or serial line.	Operating manual	Chap. 1, Chap. 3,
It is now possible to execute (EXEC) and open (OPEN) for editing a program stored in any device.	Programming manual	Chap. 14, Appendix
Thread repair. Reference (home) the spindle before.	Programming manual TC operating manual	Chap. 9 Chap. 4
Simulation in rapid, without assuming G95 or M3, M54, etc.	Operating manual	Chap. 3
Geometry associated with the tool offset.	Installation manual Operating manual	Chap. 3 Chap. 6
Live tool with M45 or as if it were a 2nd spindle	Installation manual	Chap. 3
PLC channel affected by another feedrate override set by PLC.	Installation manual	Chap. 11
Independent x1, x10, x100 factor for each handwheel.	Installation manual Programming manual	Chap. 4, Chap. 10, Appendix Chap. 11
Handling the Fagor HBE handwheel	Installation manual	Chap. 4, 9, 10, Appendix
Spindle synchronization (G77 S)	Installation manual Programming manual	Chap. 3, 9, 10, Appendix Chap. 5, 11, Appendix
Optimizing of profile machining.	Programming manual TC operating manual	Chap. 9 Chap. 4
(2) axes controlled by a single servo drive	Installation manual	Chap. 3, 4, 9, Appendix
G75 function affected by Feedrate override (%)	Installation manual	Chap. 3
Probe. Probe position by cycle parameters.	Programming manual	Chap. 10
Protection against deleting OEM screens	Operating manual	Chap. 7
Detecting temperature and battery voltage on the new CPU.		
TC option. ISO program management, also like MDI.	TC operating manual	Chap. 3
TC option. Coolant icon in all cycles.	TC operating manual	Chap. 4
TC option. Background editing.	TC operating manual	Chap. 4
TC option. Key codes for user cycles.	TC operating manual	Appendix

Introduction

SAFETY CONDITIONS

Read the following safety conditions in order to prevent accidents to staff and damage to this product and any products connected to it.

The equipment may only be repaired by Fagor Automation authorized staff.

Fagor Automation will not assume responsibility for any physical or material harm stemming from failure to comply with these basic safety norms.

Precautions against accidents

Before powering up the equipment make sure it is connected to ground

In order to prevent electric shocks make sure the ground connections have been properly made.

Do not work in damp atmospheres

To prevent electric shocks always work in atmospheres with a relative humidity of under 90% with no condensation at 45°C.

Do not work in explosive atmospheres

To avoid danger, physical harm or damage, do not work in explosive atmospheres.

Precautions to avoid damaging the product

Operating environment

This equipment is prepared for use in Industrial Environments, complying with directives and standards in force in the European Union.

Fagor Automation will not assume any responsibility for any damage that it may cause or undergo if it is set up in any other type of conditions (residential or household environments).

Install the equipment in a suitable place

Wherever possible, the CNC installation should be made well away from cooling liquids, chemicals, or where it may be subject to impacts that could damage this.

The equipment complies with European electromagnetic compatibility directives. We nevertheless recommend keeping it away from sources of electromagnetic disturbance, such as:

- Powerful loads connected to the same mains as the equipment.
- Nearby portable transmitter (Radiotelephones, amateur radio transmitters).
- Nearby radio/TV transmitters.
- Nearby arc welding machines.
- Nearby high voltage lines.
- Etc.

Environmental Conditions

The room temperature should be maintained in operating conditions should be between +5°C and +45°C.
The room temperature that should be maintained in non-operating conditions should be between -25°C and 70°C.

Protection devices in the equipment itself

Power Source Module
Has two fast 3.15 Amp./ 250V. external fuses (F) fitted for protecting the mains input.

Axis Module
All the digital input/outputs are protected by means of 1 fast 3.15 Amp./ 250V. external fuse (F) against overvoltages from the external power sources (over 33 Vdc.) and against connection of the power source the wrong way round.

Input-output Module
All the digital input/outputs are protected by means of 1 fast 3.15 Amp./ 250V. external fuse (F) against overvoltages from the external power sources (over 33 Vdc.) and against connection of the power source the wrong way round.

Input-output and Copy Module
All the digital input/outputs are protected by means of 1 fast 3.15 Amp./ 250V. fuse (F) against overvoltages from the external power sources (over 33 Vdc.) and against connection of the power source the wrong way round.

Ventilator module
Has 1 or 2 external fuses fitted depending on the model.
The fuses are fast (F), 0.4 Amp./ 250V. for protecting the fans.

Monitor
The type of fuse depends on the type of monitor. See the identification label on the equipment itself.

Precautions to be taken during repairs

Do not touch the inside of the equipment
Only authorized Fagor Automation staff may handle the items located inside the equipment.

Do not touch the connectors when the equipment is connected to the mains.
Before touching the connectors (input/outputs, feedback etc) make sure that the equipment is not connected to the mains.

Safety symbols

Symbols that may appear in the manual

WARNING Symbol
This goes with text describing action or operations that could give rise to accidents or damage of the equipment.

Symbols that may be found on the product

WARNING Symbol
This goes with text describing action or operations that could give rise to accidents or damage of the equipment.

ELECTRIC SHOCK Symbol
Means that the point indicated could be under electrical voltage.

GROUND PROTECTION Symbol
Means that the point indicated must be connected up to the central machine ground point for protecting people and equipment.

MATERIAL RETURNING TERMS

If the Monitor of the Central Processing Unit has to be sent back, please pack this in its original box with the original packing material. If the original packing material is not available, please pack this as follows:

1.- Obtain a cardboard box whose 3 internal sizes should be at least 15 cm (6 inches) larger than the equipment. The cardboard used for the box should withstand 170 Kg (375 pounds).

2.- If this is to be sent to a Fagor Automation office to be repaired, enclose a label with the device stating its owner, address, name of the person to be contacted, type of device, series number, symptoms and brief description of the fault.

3.- Wrap the equipment in a polyethylene roll or similar material to protect this.

If the monitor is to be shipped, provide special protection for the glass part of the screen.

4.- Pad the equipment in the cardboard box by filling this with polyurethane foam on all sides.

5.- Seal the cardboard box with packing tape or industrial staples.

FAGOR DOCUMENTATION FOR THE 8055TC CNC

The 8055TC CNC is based on the 8055T CNC, and has inside all the features of the 8055T CNC plus the specific features of the TC mode.

For this reason, it has the specific documentation for this model and all the documents for the 8055T CNC model.

CNC 8055 OEM Manual	For the manufacturer of the machine or the person in charge of carrying out the installation and set up of the CNC. This is the same for models 8055-T and 8055-TC. It has the Installation manual inside.
CNC 8055-T USER Manual	For the final user, that is, the person who is going to work with the CNC in the 8055-T mode. It has 2 manuals inside: Operation Manual describing how to operate the CNC. Programming manual describing how to program the CNC.
CNC 8055-TC USER Manual	This is for the final user, meaning the person who is going to work with the CNC in the 8055-TC mode.
DNC 8055 Software Manual	For the persons who are to use the DNC 8050 communication software option.
DNC 8055 Protocol Manual	For those who wish to make their own DNC communication, without using the DNC 8055 communication software option.
FLOPPY DISK Manual	For those who use the Fagor disc drive. This manual shows how said drive should be used.

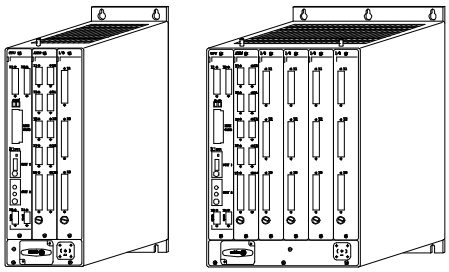
MANUAL CONTENTS

This manual is made up of the following sections:

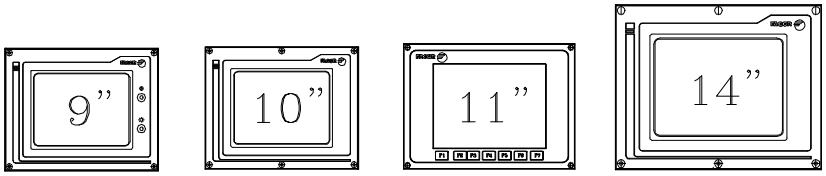
1. CONFIGURATIONS

The CNC 8055TC is modular and must have the following elements:

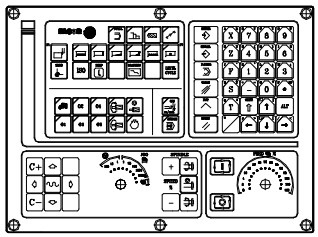
Central Unit (CPU): Is located usually in the electrical cabinet and there are 2 models: for 3 and 6 modules. For further information, see the Installation manual Chapter 1.



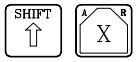
Monitor: There are several models: 9" Amber, 10" Color, 11" LCD and 14" Color. The dimensions, enclosures and connections them all are described later on in this chapter.



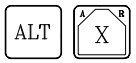
Keyboard: There is a specific keyboard to operate it in TC mode. Its dimensions and connections are described later on in this chapter.



When operating in "not TC" mode (CNC installation and start-up and standard 8055 operating mode) the access to the alphanumeric keys is rather cumbersome because one must press 2 keys for the CNC to assume the desired one.



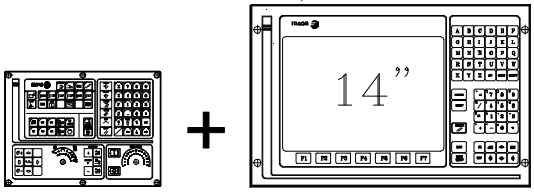
Selects the A character



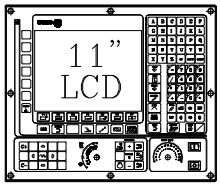
Selects the R character

In this cases, the following should be used:

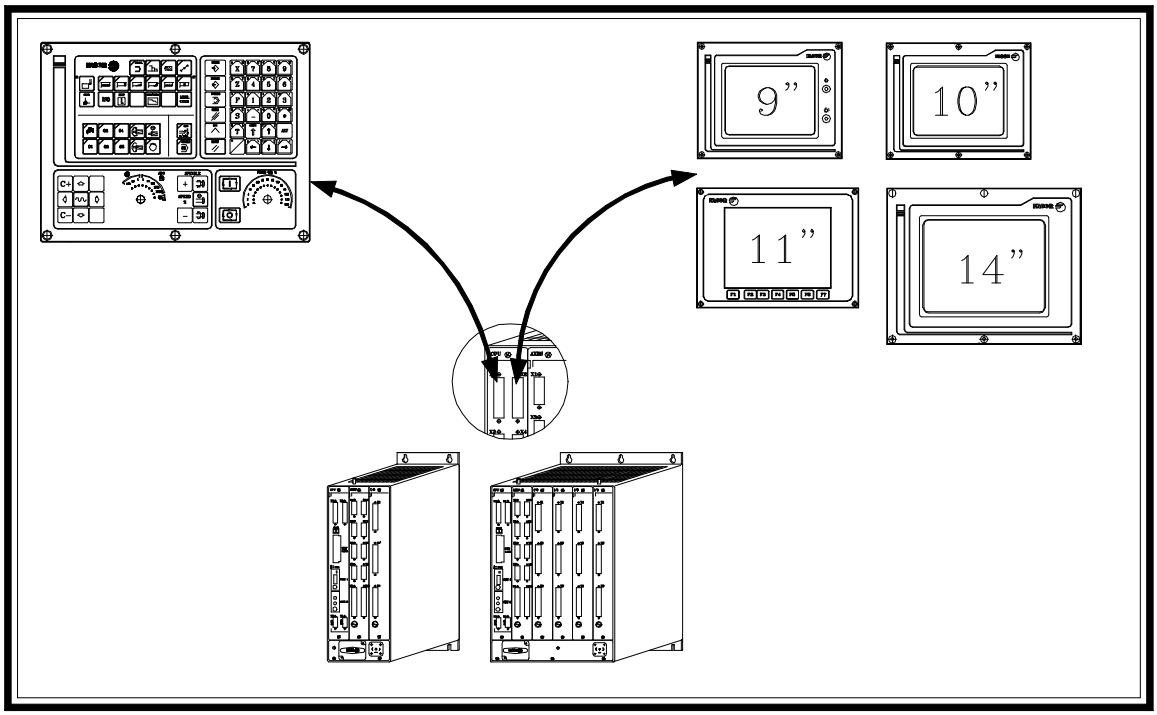
a) The TC keyboard and a 14" color monitor with alphanumeric keyboard



b) The 11" LCD monitor with full keyboard. The TC keyboard is not required.



1.1 WITH 9" AMBER, 10" COLOR, 11" LCD OR 14" COLOR MONITOR



Central Unit - Specific TC keyboard connection

It is done through connector X1 of the CPU module. Fagor Automation supplies the cable for this connection.

The characteristics of the connector are described in Chapter 1 of the Installation module (CNC configuration) in the section regarding the CPU module.

The dimensions, enclosure and connector location on the keyboard is described later on in this chapter.

Central Unit-Monitor connection

It is done through connector X2 of the CPU module. Fagor Automation supplies the cable for this connection.

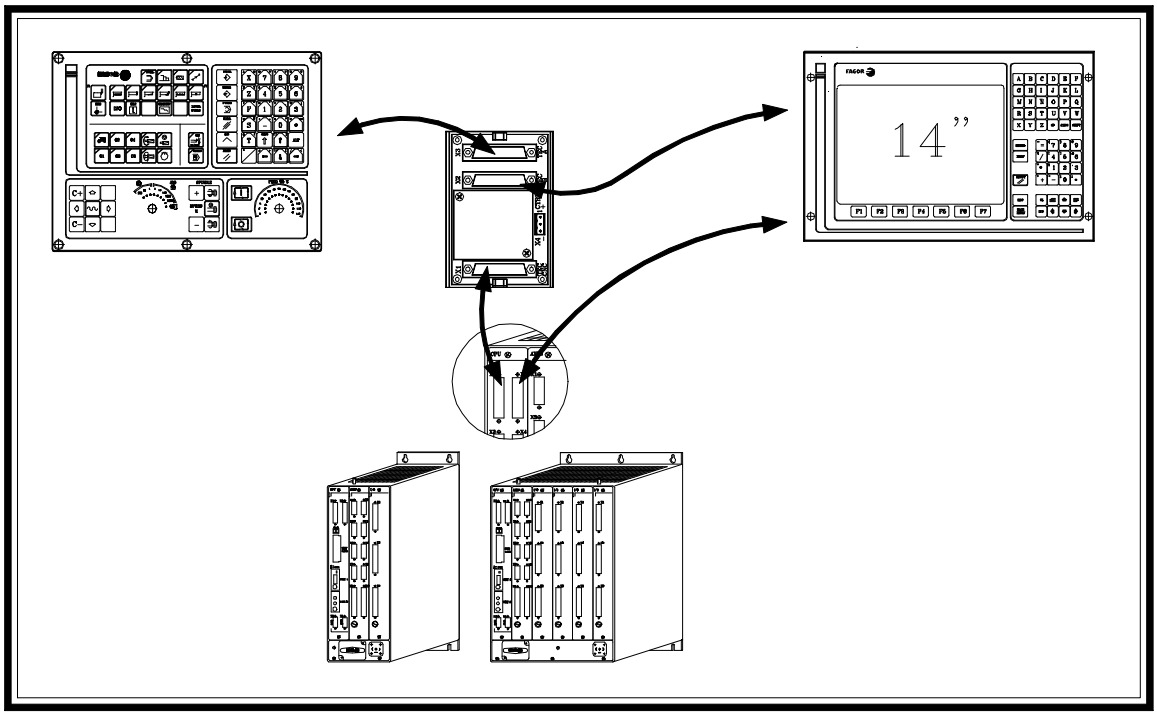
The characteristics of the connector are described in Chapter 1 of the Installation module (CNC configuration) in the section regarding the CPU module.

The dimensions, enclosure and connector location on the keyboard is described later on in this chapter.

Configuration setting.

General machine parameter CUSTOMTY (P92) = 0

1.2 WITH 14" COLOR MONITOR WITH ALPHANUMERIC KEYBOARD



Central Unit - Keyboard connection

It is done through connector X1 of the CPU module and through the keyboard switcher board.

Fagor Automation supplies the cables for this connection.

The characteristics of the connector are described in Chapter 1 of the Installation module (CNC configuration) in the section regarding the CPU module.

The dimensions, enclosure and connector location on the keyboard is described later on in this chapter.

The dimensions, connectors of the keyboard switcher board as well as how to select the keyboard active at the time is described later on in this chapter.

Central Unit - Monitor connection

It is done through connector X2 of the CPU module. Fagor Automation supplies the cable for this connection.

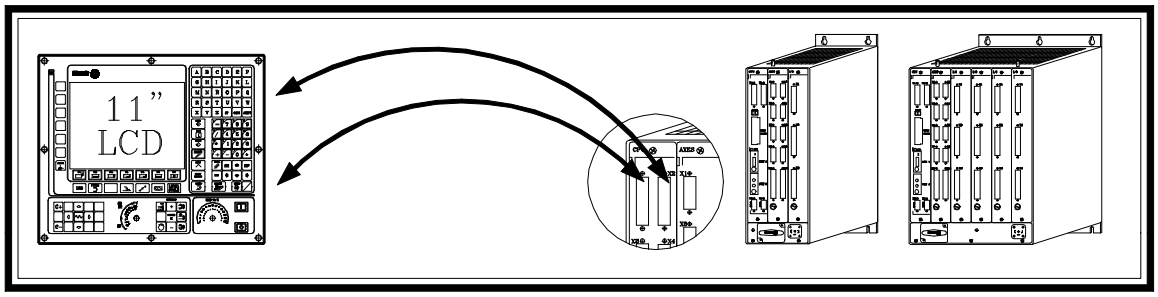
The characteristics of the connector are described in Chapter 1 of the Installation module (CNC configuration) in the section regarding the CPU module.

The dimensions, enclosure and connector location on the keyboard is described later on in this chapter.

Configuration setting

General machine parameter CUSTOMTY (P92) = 0

1.3 WITH 11" LCD MONITOR WITH FULL KEYBOARD



Central Unit - Monitor / Keyboard

It is connected to the keyboard through connector X1 of the CPU module and to the monitor through connector X2 of the CPU module.

Fagor Automation supplies the cables for these connections.

The characteristics of the connectors are described in Chapter 1 of the Installation module (CNC configuration) in the section regarding the CPU module.

The dimensions, enclosure and connection of the Monitor / Keyboard is described later on in this chapter.

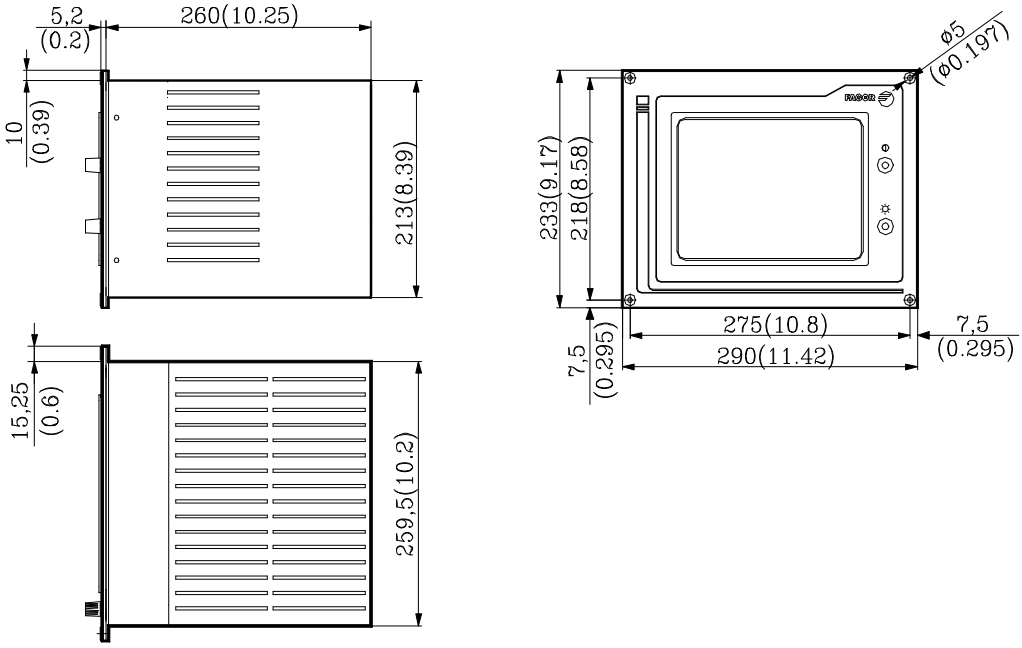
Configuration setting.

General machine parameter CUSTOMTY (P92) = 255

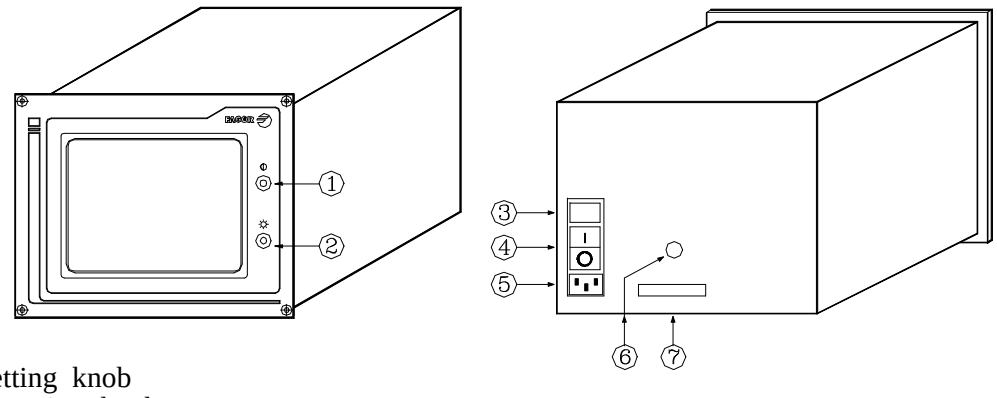
1.4 MONITORS

1.4.1 9" AMBER MONITOR

Dimensions in mm (inches):



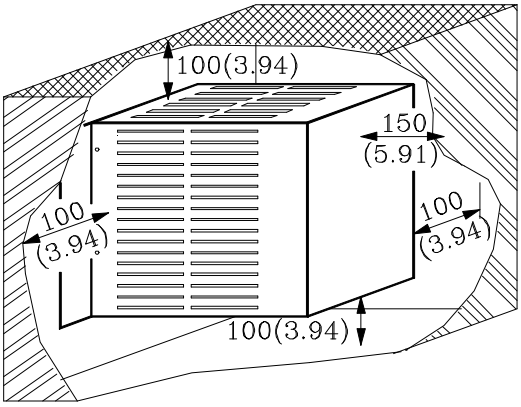
Elements:



1.- Contrast setting knob
2.- Brightness setting knob
3.- Mains fuses. 2 fast ones (F), 1 per mains phase, of 3.15Amp./250V for mains protection.
4.- ON/OFF switch
5.- Mains plug. The plug provided should be used to connect it to 220V AC and ground.
6.- Ground terminal. Used to connect the general machine ground. Metric 6mm.
7.- 25-pin SUB-D type male connector to connect it with the Central Unit.

Enclosure:

In order to guarantee proper ambient conditions, the shortest distance, in millimeters, that should be left between each of the Monitor walls and the enclosure in which this is placed, must be as follows:

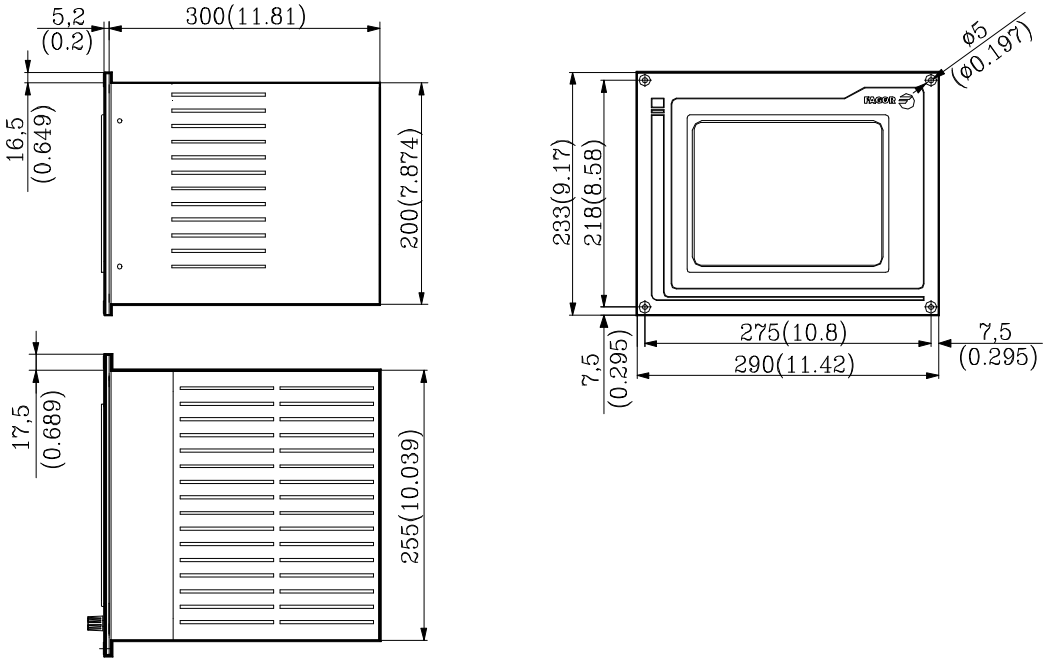


When a fan is used to improve the ventilation of the enclosure a fan with direct current motor should be used, as alternating current (AC) motors product magnetic fields which could distort the images displayed on the screen.

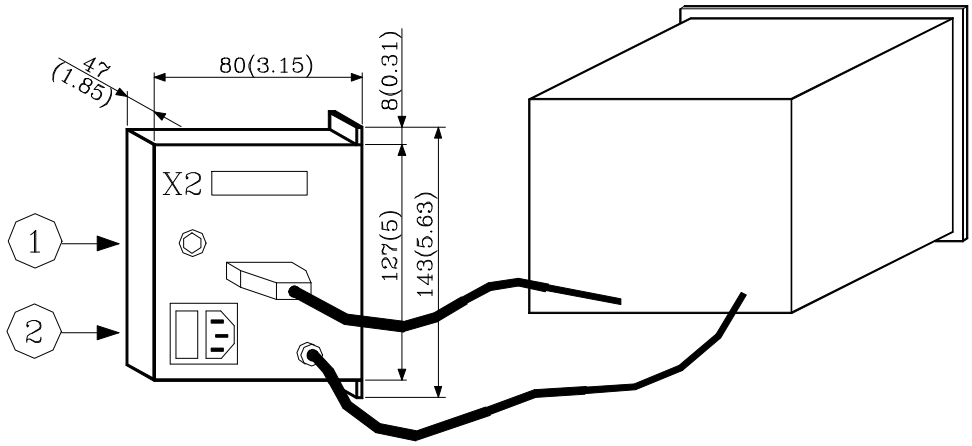
The temperature inside the enclosure should be between 0 and 50°C (32 to 122°F).

1.4.2 10" COLOR MONITOR

Dimensions in mm (inches):



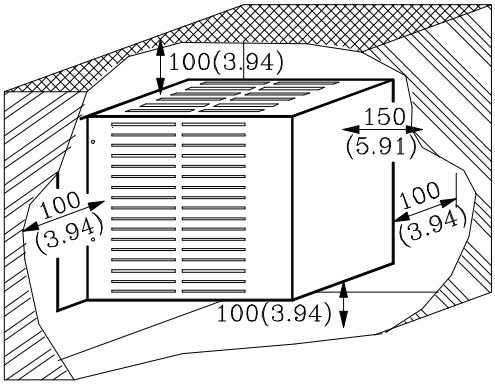
Elements:



1- Mains plug. The plug provided should be used to connect it to 220V AC and ground.
2- Ground terminal. Used to connect the general machine ground. Metric 6mm.
X2- 25-pin SUB-D type male connector to connect it with the Central Unit.

Enclosure:

In order to guarantee proper ambient conditions, the shortest distance, in millimeters, that should be left between each of the Monitor walls and the enclosure in which this is placed, must be as follows:

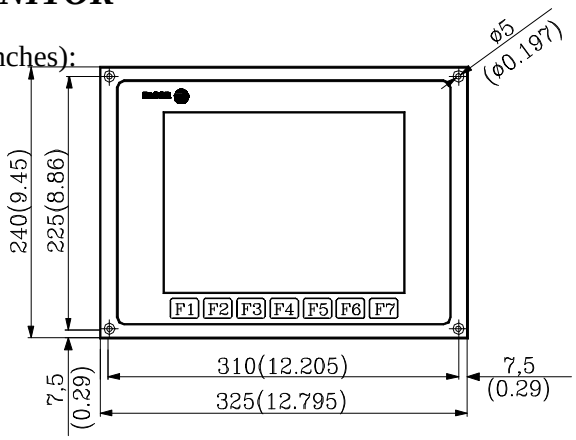


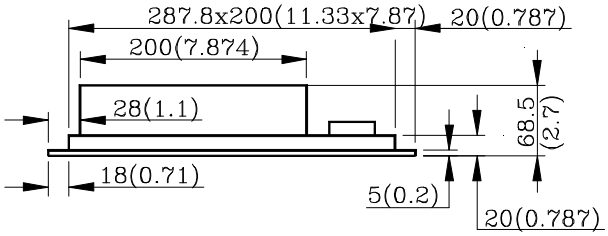
When a fan is used to improve the ventilation of the enclosure a fan with direct current motor should be used, as alternating current (AC) motors product magnetic fields which could distort the images displayed on the screen.

The temperature inside the enclosure should be between 0 and 50°C (32 to 122°F).

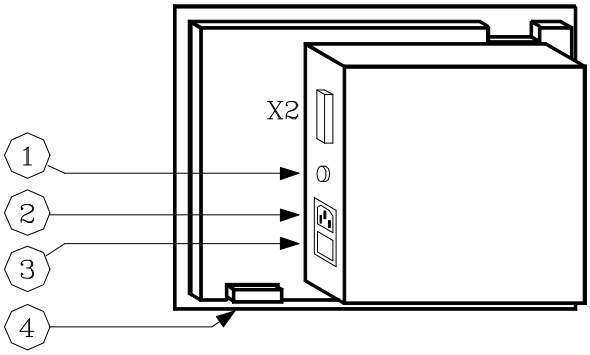
1.4.3 11" LCD MONITOR

Dimensions in mm (inches):

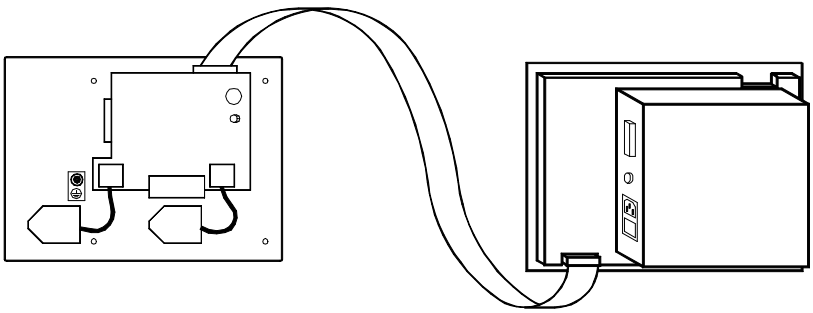




Elements:



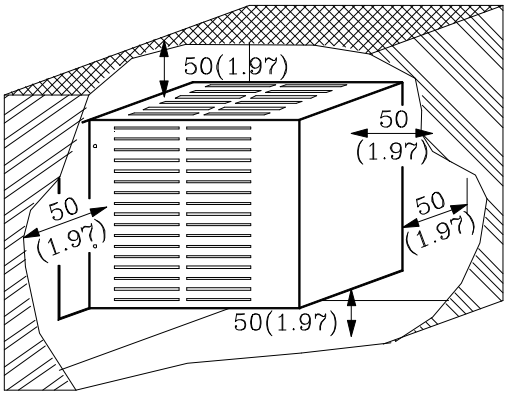
1- Mains plug. The plug provided should be used to connect it to 220V AC and ground.
2- Ground terminal. Used to connect the general machine ground. Metric 6mm.
3.- ON/OFF power switch.
4- 25-pin SUB-D type female connector to connect it with the Keyboard.



X2 25-pin SUB-D type male connector to connect the video cables to the Central Unit.

Enclosure:

In order to guarantee proper ambient conditions, the shortest distance, in millimeters, that should be left between each of the Monitor walls and the enclosure in which this is placed, must be as follows:



When a fan is used to improve the ventilation of the enclosure a fan with direct current motor should be used, as alternating current (AC) motors product magnetic fields which could distort the images displayed on the screen.

The temperature inside the enclosure should be between 0 and 50°C (32 to 122°F).

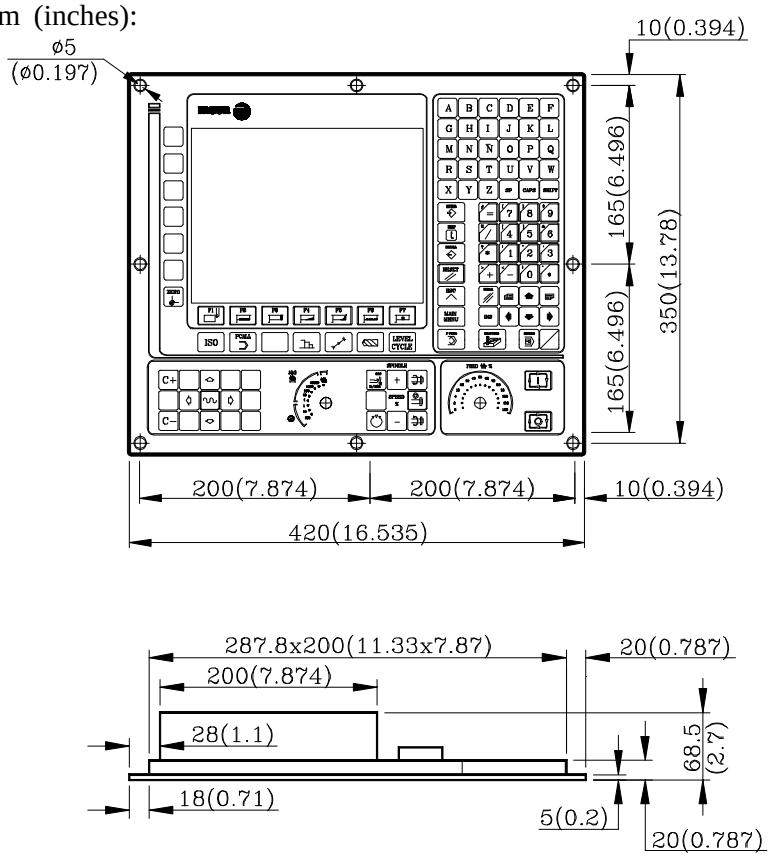
Note:

Defective Pixels.

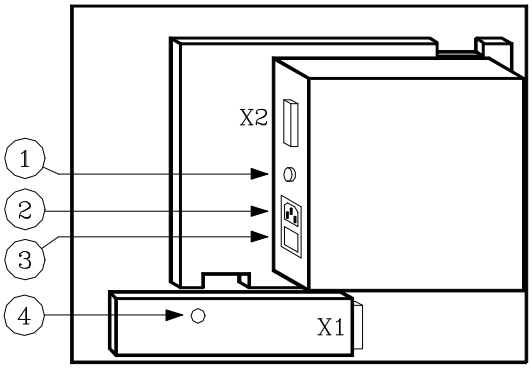
Due to the current status of the Color TFT LCD technology, all manufacturers consider good LCDs those having a certain number of defective pixels. The widely accepted criteria are basically: the number of defective pixels or sub-pixels and their concentration on the LCD surface.

1.4.4 11" LCD MONITOR WITH FULL KEYBOARD

Dimensions in mm (inches):



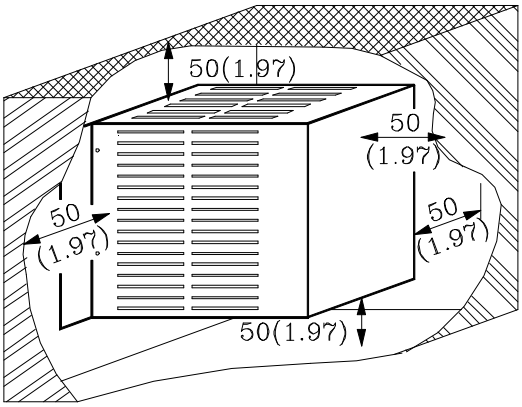
Elements:



1.- Ground terminal. Used to connect the general ground of the machine. It is metric 6mm.
2.- Mains plug for connecting 220V AC and ground.
3.- ON/OFF Power switch
4.- Buzzer.
X1 25-pin SUB-D type female connector to connect keyboard cable to the Central Unit.
X2 25-pin SUB-D type male connector to connect the video cable to the Central Unit.

Enclosure:

In order to guarantee proper ambient conditions, the shortest distance, in millimeters, that should be left between each of the Monitor walls and the enclosure in which this is placed, must be as follows:



When a fan is used to improve the ventilation of the enclosure a fan with direct current motor should be used, as alternating current (AC) motors product magnetic fields which could distort the images displayed on the screen.

The temperature inside the enclosure should be between 0 and 50°C (32 to 122°F).

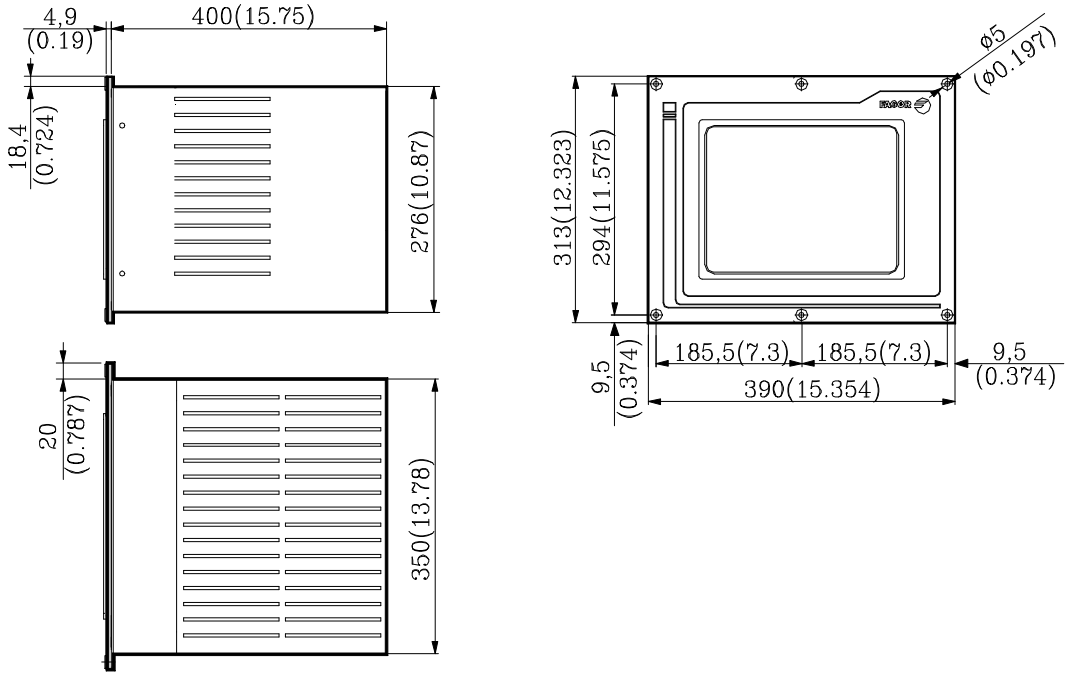
Note:

Defective Pixels.

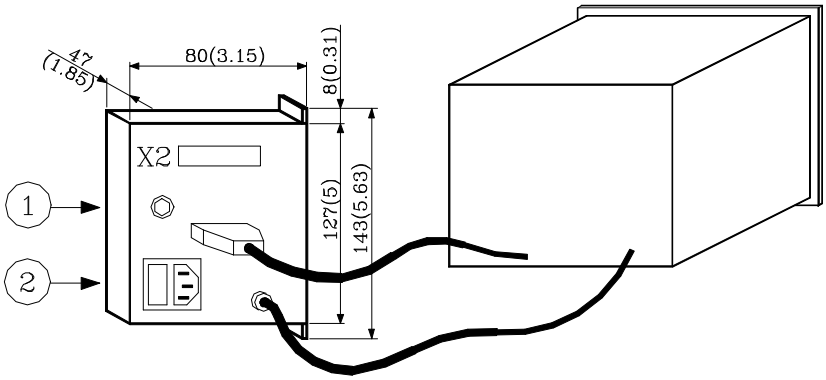
Due to the current status of the Color TFT LCD technology, all manufacturers consider good LCDs those having a certain number of defective pixels. The widely accepted criteria are basically: the number of defective pixels or sub-pixels and their concentration on the LCD surface.

1.4.5 14" COLOR MONITOR

Dimensions in mm (inches):



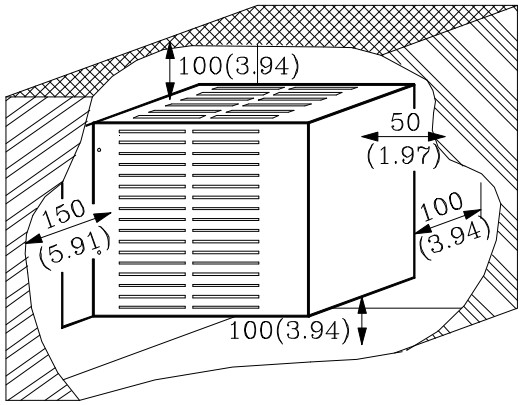
Elements:



1.- Ground terminal. Used to connect the general ground of the machine. It is metric 6mm.
2.- Mains plug for connecting 220V AC and ground.
X2 25-pin SUB-D type male connector to connect the video cable to the Central Unit.

Enclosure:

In order to guarantee proper ambient conditions, the shortest distance, in millimeters, that should be left between each of the Monitor walls and the enclosure in which this is placed, must be as follows:

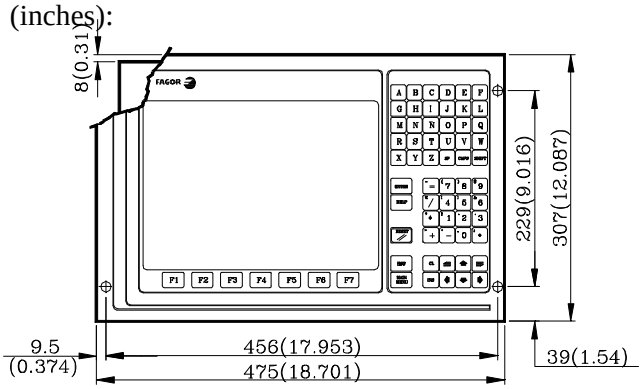


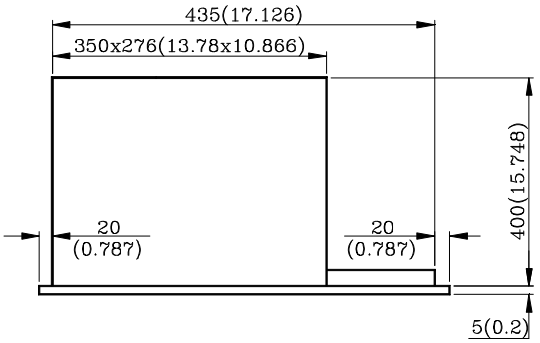
When a fan is used to improve the ventilation of the enclosure a fan with direct current motor should be used, as alternating current (AC) motors product magnetic fields which could distort the images displayed on the screen.

The temperature inside the enclosure should be between 0 and 50°C (32 to 122°F).

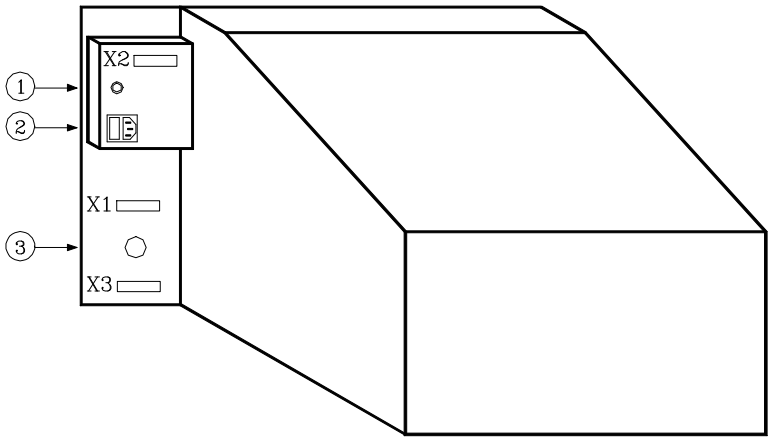
1.4.6 14" COLOR MONITOR WITH ALPHANUMERIC KEYBOARD

Dimensions in mm (inches):





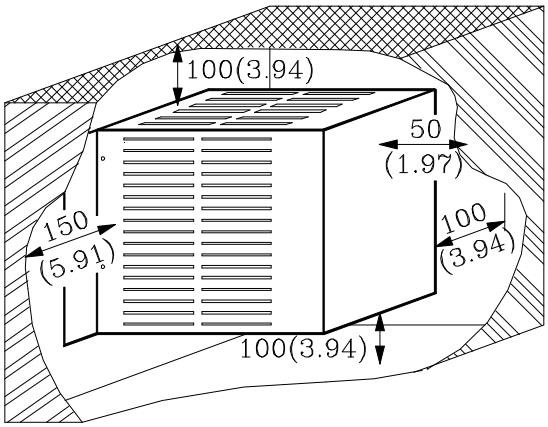
Elements:



1.- Ground terminal. Used to connect the general ground of the machine. It is metric 6mm.
2.- Mains plug for connecting 220V AC and ground.
3.- Buzzer.
X1 25-pin SUB-D type female connector to connect keyboard cable to the Central Unit.
X2 25-pin SUB-D type male connector to connect the video cable to the Central Unit.
X3 Reserved.

Enclosure:

In order to guarantee proper ambient conditions, the shortest distance, in millimeters, that should be left between each of the Monitor walls and the enclosure in which this is placed, must be as follows:

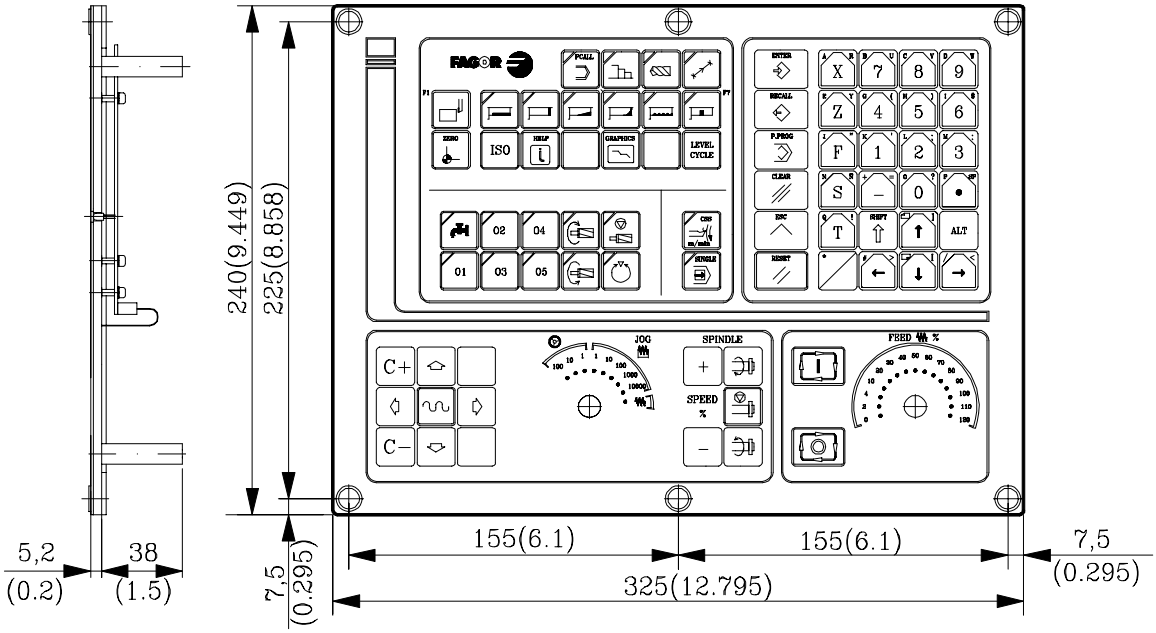


When a fan is used to improve the ventilation of the enclosure a fan with direct current motor should be used, as alternating current (AC) motors product magnetic fields which could distort the images displayed on the screen.

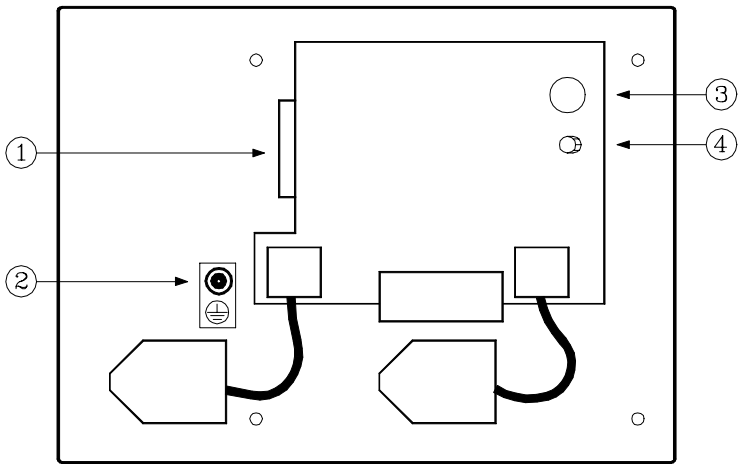
The temperature inside the enclosure should be between 0 and 50°C (32 to 122°F).

1.5 SPECIFIC TC MODEL KEYBOARD

Dimensions in mm (inches):



Elements:



1.- 25-pin SUB-D type female connector to connect the keyboard with the Central Unit or with the keyboard switcher board.

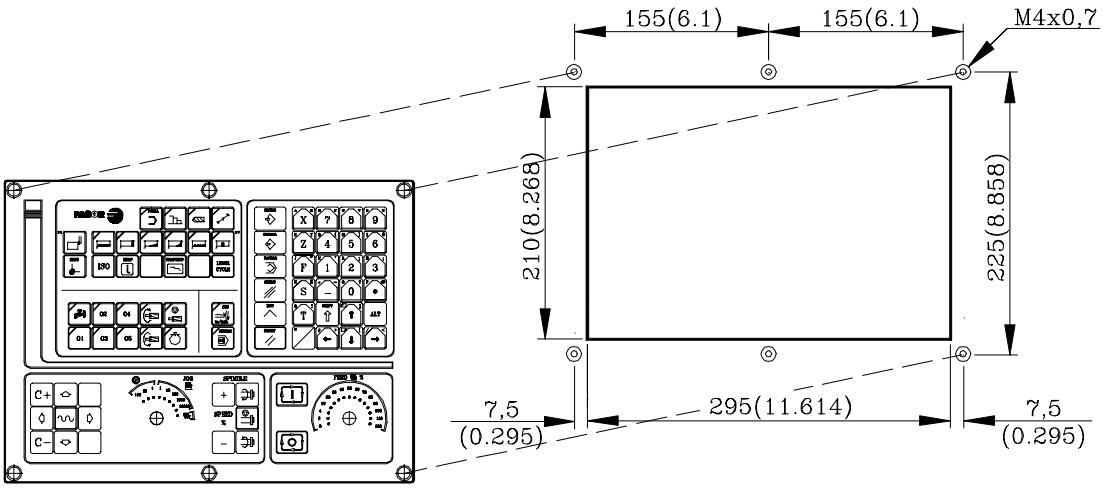
2.- Ground terminal.

3.- Buzzer

4.- Buzzer volume adjusting potentiometer

Enclosure:

The keyboard must be mounted as indicated below:

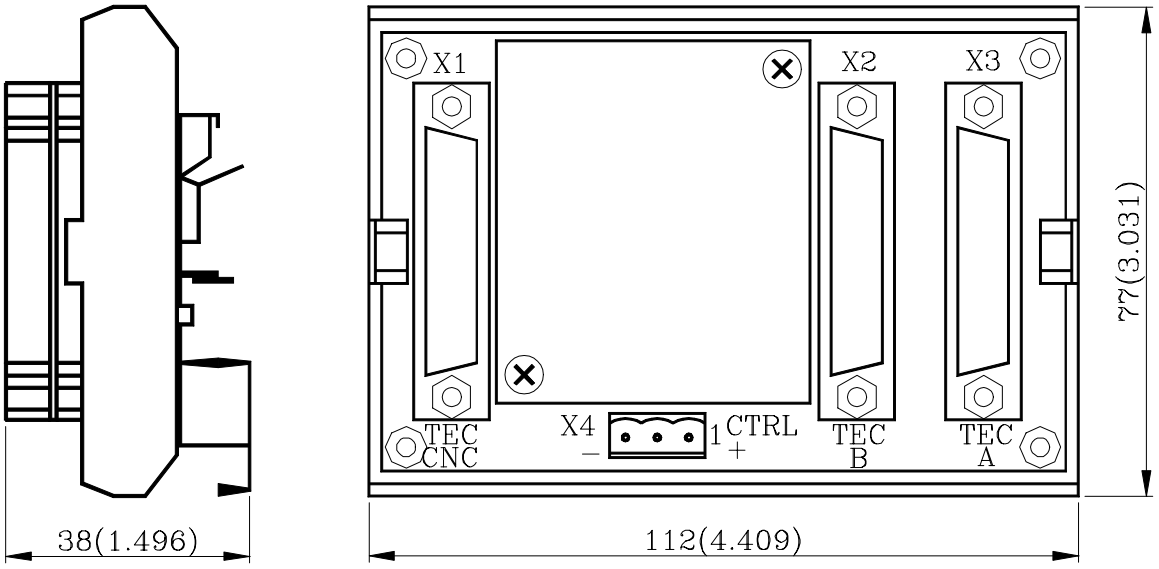


1.6 KEYBOARD SWITCHER BOARD

It must be used when having an TC keyboard and a 14" color monitor with alphanumeric keyboard.

It is used to select the keyboard attended to by the Central Unit: the TC keyboard or the one at the monitor.

Dimensions in mm (inches) and elements:

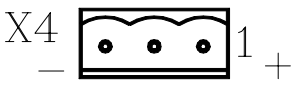


X1 25-pin SUB-D type female connector for connection with the Central Unit.

X2 25-pin SUB-D type female connector for connection with the keyboard of the monitor.

X3 25-pin SUB-D type female connector for connection with the TC keyboard.

X4 3-pin WEIDMÜLLER type male connector used for selecting the keyboard attended to by the Central Unit.



Pin		Value	Function
1	Input	0V	CNC attends to the 8050MC keyboard
		24V	The CNC attends to the 8050M keyboard
2	--	--	Not used at this time
3	Input	0V	External power supply

Connector X4 may be controlled either from the electrical cabinet or by the operator by means of a switch.

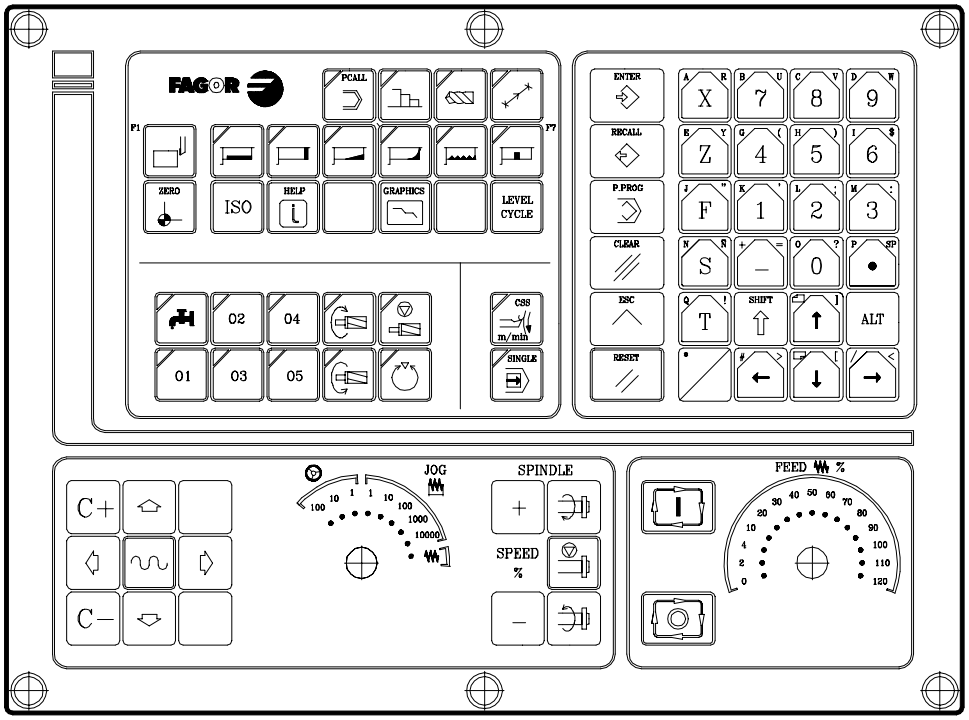
If connector X4 is not under power, the CNC attends to the TC keyboard.

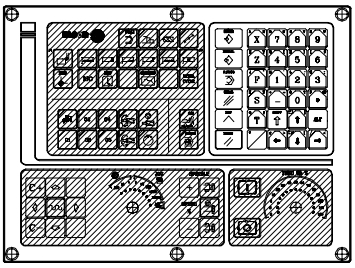
The maximum cable length permissible between the Central Unit and the Keyboard is 25m (82 pies).

The appendix of this manual includes a section with examples about selecting keyboards.

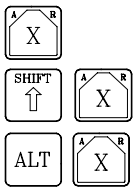
2. GENERAL CONCEPTS

2.1 KEYBOARD





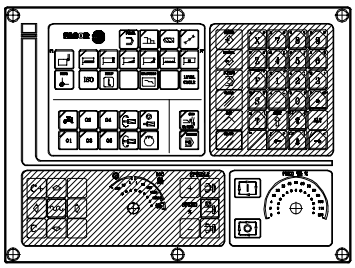
Alphanumeric keys and command keys.



Selects character X

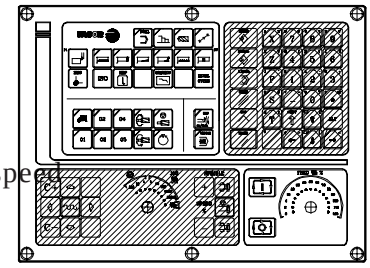
Selects character A

Selects character R



Specific keys for the TC model

Enable Selection and definition of Machining Operations
Governing external devices
Selecting the spindle's operating mode
Selecting single or automatic execution mode



The JOG key

Enables Moving the axes of the machine
Governing the spindle
Modifying the feedrate of the axes and the spindle

Starting and stopping execution

spee

2.2 GENERAL

The 8055TC CNC is based on the 8055T CNC and has inside all the performance features of the 8055T CNC plus the specific features of the TC mode.

For example, the setting of the numerical Control must be done in "T" mode.

In the TC operating mode the programs P900000 to P999999 are reserved for the CNC itself, that is, these cannot be used as part-programs by the user as they have a special significance.

Furthermore, to be able to work in TC mode, the CNC has to have in its memory programs P999997 and P999998, which are supplied by Fagor Automation.

Every time the CNC detects a new software version, updates these programs automatically and makes a backup copy of the old ones in the "Memkey Card" (CARD A).

Also routines 0000 a 8999 are free for use and routines 9000 to 9999 are reserved for the CNC itself.

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

Warning: Programs P999997 and P999998 are associated with the software version.

Fagor Automation shall not be held responsible of any possible malfunction if programs P999997 and P999998 have been deleted from memory or do not correspond to the software version.

Some of the routines reserved for the CNC itself have the following meaning:

9998 Routine to be executed by the CNC at the beginning of each part-program.
9999 Routine to be executed by the CNC at the end of each part-program.

Every time a new part-program is edited the CNC adds a call to the corresponding routine at the beginning and end of each program.

Warning

Both subroutines must be defined by the machine manufacturer even if no operation is to be carried out at the beginning or at the end of the part-program.

Otherwise, the CNC will issue an error when attempting to run a part-program.

Example of how to define subroutine 9998.

```
(SUB 9998) ; Definition of subroutine 9998.
           ; Programmed blocks defined by the machine manufacturer
(RET)      ; End of subroutine
```

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

P999998 This is a routines program used by the CNC for interpreting the programs edited in TC format and executing these afterwards.

Warning

No modifications of this program are allowed. If this program is modified, Fagor Automation will not be held responsible for the performance of the CNC.

If the manufacturer needs to create his own subroutines (home search subroutine, tool change, etc. ...) as well as subroutines 9998 and 9999 should be included in another program, for example P999999.

P999997 This is a text program which contains:

All the phrases and texts displayed on the different screens in the TC mode.
The help texts for the icons in work cycles shown at the bottom left side of the screen.
The messages (MSG) and errors (ERR) to be issued at the TC model.

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

Points to consider:

All the lines of the program have to start with the character ";"
If a line starts with ";;", the CNC will understand that the whole line is a program comment.
The format of a line is as follows:

";Nr. of text - explanatory remark (not displayed) - $Text to be displayed"

Examples

;; General text ..... The CNC treats this as a remark
;;44 Feedrate $M/MIN ..... The CNC treats this as a remark.
;44 $M/MIN ..... This is message 44 and the text "M/MIN" is displayed
;;44 Feedrate $M/MIN ..... This is message 44, and has the explanatory remark "Feedrate" which is not displayed and the text "M/MIN" is shown.

Notes regarding messages:
The format must be respected. Only the text after "SAVEMSG:" may be translated

Example:
Original: N9500(MSG"SAVEMSG: TURNING CYLE")
Translated: N9500(MSG"SAVEMSG: ZILINDRAKETA ZIKLOA")

Notes regarding errors:
The format must be respected. Only the text between quotes("xxxx") may be translated

Example:
Original: N9000(ERROR"Cycle with neither roughing nor finishing operation")
Translated: N9000(ERROR"Arbastatu eta Akabatu gabeko zikloa")

Warning

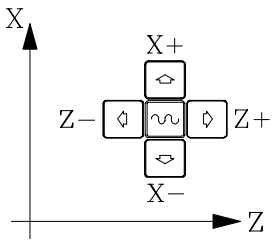
When modifying program 999997, it is recommended to make a backup copy because the CNC replaces it every time another language is selected or the software version is updated.

P998000 ... P998999 These are user-defined profile part-programs. In TC mode the user defines them with 3 digits, (from 0 to 999) and the CNC saves these internally as P 998xxx.

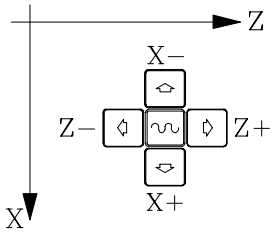
2.2.1 SYSTEM OF COORDINATES

The general machine parameter GRAPHICS (P16) shows the system of coordinates available to the machine and associates the JOG keys with said movements.

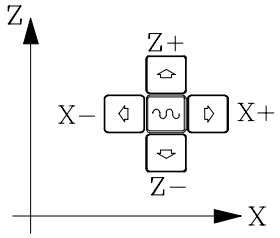
GRAPHICS=0



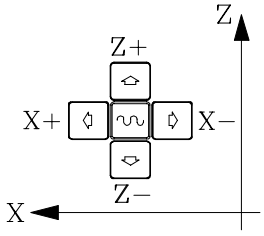
GRAPHICS=1



GRAPHICS=2



GRAPHICS=3



2.2.2 GENERAL LOGIC CNC INPUTS

TOOLINSP (M5050)

If during the execution of an operation or a part,  is pressed, the execution is interrupted and a tool inspection or tool change can be carried out.

General logic input TOOLINSP (M5050) indicates to the CNC when tool inspection is available.

TOOLINSP (M5050) = 0 Tool inspection mode is available when interrupting execution by pressing 

TOOLINSP (M5050) = 1 To access the tool inspection mode, interrupt the execution and then, press 

2.2.3 GENERAL LOGIC OUTPUTS OF THE CNC

The general logic output CUSTOM (M5512) shows the CNC the operation mode that is selected:

CUSTOM (M5512) = 0 "T" Operating mode is selected.
CUSTOM (M5512) = 1 "TC" Operating mode is selected.

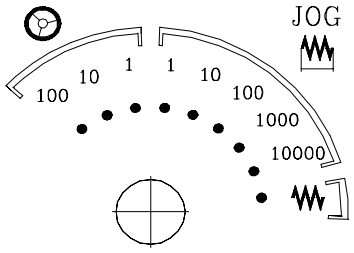
When having two keyboards, MC keyboard and an 14" monitor with keyboard, this variable can be used in the PLC:
- to govern the keyboard switcher board.
- to know the source of the keys and inhibit the desired ones.

SELECT0 a SELECT7 (M5524 a M5531)
SELECTOR (R564)

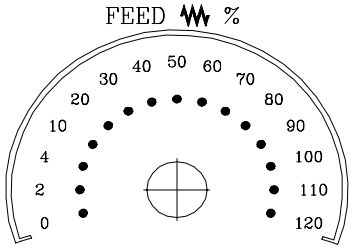
The general logic outputs "SELECT" indicate the position selected at each multi-position switch of the keyboard.
SELECTOR indicates the selected position and SELECT the value being applied by the CNC.

They are usually the same, except when a position is selected which has been inhibited by the input KEYDIS4 (R503).

If while positions 60% through 120% are inhibited, the 100% position is selected, SELECTOR will show the selected position (100%) and SELECT the value being applied (50%).



Selected position / Applied value	SELECTOR(3) SELECT3	SELECTOR(2) SELECT2	SELECTOR(1) SELECT1	SELECTOR(0) SELECT0
Handwheel x 100	0	0	0	0
Handwheel x 10	0	0	0	1
Handwheel x 1	0	0	1	0
JOG 10000	0	0	1	1
JOG 1000	0	1	0	0
JOG 100	0	1	0	1
JOG 10	0	1	1	0
JOG 1	0	1	1	1
Continuous JOG	1	0	0	0



Selected Position / Applied value	SELECTOR(7) SELECT7	SELECTOR(6) SELECT6	SELECTOR(5) SELECT5	SELECTOR(4) SELECT4
Feed Override 0%	0	0	0	0
Feed Override 2%	0	0	0	1
Feed Override 4%	0	0	1	0
Feed Override 10%	0	0	1	1
Feed Override 20%	0	1	0	0
Feed Override 30%	0	1	0	1
Feed Override 40%	0	1	1	0
Feed Override 50%	0	1	1	1
Feed Override 60%	1	0	0	0
Feed Override 70%	1	0	0	1
Feed Override 80%	1	0	1	0
Feed Override 90%	1	0	1	1
Feed Override 100%	1	1	0	0
Feed Override 110%	1	1	0	1
Feed Override 120%	1	1	1	0

2.3 POWER-UP

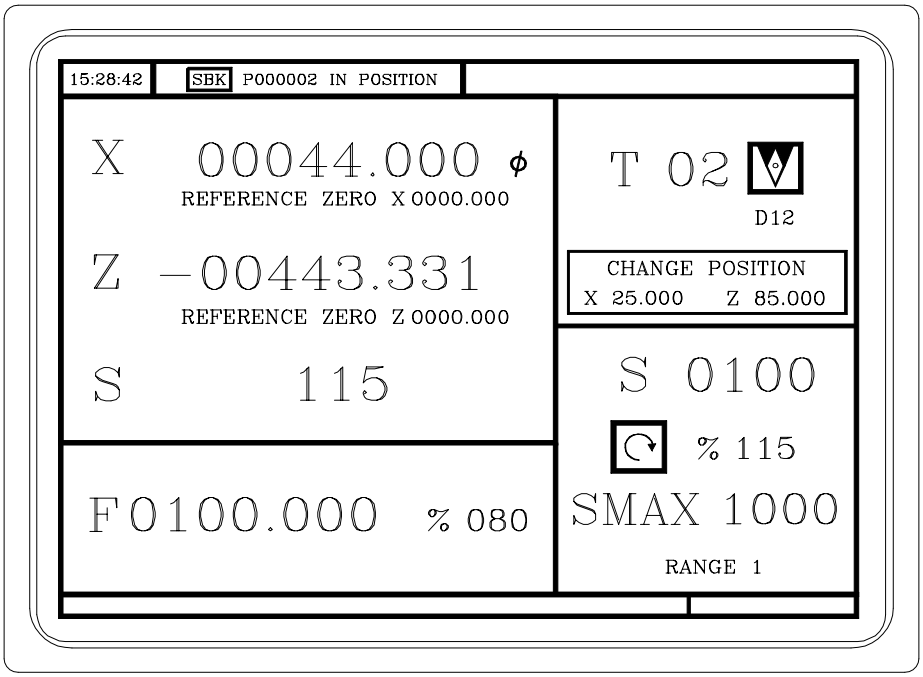
Both on CNC power-up and after the keystroke sequence: RESET SHIFT the CNC acts as follows:

Shows «page 0» if it has been defined by the manufacturer. To access this operating mode, press any key.

If there is no «page 0», the CNC will display the standard screen for the selected work mode.

There are two operating modes: TC mode and T mode. To switch from one mode to the other, press SHIFT ESC

The standard TC mode screen is:



Warning

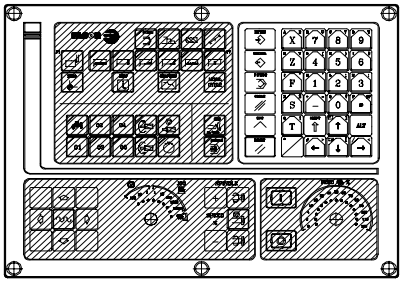
CNC setting should be done in T mode.

Some errors should also be removed in T mode.

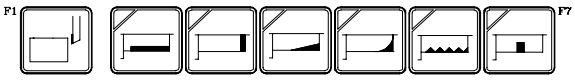
2.4 OPERATING IN 8055T MODE WITH TC KEYBOARD

The TC keyboard has been designed to also operate in T mode. The alphanumeric keyboard must be used for the keys replacing softkeys F1 to F7.

Alphanumeric keyboard:

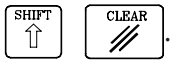


The keys which replace softkeys F1 to F7 are:



To toggle from operating mode to the other press key sequence 

2.5 VIDEO OFF

The CRT can be blanked out by hitting the keystroke sequence: 

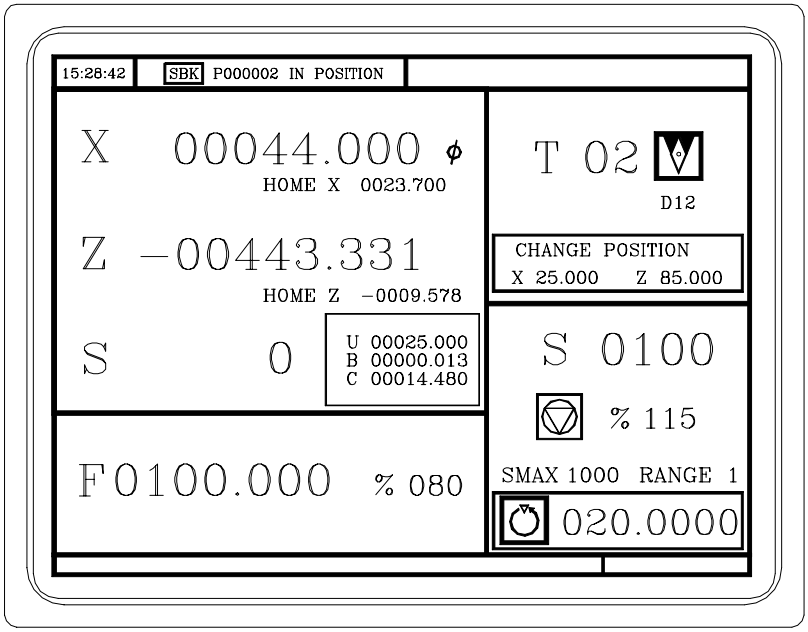
.

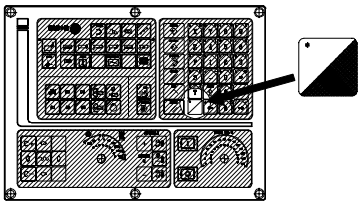
To recover the video signal, just press any key.

On the other hand, when receiving any message (PLC, program, etc.) the CNC also recovers the display.

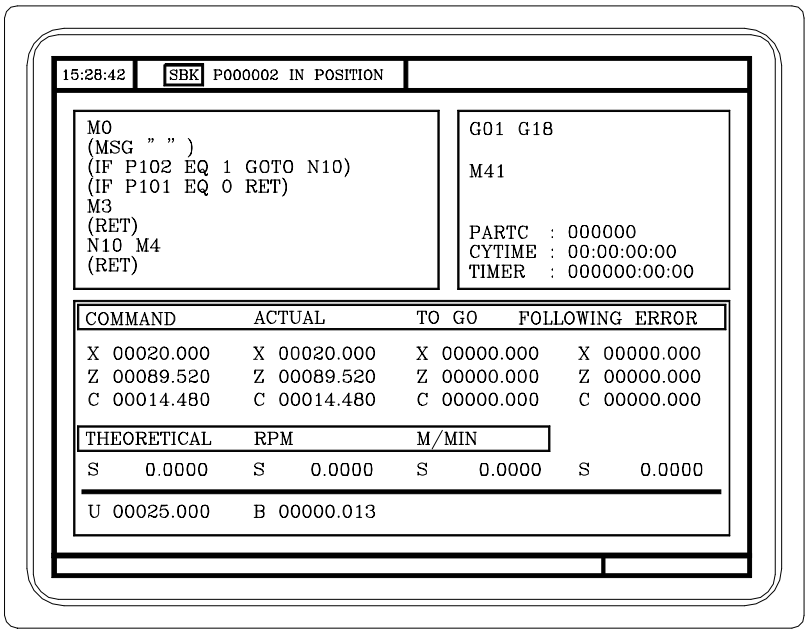
3. OPERATING IN JOG MODE

The standard TC operating mode screen is:



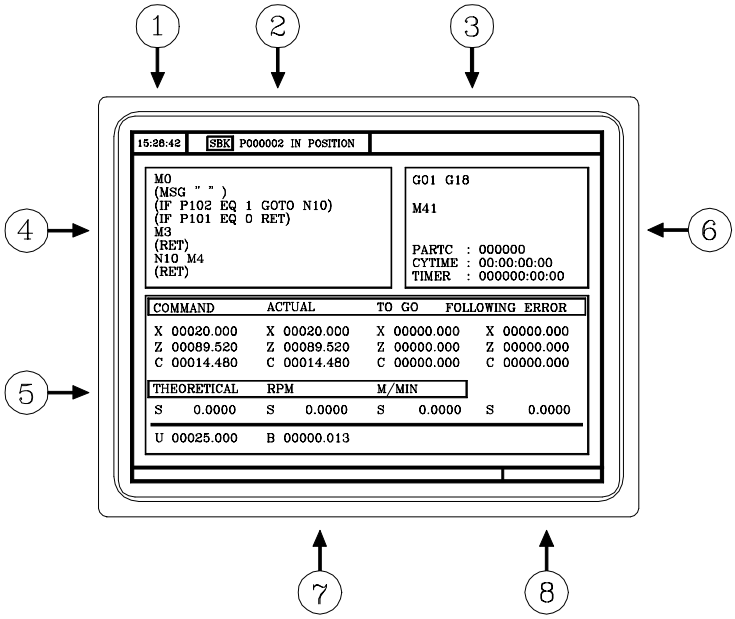
If one presses key 

The CNC displays the special TC operating mode screen.



3.1 INTRODUCTION

The standard TC operating mode screen contains the following information:



1.- Clock

2.- This window can display the following data:

SBK	when the Single Block execution mode is selected.
DNC	when the DNC mode is activated.
P.....	number of the program selected.
Message	«In Position» - «Execution» - «Interrupted» - «RESET»
PLC messages	

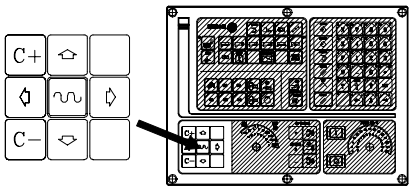
3.- The CNC messages are shown in this window.

4.- This window can display the following data:

* The X, Z coordinates of the axes. The ϕ symbol indicates that the axis is working in diameters.

* In small characters, the axis coordinates referred to machine zero reference (home). This values are very useful when allowing the operator to set a tool change position (see zone 6). The CNC does not show this data when text 33 has not defined in program 999997.

* The coordinates of the auxiliary axes which are defined. The "C" axis is displayed only when it is enabled (G15) and it can be governed manually with the [C+] and [C-] keys.

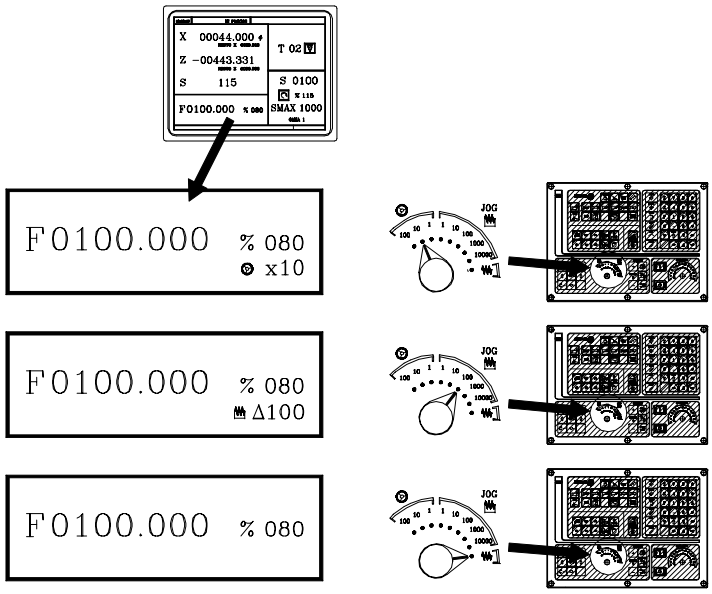


* The real spindle rpm "S".

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

In all cases the feedrate of the «F» axes that has been selected and the % of F which is being applied are shown.

All the possible cases are shown next.



6.- This window displays, in large characters, the tool number «T» selected.

The graphic representation corresponding to the location code associated with the tool.

The offset number «D» associated with the tool. If the tool number and the offset number coincide, the CNC will not display value «D».

The coordinates for the tool change point referred to home. The CNC does not display this window when text 47 of program 999997 is not defined.

7.- This window shows all the details of the spindle :

* The theoretical turning speed selected. «S» value when working in rpm. and «CSS» value when working at constant surface speed.
* The condition of the spindle. This is represented by an icon and can be turning to the right, to the left or idle.
* The % of the spindle speed being applied.
* The maximum revolutions of the spindle.
* The range of spindle active.
* The range of the active spindle. The CNC does not display this information when text 28 of program 999997 is not defined.

8.- Angular increment when working in spindle orientation mode.

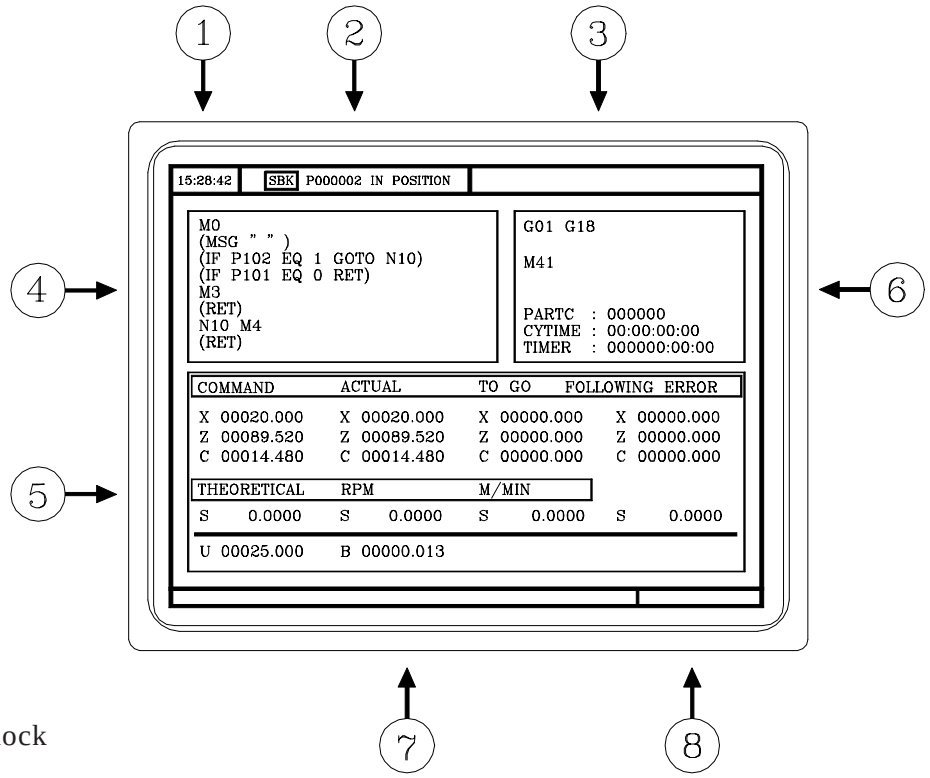
9.- Whenever a work cycle is accessed, the CNC shows the help text associated with the icon selected in this window.

This help text must be defined in P999997 program and be written in the language required.

The format and the points to be considered in the P999997 program are detailed in Chapter 2.

10.-Reserved.

The special screen for TC operating mode contains the following information:



1.- Clock

2.- This window can display the following data:

SBK	when the Single Block mode of execution is selected.
DNC	when the DNC mode is active.
P.....	number of the program selected.
Message	«In Position» - «Execution» - «Interrupted» - «RESET»
PLC messages	

3.- The CNC messages are shown in this window.

4.- In manual operating mode this window does not display any data, but during execution, it shows the lines of the program being executed.

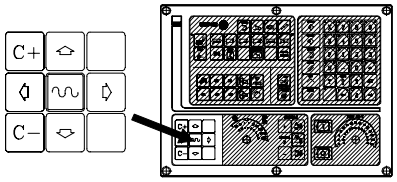
5.- The X, Z and C axes have the following fields:

COMMAND	States the coordinate programmed, that is, the position that the axis must reach.
ACTUAL	States the actual coordinate or actual position of the axis.
TO GO	States the distance that the axis has still to go to reach the coordinate programmed.
FOLLOWING ERROR	Difference between the theoretical and real values of the position.

The spindle (S) has the following fields available:

THEORETICAL	theoretical speed S programmed.
RPM	speed in rpm.
M/MIN	speed in meters/ minute.
FOLLOWING ERROR	When operating with spindle guided stop (M19) this indicates the difference between theoretical and real speeds.

The auxiliary axes only show the actual position of the axis
The "C" axis is displayed only when it is enabled (G15) and it can be governed manually with the [C+] and [C-] keys.



6.- This window shows the state of the «G» functions and the auxiliary functions «M» that are activated. It also displays the value of variables.

PARTC States the number of consecutive parts that have been executed with the same program.

Whenever a new program is selected, this variable assumes value 0.

CYTIME States the time elapsed during the execution of the parts. It is expressed in the following format: “hours : minutes : seconds : hundredths of second”.

Whenever the execution of a program is started, even though this is repetitive, this variable assumes value 0.

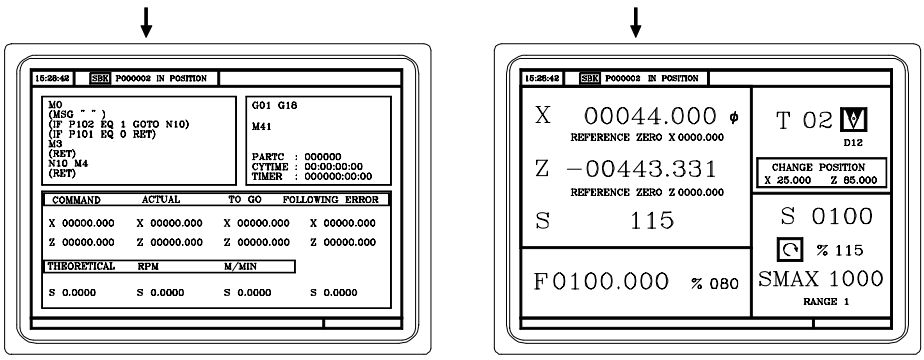
TIMER States the reading of the clock enabled by the PLC. It is expressed in format “hours : minutes : seconds”.

7.- Reserved.

8.- Reserved.

Warning

Whenever a part-program or an operation stored as part of a part-program is selected for simulation or execution, the CNC selects this part-program in the top center window and highlights it next to the symbol.



When the selected program is highlighted, the CNC acts as follows:

If is pressed, the CNC executes the selected part-program.

If CLEAR is pressed the program is de-selected, the CNC deletes it from the top center window.

3.2 AXIS CONTROL

3.2.1 WORK UNITS

Whenever the TC work mode is accessed, the CNC assumes the work units, «mm or inches», «radii or diameters», «millimeters/minute or millimeters/revolution», etc., that are selected by machine parameter.

To modify these values, the "T" mode has to be accessed, modifying the relevant machine parameter.

3.2.2 COORDINATE PRESET

Coordinate preset must be made axis to axis, in the following stages:

1st Press the key for the axis required 

or 

The CNC will frame the position for said axis, to indicate that this is selected.

2nd Enter the value required for preset of the axis.
To exit coordinate preset press 

3rd Press 

so that the CNC assumes said value as the new value for the point.

The CNC requests confirmation of the command. Press 

to confirm or 

to exit preset.

3.2.3 HANDLING THE FEEDRATE OF THE AXES (F)

To fix any particular value for the axis feedrate the following steps have to be carried out:

1st Press 

The CNC will frame the present value, to indicate that this is selected.

2nd Enter the new feedrate required.
To exit coordinate preset press 

3rd Press  for the CNC to assume said value as the new feedrate for the axes.

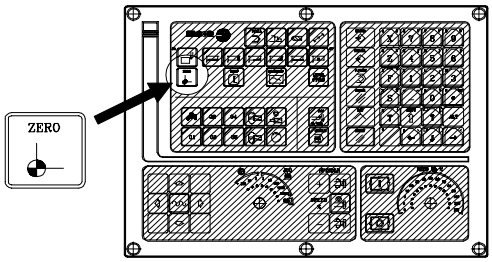
3.3 SEARCH FOR MACHINE REFERENCE ZERO (HOME)

The search for machine reference zero can be done in 2 ways:

- search for machine reference zero for all the axes.
- search for machine reference zero for only one axis.

Search for machine reference for all the axes

To carry out a search for machine reference zero for all axes the user should press key:



The CNC will request confirmation of the command (text 48 of program 999997)

Press  ,The CNC will execute the machine reference zero routine defined by the manufacture in the general machine parameter P34 (REFPSUB).

Warning:

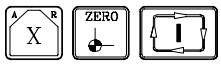
After carrying out the search for machine reference zero (home) position in this mode, **the CNC saves** the part zero or zero offset that is active at the time.

A home search routine, general machine parameter P34 other than 0 has to be defined. Otherwise the CNC will display the relevant error.

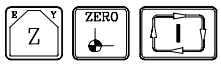
Search for machine reference zero for only one axis

To carry out the search for machine reference zero for only one axis the key for the required axis should be pressed as well as the key for machine reference zero search.

In either case, the CNC will request confirmation of the command (text 48 of program 999997)



Carries out the home search on the X axis



Carries out the home search on the Z axis

Warning:

After carrying out the search for machine home position in this mode **the CNC does not save** the part zero or zero offset that is active at the time and assumes as new part zero the position taken by machine reference zero (home).

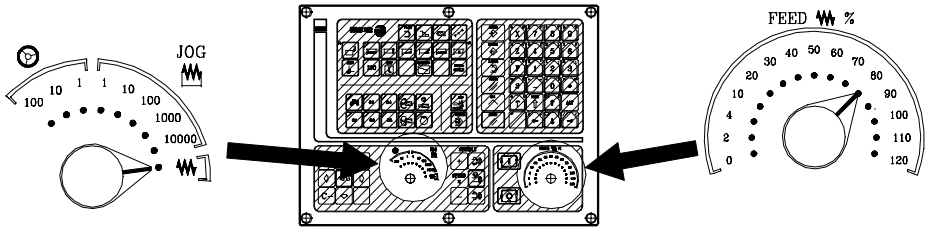
3.4 MANUALLY MOVING THE MACHINE

The axes of the machine can be moved in the following ways:

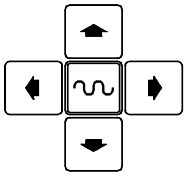
- [X] [target position] [I] or [Z] [target position] [I]
- continuous movement
- incremental movement
- movement by electronic handwheel

3.4.1 CONTINUOUS MOVEMENT

Place the left-hand switch in position ⩘ and on the right-hand switch select the percentage (0% to 120%) of the feedrate selected to be applied.



Continuous movement should be done axis to axis. To do this press the JOG key for the direction of the axis to be moved.



The axis moves with a feedrate equal to the percentage (0% to 120%) of the «F» feedrate selected.

If during movement the key [~] is pressed the maximum feedrate possible is carried out, as is stated in the “G00FEED” axis machine parameter. This feedrate will be applied as long as said key is pressed, and when released the previous feedrate will be resumed.

Depending on the state of the “LATCHM” general logic input the movement will be made in the following way:

* If the PLC sets this mark at a low logic level (0V), the axis will only move while the relevant JOG key is pressed.

* If the PLC sets this mark at a high logic level (24V), the axis will start to move when the JOG key is pressed and will not stop until said JOG key or another JOG key is pressed again, and in this case the movement is transferred to what is indicated by the next key pressed.

When operating with feedrate "F" in millimeters/revolution the following cases may arise:

a) The spindle is started. [↻] or [↺]

The CNC moves the axes to the F programmed.

S 0500

b) The spindle is stopped but there is a spindle speed S selected. [▽] % 115

The CNC calculates the feedrate in millimeters/minute corresponding to the theoretical "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 in millimeters/minute.

S 0000

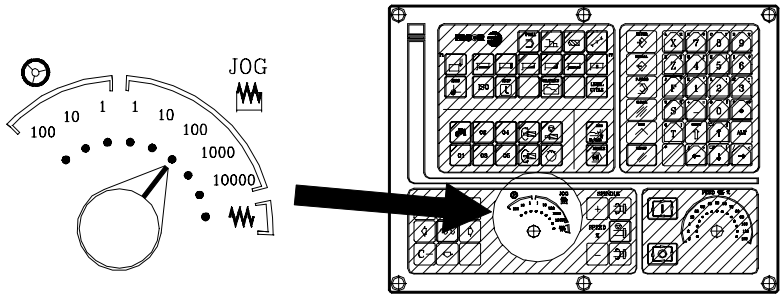
c) The spindle is stationary and there is no spindle speed S selected. [▽] % 115

If feedrate F has value 0, the CNC moves the axes at rapid feedrate.

If feedrate F has any other value, the axes will only be able to be moved if key [~] is pressed and the key for one axis. The CNC moves the axis at fast feedrate.

3.4.2 INCREMENTAL MOVEMENT

Place the left-hand switch in one of the positions JOG



Incremental movement must be done axis to axis. To do this press the JOG key for the direction of the axis to be moved.

Each time a key is pressed, the corresponding axis moves the amount set by the switch. This movement effects the «F» feedrate selected.

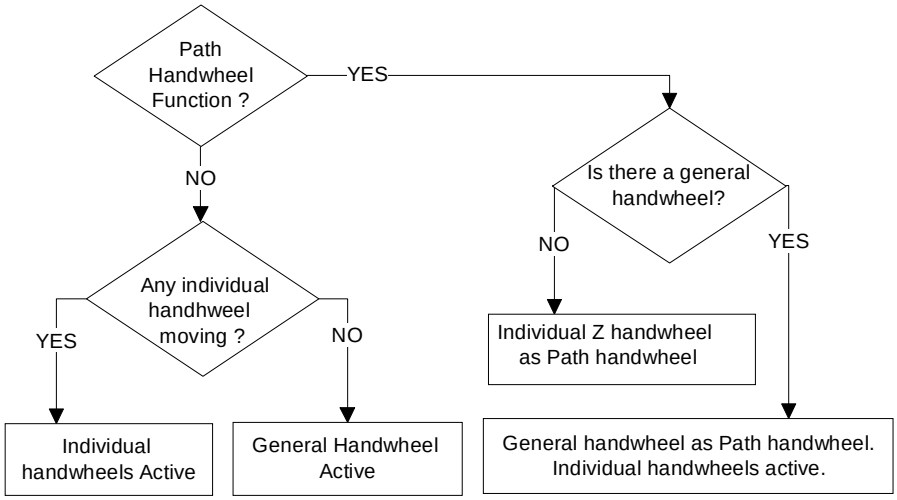
Position of the switch	Movement per turn		
1	0.001 mm	or	0.0001 inches
10	0.010 mm	or	0.0010 inches
100	0.100 mm	or	0.0100 inches
1000	1.000 mm	or	0.1000 inches
10000	10.000 mm	or	1.0000 inches

3.4.3 MOVEMENT BY MEANS OF ELECTRONIC HANDWHEEL

The various handwheel configurations are:

General handwheel	Is the typical handwheel. It can be used to jog any axis one by one. Select the axis and turn the handwheel to move it.
Individual handwheel:	It replaces the mechanical handwheels. Up to 3 handwheels can be used (one per axis). It only moves the axis it is associated with.
Path handwheel:	For chamfering and rounding corners. 2 axes are moved along a selected path (chamfer or rounding) by moving a single handwheel. This feature must be activated via PLC. The general handwheel is assumed as the "path handwheel" or the individual handwheel associated with the Z axis
Feed handwheel	To control the feedrate of the machine. This feature must be activated via PLC.

When using several handwheel types, the CNC sets the following priorities:



Operation when the "path handwheel" function is not active.

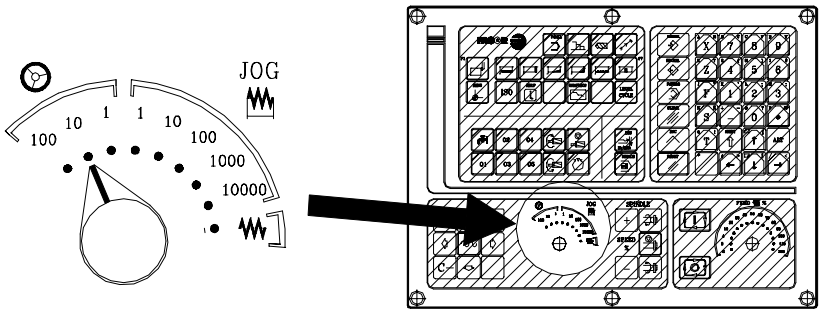
Any Individual handwheel has priority.
To jog with the general handwheel, select the axis and turn the handwheel.

Operation when the path handwheel function is active.

If there is no General Handwheel, the CNC assumes as path handwheel the individual handwheel associated with the Z axis.

If there is General handwheel, the CNC assumes it as path handwheel. The individual handwheels keep working.

To move any of them, turn the switch to any of the handwheel positions.



Positions 1, 10 and 100 indicate the multiplying factor being applied besides the internal x4 to the feedback pulses supplied by the electronic handwheel.

For example, when having a 25 lines/turn handwheel with a display format of 5.3 mm or 4.4 inches and the machine parameter "MPGRES=0":

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

To apply a different multiplying factor for each handwheel, the HANFCT variable must be used. Refer to the section on variables associated with electronic handwheels in chapter 10 of this manual.

Warning:

Depending on how fast the handwheel is turned and on the selected handwheel switch position, the CNC might be demanded to move the axis faster than the limit set by general machine parameter "G00FEED". In that case, the CNC will move the axis the indicated distance but it will limit the axis speed to that parameter value.value.

3.4.3.1 GENERAL HANDWHEEL

Select the axis to be jogged

Press one of the JOG keys of the axis to be jogged. The selected axis will be highlighted.

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.

Jog the axis

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

3.4.3.2 INDIVIDUAL HANDWHEEL

Jog the axis

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

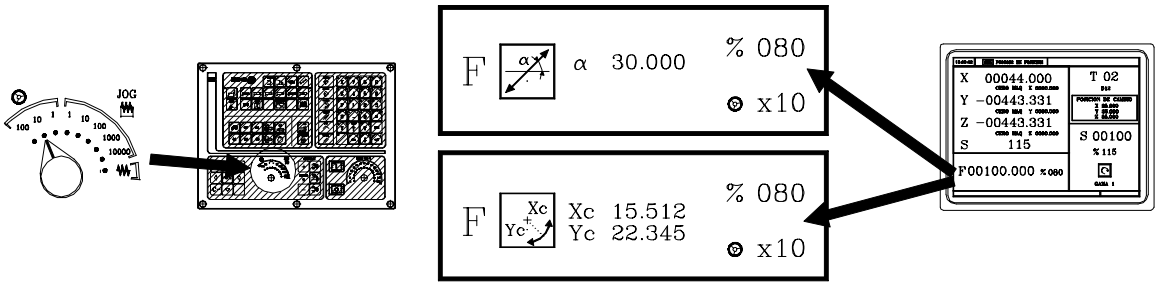
3.4.3.3 PATH HANDWHEEL

With this feature, it is possible to jog two axes at the same time along a linear path (chamfer) or circular path (rounding) with a single handwheel.

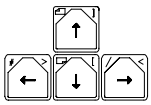
The CNC assumes as the path handwheel the general handwheel or, when this one is missing, the one associated with the Z axis.

This feature must be handled by the manufacturer of the machine.

While in handwheel mode and selecting the “path handwheel”, the CNC shows the following data:



When choosing a linear jog (upper drawing) , the angle of the path must be indicated and when choosing a circular jog (lower drawing), the arc center coordinates must be indicated in radius.

To define these variables, press the [F] and, then, one of these keys: 

Example: The [O2] key is used to activate or deactivate the "path handwheel" mode and the [O3] key to indicate the type of movement.

```
DFU B29 R561 = CPL M5054   Activate / deactivate the "path handwheel" mode
DFU B31 R561 = CPL M5053   Selects the type of movement, along a straight  line or an arc
```

3.4.3.4 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.

To do this, proceed as follows:

Inhibit all the feedrate override switch positions from the PLC.
Detect how far the handwheel is turned (reading of pulses received)
Set the corresponding feedrate from the PLC depending on the pulses received from the handwheel.

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

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

To use this feature, the handwheel must be associated with one of the axes of the machine. General machine parameters “AXIS1....8” or “HANDWHE1....4” set with values: “21....29”

Example: The machine has a button to activate and deactivate this feature (feed handwheel) and the feedrate control is carried out with the second handwheel.

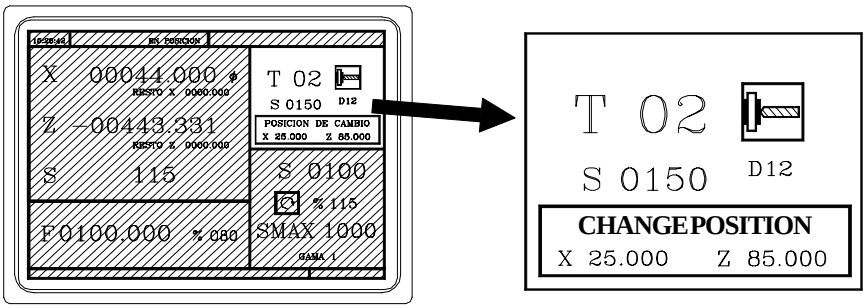
```
CY1
R101=0                                  Resets the register containing the previous handwheel
reading
END

PRG
DFU I71 = CPL M1000                     Every time the button is pressed, mark M1000 is inverted
M1000 = MSG1                            If the feature is active, a message is displayed.
NOT M1000                               If the feature is not active
= AND KEYDIS4 $FF800000 KEYDIS4         enables all the positions of the feedrate override
switch
= JMP L101                              and goes on with program execution
If the feature is active
DFU M2009                               and a leading edge (up flank) occurs at the clock mark
M2009
= CNCRD(HANPS,R100,M1)                  We read the number of handwheel pulses contained in
R100
= SBS R101 R100 R102                    calculates the number of pulses received from the last
reading
= MOV R100 R101                         updates R101 for the next reading
= MLS R102 3 R103                       calculates in R103 the proper % of feedrate override
= OR KEYDIS4 $7FFFFF KEYDIS4            inhibits all the other positions of the feedrate override
switch
CPS R103 LT 0 = SBS 0 R103 R103         ignores the handwheel turning direction
CPS R103 GT 120 = MOV 120 R103          Limits the maximum feedrate override to 120%.
DFU M2009                               With the leading edge (up flank) of the clock mark
M2009
= CNCWR(R103,PLCFRO,M1)                 set the calculated feedrate override (PLCFRO=R103)

L101
END
```

3.5 TOOL CONTROL

The standard screen for TC operating mode displays the following information about the tool.



This window displays the following information:

> In large characters, the number "T" of the selected tool and a graphic representation of its tip.

> The offset number «D» associated with the tool.

> The selected rpm "S" for the live tool .
This value is only displayed when a live tool is selected.

> The coordinates for the tool change point.
The CNC does not display this window when text 47 of program 999997 is not defined.

To select any other tool take the following steps:

1st Press [T]

The CNC will frame the tool number

2nd Enter the tool number to be selected

To exit the selection process press [ESC]

3rd Press [I] key for the CNC to select the new tool.

The CNC will handle the tool change

4th After selecting the new tool, the CNC updates the graphic representation for the location code associated with the new tool.

It is possible to temporarily assign another offset to the tool without modifying the one it has associated.

To access the "D" filed, press [T] and [↓]

Key in the desired tool offset number and press [I]

The CNC temporarily assumes the new offset of the current tool. The internal table is not modified, the tool keeps having the same tool offset associated with it when it was calibrated.

3.5.1 TOOL CHANGE

Depending on the type of tool changer, one can have:

Machine with automatic tool changer
Machine with manual tool changer

In both cases the CNC:

Executes the routine associated with the tool change (general machine P60 «TOOLSUB»).

Sends the PLC all the information required for this to handle the tool change.

And assumes the new values for the tool (offsets, geometry, etc. ...).

An example of how a manual tool changer is handled.

Subroutine 55 as associated with the tools. General machine parameter P60 «TOOLSUB» = 55.

Define the general machine parameter P71 "TAFTERS" = YES so that the tool is selected after executing the subroutine.

The movement to the change point, block N3, is carried out only when an operation of cycle of the TC mode is being executed.
If a cycle is selected ..... CYCEXE other than 0
If the program is in execution ..... OPMODA bit 0 = 1
Therefore (IF ((CYCEXE NE 0) AND (OPMODA AND $1) EQ 1) there is no movement.

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

```
(SUB 55)
      (P100 = NBTOOL)            ; Assigns the No. of tool requested to P100
      (P101 = MS3)               ; If spindle clockwise P101=1
      (P102 = MS4)               ; If spindle counter-clockwise P102=1
      (IF ((CYCEXE NE 0) AND (OPMODA AND $1) EQ 1) GOTO N5
N3 G0 G53 .... XP??? ZP???       ; Movement to change point
N5 M5                            ; Spindle stop
      (MSG "SELECT T?P100 - THEN PRESS START")
                                 ; Message for requesting tool change
      M0                         ; Program stop and wait until START is pressed
      (MSG "" "")                ; Erases previous message
      (IF P102 EQ 1 GOTO N10)    ; Recovers turning direction of spindle
      (IF P101 EQ 0 RET)
      M3
      (RET)
N10 M4
      (RET)
```

After completing the subroutine, the CNC executes function T??, sends the PLC all the information required for the latter to handle the tool change and assumes the new values for the tool, (tool offsets, geometry, etc.)

3.5.1.1 VARIABLE TOOL CHANGE POINT

If the manufacturer wishes the user can be allowed to define the tool change point at all times. This feature logically depends on the type of machine and type of changer.

This feature allows the tool change to be made beside the part, thus avoiding movements to a change point farther away from the same.

To allow this:

Define text 47 of the program 999997 for the CNC to request the coordinates on X and Z of the change point.
For example: ;47 $CHANGE POSITION

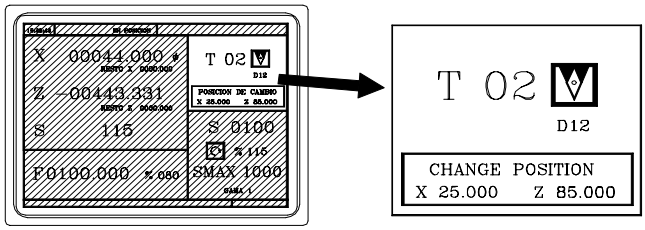
These coordinates should always refer to machine reference zero (home), for the zero offsets not to affect the tool change point.

For this reason, the CNC can display, along with coordinates X, Z and in small characters, the coordinates for the axes referring to home.

For the CNC to show the coordinates of the axes referring to home text 33 of program 999997 has to be defined. For example: ;33 $REFERENCE ZERO (HOME)

Since the tool change point can be modified by the operator at any time, the subroutine associated with the tools must take these values into account.

Arithmetical parameters P290 and P291 contain the values set by the operator as change position on X, Z.



Arithmetical parameter P290
Change position on X
Arithmetical parameter P291
Change position on Z

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

Where it says: G0 G53 XP??? ZP??? ; Movement to the change point.
It should say: G0 G53 XP290 ZP291 ; Movement to the change point defined by the user.

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

Press [T] to select the "T" . Then press key for the relevant axis [X] [Z] or:

[↑] [↓] [←] [→]

After moving over the coordinates for the axis to be defined, one can:

a) Enter the value manually. Key in the value required and press [ENTER]

b) Assign the present position of the machine.

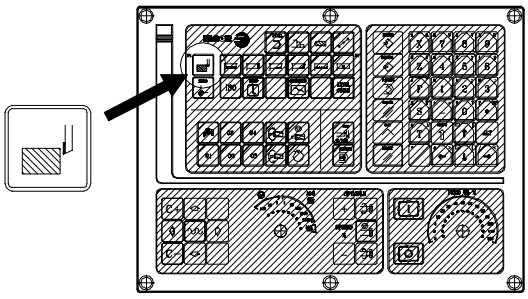
Move the axis, by means of the handwheel or the JOG keys, up to the point required.

Press [RECALL] . The CNC assigns said coordinate to the field selected.

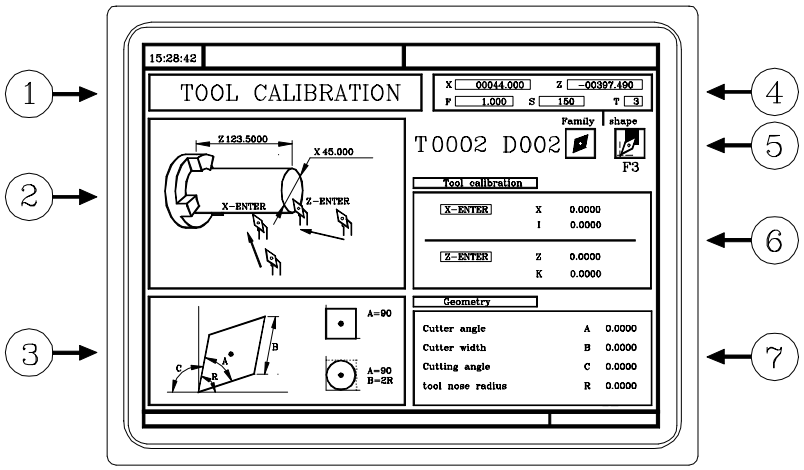
Press [ENTER]

3.5.2 TOOL CALIBRATION

To access tool calibration mode press key



The CNC displays the following information:



1.- Indicator of the operating mode selected: «Tool calibration».

2.- Help graphics for tool calibration.

3.- Help graphics for defining the tool geometry.

4.- Current status of the machine.
Actual X, Z coordinates, actual axis feedrate F, actual spindle speed S and currently selected tool T.

5.- Tool number, offset number, family and location code (shape).

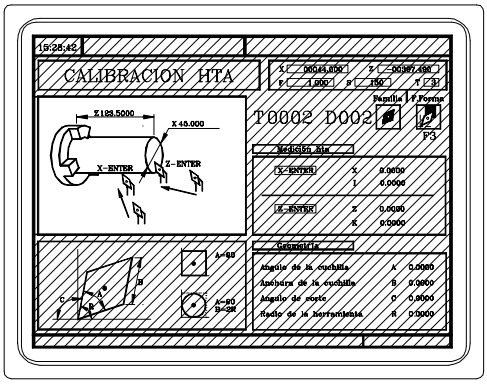
6.- Tool length values defined for this tool.

7.- Tool geometry values.

To calibrate a tool, proceed as follows:

1.- Select a tool and access the tool calibration mode.

2.- Use a part of known dimensions



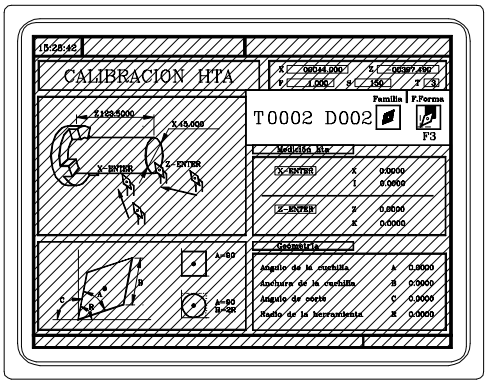
Place that part in in the chuck.

Define the part dimensions:

ENTER (value) X

Z (value) ENTER

3.- Define the tool data.



Define the tool number «T»:

(tool number) ENTER

· If it is defined, the values stored in the tool table will be displayed.
· If it is not defined, all the data will be reset to "0".

Define the tool offset «D»:

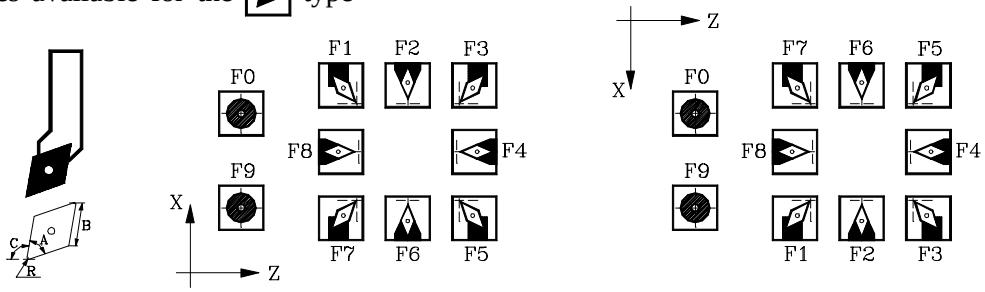
(tool offset number) ENTER

Select the type or family of the tool using the key

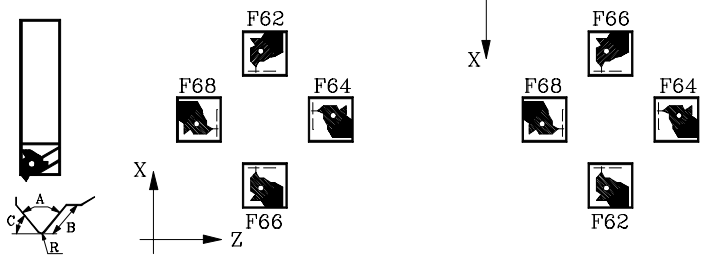
The possible types are:

Define the location code (shape) of the tool using the

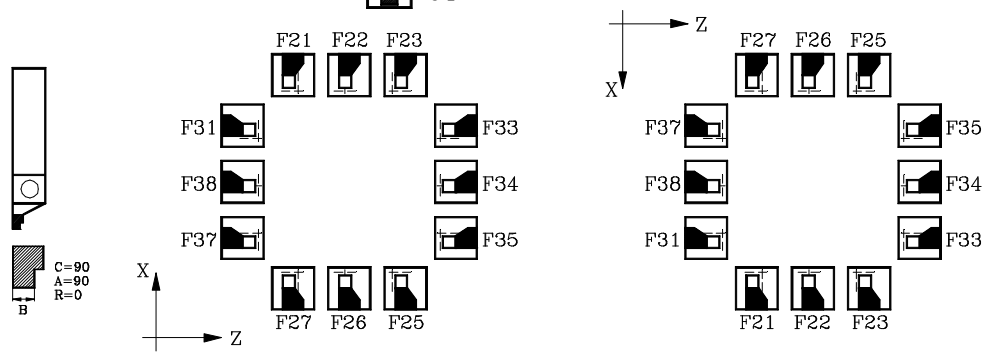
Location codes available for the type



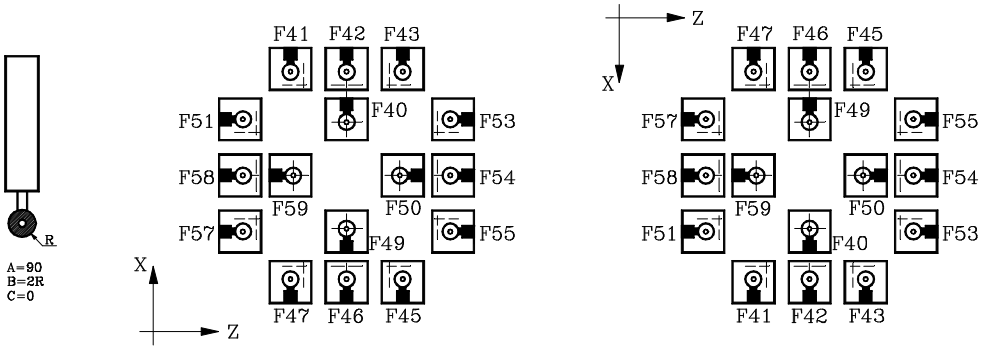
Location codes available for the type



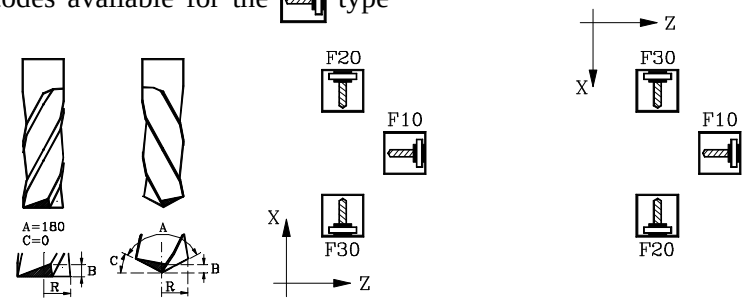
Location codes available for the type



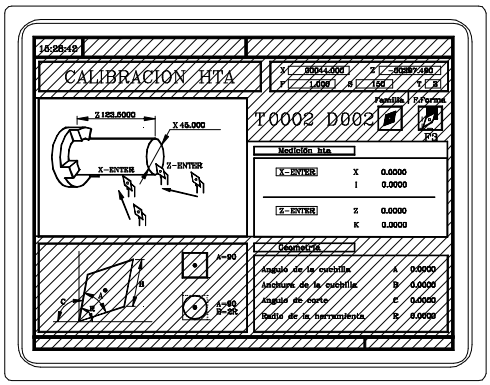
Location codes available for the type



Location codes available for the type



4.- Tool calibration. There are two methods:



When using a tool presetting table, define the X, I, Z, K data.
- The X, Z data indicate the tool dimensions in X and Z.
- The I, K data indicate the tool wear to be compesated for.

When not using a tool presetting table.
- Approach the tool to the part until touching it along the X axis and press ENTER X

- Approach the tool to the part until touching it along the Z axis and press Z ENTER

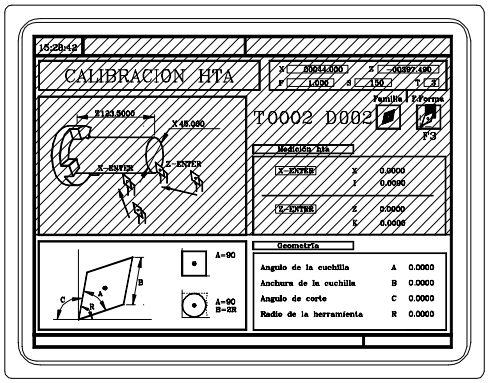
- The tool is now calibrated. The CNC updates the X, Z data and sets the I, K data to "0".

The actual tool length is (X+I) and (Z+K) and the «I» value must always be in diameter.

5.- Define the values for the tool geometry.

The window on the right contains the tool geometry values and the window on the left contains a help diagram.

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



6.- To calibrate another tool, repeat steps 3, 4 and 5.

To quit the tool calibration mode, press ESC.

3.5.2.1 MODIFY VALUES DURING EXECUTION

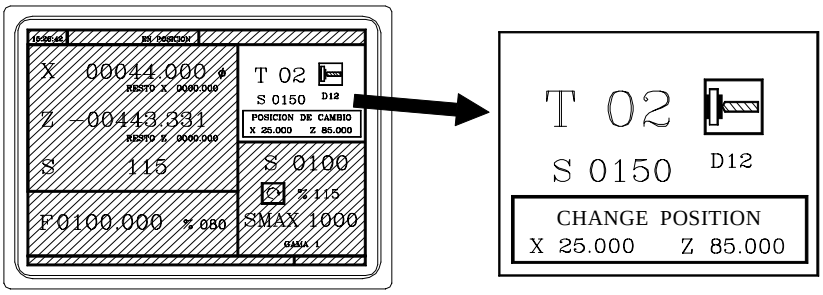
It is possible to modify the tool values (dimensions and geometry) without interrupting program execution.

To do this, press , the CNC will display the Tool Calibration screen with the data for the active tool and it is possible to change its data or that of any other tool.

To quit this screen, press ESC.

3.5.3 LIVE TOOL

When a live tool is selected, the standard screen of the TC operating mode shows the following information:



To select the rpm "S" of the live tool, take these steps:

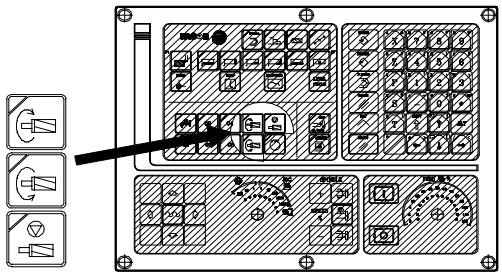
1. Press [T] to select the "T" field.
2. Press [S] or [↓] to select the rpm "S" of the live tool.
3. Key in the value and press [ENTER]

The keys for the live tool are:

Live tool clockwise .....

Live tool counter-clockwise .....

Stop the live tool.....



When the machine uses a live tool, the following considerations must be born in mind:

Set one of the general machine parameters P0 through P9 to "13"
The location code (shape) of the live tool must be "10", "20" or "30".
It is up to the PLC to 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) indicates the status of the key.

Bit 3 of Register 562 (B3 R562) indicates the status of the key.

Bit 5 of the Register 562 (B5 R562) indicates the status of the key

Here is an example of the section of the PLC program in charge of managing the live tool:

( ) = CNCRD (TOOL, R101, M1)
Assigns the number of the active tool to register R101.

= CNCRD (TOF R101, R102, M1)
Assigns the location code (shape) of the active tool to register R102.

CPS R102 EQ 10 OR CPS R102 EQ 20 OR CPS R102 EQ 30 = M2
If the active tool is a live tool, (location code=10, 20 or 30), it activates mark M2.

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

If while the TC operating mode is selected (CUSTOM=1) ...
... 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" to stop the live tool.

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

If in TC mode (CUSTOM=1) a live tool is selected (M2) and the "live tool clockwise" key is pressed, (DFU B3R562) ...
... the PLC reads in R117 the rpm selected for the live tool (LIVRPM) and it assigns them to general parameter P100... It, then, "tells" the CNC to execute block "M45 SP100" (clockwise turning of the live tool at the selected rpm).

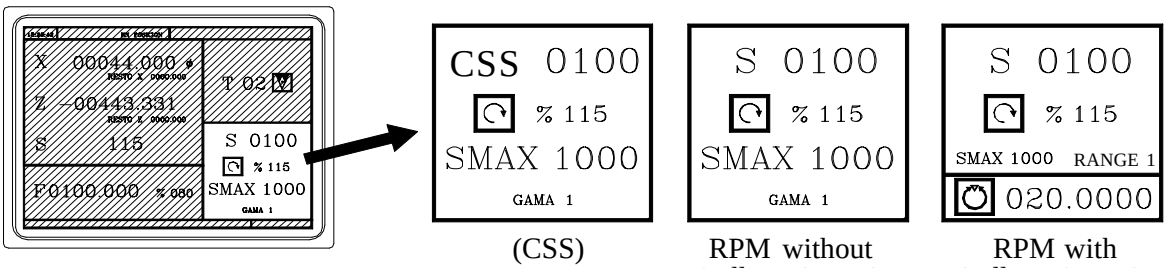
CUSTOM AND M2 AND DFU B5R562 = CNCRD (LIVRPM, R117, M1) = CNCWR (R117, GUP100, M1= CNCEX1 (M45 S-P100, M1)

If in 8055 TC mode (CUSTOM=1) a live tool is selected (M2) and the "live tool counter-clockwise" key is pressed (DFU B5R562) ...
... the PLC "tells" the CNC to execute block M45 S-P100 (counter-clockwise turning of the live tool at the selected rpm).

3.6 SPINDLE CONTROL

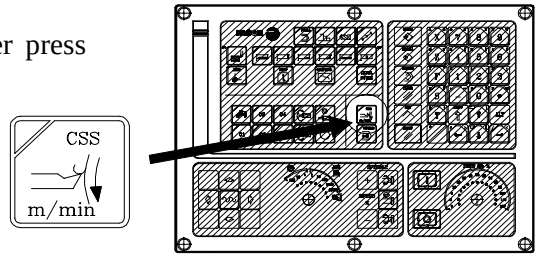
The standard TC working mode screen has a window for displaying information about the spindle.

Since the CNC allows operating with the spindle in RPM, at Constant Surface Speed or in the Spindle Orientation mode, the information being displayed will be different for each mode.



(CSS) RPM without spindle orientation RPM with spindle orientation

To change from one mode to another press

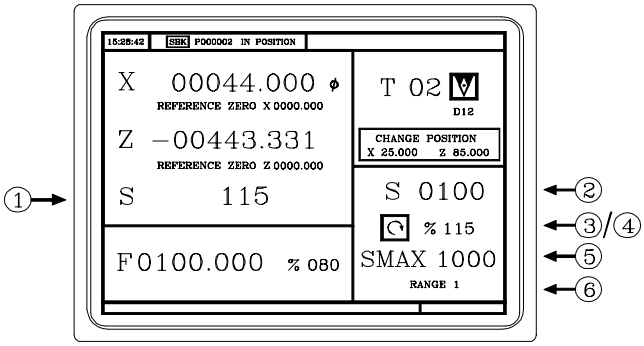


Both on power-up of the CNC and after the key sequence RESET SHIFT the CNC selects the operating mode as Revolutions / minute (RPM)

When working at Constant Surface Speed (CSS), the CSS m/min key is lit up.

3.6.1 SPINDLE IN RPM

The CNC displays the following information:



1.- Actual spindle speed in rpm.

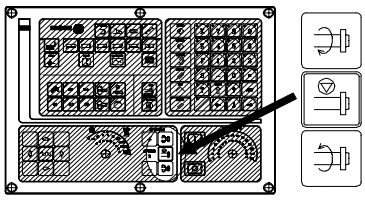
2.- Theoretical spindle speed in rpm.

To select any other speed press [S]. The CNC will frame the present value.

Enter the new value and press [I]. The CNC assumes said value and updates the actual speed of the spindle.

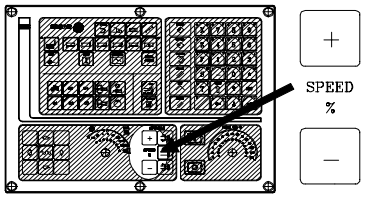
3.- State of the spindle: [↻] turning clockwise, [↺] turning counter-clockwise or [▽] stopped.

To modify the state of the spindle press keys:



4.- % of the theoretical turning speed of the spindle that is being applied.

To modify this percentage (%) press keys:



5.- Maximum spindle speed in rpm.

To select any other speed press [S] twice. The CNC will frame the present value.

Enter the new value and press [ENTER]. The CNC assumes said value and will not let the spindle exceed this number of revolutions.

6.- Spindle range currently selected.

When having an automatic gear changer, this value cannot be modified.

When not having an automatic gear changer, press [S] and then use the [↓] key to frame the current value.

Enter the range number to be selected and press [ENTER] or [I]

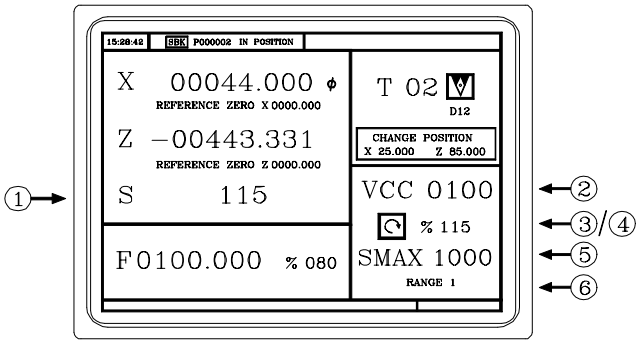
Note: When the machine does not have spindle ranges this message is superfluous. For this reason, when text 28 of program 999997 is not defined, the CNC does not display this message.

3.6.2 CONSTANT SURFACE SPEED

In Constant surface speed mode the user sets the tangential speed that there must be at all times between the tool tip and the part.

The spindle revolutions therefore depend on the position taken by the tool tip with respect to the turning axis. If the tool tip moves away from the turning axis, the spindle revolutions thus go down, and if it gets closer, they go up.

When Constant Surface Speed is selected the CNC displays the following information.



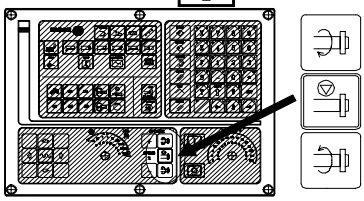
1.- Actual spindle speed in rpm.

2.- Theoretical constant surface speed. Defined in m/minute or in feet/minute.

To select any other speed press [S]. The CNC will frame the present value.

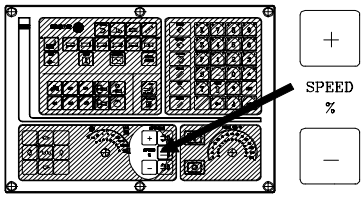
Enter the new value and press [I]. The CNC assumes this value and if the spindle is started it will update the actual speed of the spindle (in rpm.)

3.- State of the spindle: [↻] turning clockwise, [↺] turning counter-clockwise or [▽] stopped.
To modify the state of the spindle press:



4.- % of the theoretical Constant Surface Speed being applied.

To modify this percentage (%) press :



5.- Maximum spindle speed in rpm.

To select any other speed press [S] twice and the CNC will frame the present value.

Enter the new value and press [ENTER]. The CNC will assume this value.

6.- Spindle range currently selected.

When having an automatic gear changer, this value cannot be modified.

When not having an automatic gear changer, press [S] and then use the [↓] key to frame the current value.

Enter the range number to be selected and press [ENTER] or [I]

Note: When the machine does not have spindle ranges this message is superfluous. For this reason, when text 28 of program 999997 is not defined, the CNC does not display this message.

3.6.2.1 OPERATING AT CONSTANT SURFACE SPEED (CSS)

When Constant Surface Speed operating mode is selected (CSS), the CNC assumes the spindle range selected at present.

In this operating mode, when a new constant surface speed is selected, the following cases may arise:

a) The spindle is stopped

The CNC selects the new speed but does not apply this until the spindle moves.

b) The spindle is started

The CNC, depending on the position of the axis, calculates and makes the spindle turn at the corresponding rpm. speed for the Constant Surface Speed to be as defined.

As the axes move, when working at Constant Surface Speed, the following cases may arise:

a) The spindle is started

The CNC moves the axes to the F programmed.

As the X axis is moved, the CNC makes the spindle speed (rpm.) match to maintain the constant surface speed selected.

If the tool tip moves away from the turning axis the spindle revolutions therefore go down, and if it gets nearer, they go up.

The CNC limits the spindle revolutions to the maximum speed set «SMAX».

b) The spindle is stopped but a spindle speed S is selected

The CNC calculates, in millimeters/minute, the feedrate corresponding to the last programmed "S" and moves the axis.

For example, if «F 2.000» and «CSS 500»:
F (mm/min) = F (mm/min.) x S (rev/min) = 2 x 500 = 1000 mm/min
The axis moves at a feedrate of 1000 millimeters/minute

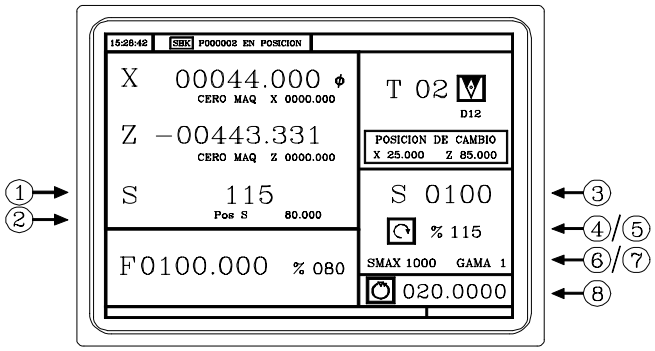
c) The spindle is stationary and there is no spindle speed S selected.

If feedrate F has value 0, the CNC moves the axes at fast feedrate.

If feedrate F has any other value, the axes will only be able to be moved if key  is pressed and the key for one axis. The CNC moves the axis at fast feedrate.

3.6.3 SPINDLE ORIENTATION

When having spindle orientation (general machine parameter REFEED1 (P34) other than 0) the CNC shows the following information:



1.- Actual spindle speed in rpm.

2.- Angular spindle position in degrees.

This data is displayed when working in Spindle Orientation mode. When switching to RPM mode, only the actual spindle speed is shown (1).

3.- Theoretical spindle speed in rpm.

To select another speed, press [S] The CNC will frame the current value.

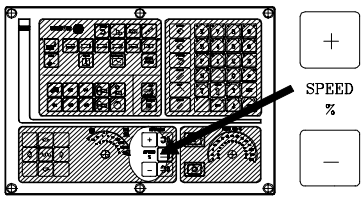
Enter the new value and press [I]. The CNC assumes this value and it updates the actual spindle speed.

4.- Spindle status: [↻] turning clockwise, [↺] turning counter-clockwise or [▽] .

When working in Spindle Orientation mode, it always shows the [▽] symbol.

5.- % 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 change this percentage (%) press:

6.- Maximum spindle rpm.

To select another speed, press [S] twice. The CNC will frame the current value.

Enter the new value and press [ENTER] The CNC assumes this value and will not let the spindle exceed these rpm.

7.- Selected spindle range.

To select another range when there is no automatic range changer, press [S] and then use the [↓] key to frame the current value.

Enter the range number to be selected and press [ENTER] or [I]

When the machine does not have a range changer, this message is superfluous. That's why the CNC does not show this message when text 28 of program 999997 is not defined.

8.- Angular spindle increment when operating in Spindle Orientation mode.

To select another value, press [S] key three times. The CNC will frame the current value.

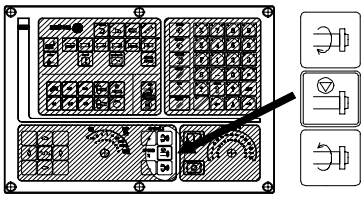
Key in the new value and press [ENTER]

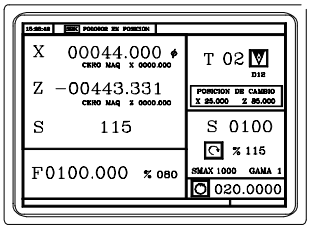
3.6.3.1 OPERATION WITH SPINDLE ORIENTATION

When having spindle orientation, the CNC uses the same screen as when operating in RPM mode.

RPM mode.

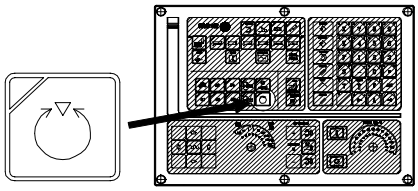
To select this mode, press one of these three keys. The screen will not show the angular position of the spindle.





Spindle Orientation mode:

To select this operating mode, press the key for spindle orientation:



The spindle will stop (if it was turning), it then searches home and, finally, it turns to the angular position indicated at the lower right-hand side of the screen (the top figure shows 20º).

Every time the key for Spindle Orientation is pressed, the spindle position is incremented by that amount (the top figure shows 20).

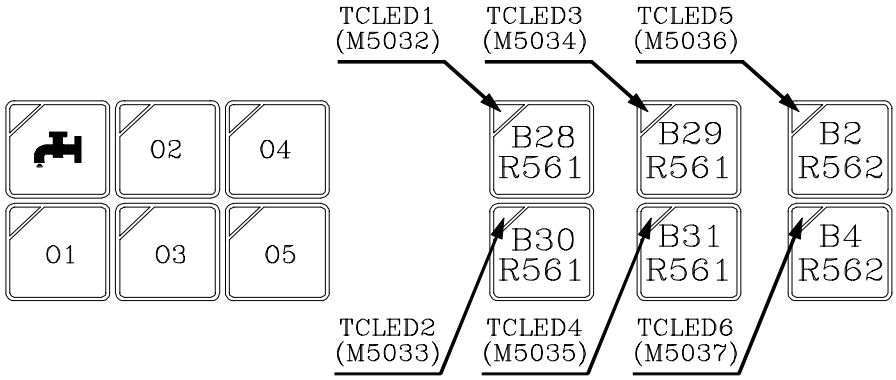
3.7 CONTROL OF EXTERNAL DEVICES

The CNC allows up to 6 external devices to be activated and deactivated from the keyboard. One of these is the cooling fluid.

The activation and deactivation of the devices must be carried out by the machine manufacturer by means of the PLC program.

The CNC will inform the PLC of the status of each one of the keys. The relevant Register bit will have value 1 when the key is pressed and value 0 when this is not pressed.

The Register bit for each one of the keys is as follows:



The status of the light for each one of these keys must be controlled by the machine manufacturer by means of the PLC program, with the TCLED* input variables shown in the figure being available for this purpose.

Examples:

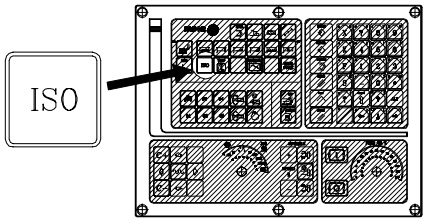
Control of the coolant: DFU B28R561 = CPL TCLED1
= CPL O33

Control of the tail-stock (O1). To activate or deactivate the tail-stock a number of conditions must be satisfied such as spindle stopped, ...

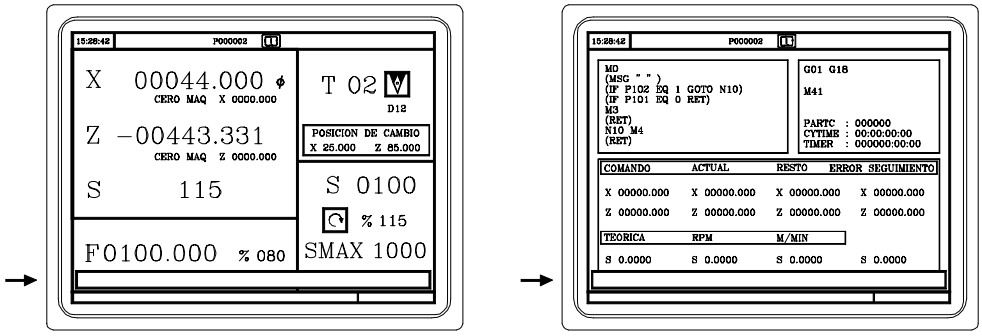
DFU B30R561 AND (Remaining conditions) = CPL TCLED2
= CPL O34

3.8 HANDLING ISO CODE

With the ISO key, it is possible to access the MDI mode or the ISO work mode.



To access the MDI mode, put the CNC in jog mode and then press ISO
The CNC displays a window at the bottom of the standard (or special) screen



An ISO coded block may be edited in this window to be executed later on, like in MDI in the "T" mode.

To access the ISO mode, while working with operations or cycles, press ISO once and while working in JOG mode, press that key twice.

When accessing the ISO mode, a special screen is displayed where it is possible to edit up to 6 program blocks in ISO or in high level language.

Example: [ISO]
G95 G96 S120 M3 ENTER
G0 Z100 ENTER
G1 X30 F0.1 ENTER

Once the desired block or blocks have been edited, press ESC The upper right-hand side of the screen will show the symbol

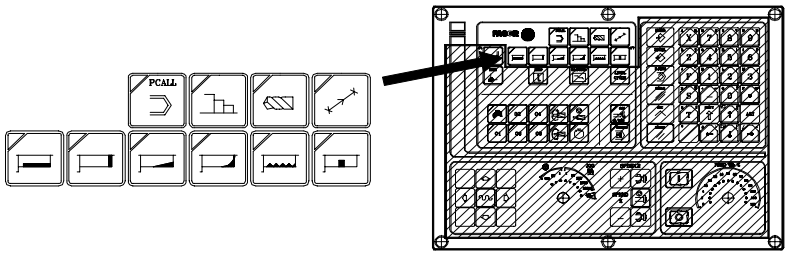
From this moment on, the blocks being edited may be simulated, executed or stored as any operation or cycle.

Press GRAPHICS to simulate it. Press to execute it.
It is possible to combine blocks edited in ISO code with standard or User machining cycles to make up part-programs. The chapter on "Storing part-programs" in this manual describes how to edit them and operate with them.

To store blocks edited in ISO code, press P.PROG

4. OPERATING WITH OPERATIONS OR CYCLES

The following CNC keys should be used to select the machining Operations or Cycles:



When pressing the CNC shows all the user cycles already defined by the machine manufacturer by means of the application program: WGDRAW.

The user cycle is edited like any other standard cycle of the TC mode.

Once all the necessary data has been defined, the user can Simular or Execute the cycle like any other standard cycle of the TC mode.

When pressing any other key, the CNC selects the corresponding standard machining cycle, changes the screen and lights up the lamp of the key just pressed indicating that the cycle has been selected.

The machining operations or cycles that can be selected with each one of these keys are:

- Positioning cycle
- Facing cycle
- Rounding cycle
- Grooving cycle
- Profile Cycle
- Turning cycle
- Taper cycle
- Threading cycle
- Drilling and tapping cycles

When the Machining Operation or Cycle involves several levels key [LEVEL CYCLE] has to be pressed to select the cycle level required:

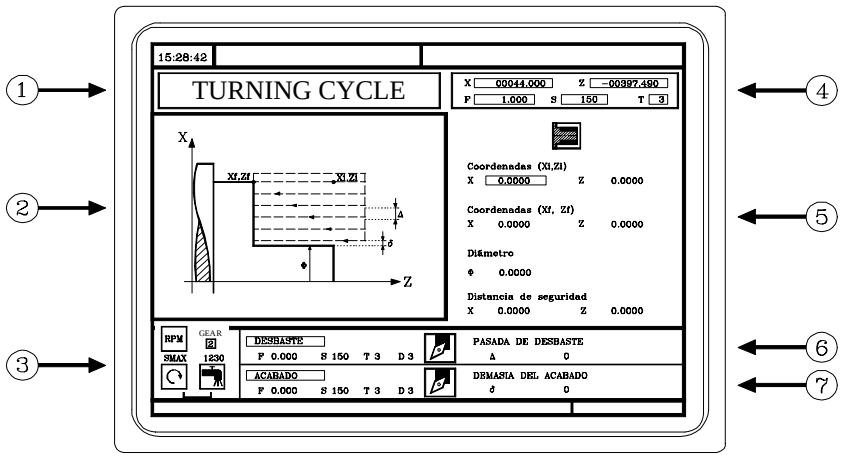
With this CNC, it is possible to combine ISO-coded blocks with standard and/or user-defined machining operations to create part-programs as described in the chapter on "Part-program storage" in this manual.

To deselect the cycle and return to the standard display, press the key corresponding to the selected cycle (the one with the indicator lamp on) or [ESC]

Note: the operations or cycles can modify global parameters 150 through 299 (both included).

4.1. OPERATION EDIT MODE

After selecting the cycle edition mode the CNC displays a screen looking like this:



1.- Denomination of the Operation or Work cycle selected.

2.- Help graphics.

3.- Spindle Conditions for cycle execution.

4.- Present status of the machine. Coordinates and machining conditions.

5.- Data defining the geometry of the Machining Cycle.

6.- Machining conditions for the roughing operation.

7.- Machining conditions for the finishing operation.

The CNC will display an icon, a coordinate or one of the data items defining the operation or cycle in highlighted print, to show that this item has been selected.

To select another icon, data item or coordinate, one can:

a) Use keys ← → ↑ ↓ . The CNC will select the previous or next data item.

b) Press key X or Z . The CNC will select the first data item for said axis. Pressing this key again will select the following piece of data for said axis.

c) Press key F or T . The CNC selects the relevant roughing data. Pressing said key again will select the relevant finishing data.

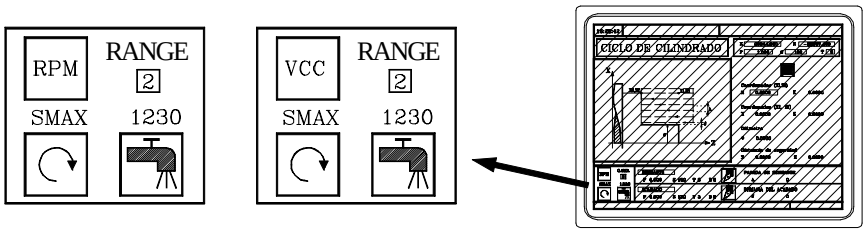
d) Press key S. The CNC selects data item «S» for roughing. Pressing this key a second time selects data item «S» for finishing, and pressing it again selects the data for spindle SMAX.

The coordinates for axis X are defined in operating units, radii or diameters.

Later on, in each one of the operations or cycles, the units in which the data associated with axis X (safety distance, pass, excess stock, etc.) are displayed.

4.1.1 DEFINITION OF SPINDLE CONDITIONS

Work mode (RPM) or (CSS)



Move over the "RPM" or "CSS" icon. this can be done:

a) By using the keys

b) by pressing . The CNC selects this item directly.

After selecting the item, press or to change the work mode.

Spindle range

Move over this item, key in the desired value and press ENTER

Maximum spindle turning speed in rpm (S)

Move onto this item, type in the required value and press ENTER

Spindle turning direction

There are 2 ways to select the spindle turning direction:

a) Move onto this item and press to change the icon.

b) Start the spindle in the direction required by means of the JOG keys

The CNC starts the spindle and assumes said turning direction as spindle turning data for the cycle.

Coolant

Place the cursor over this data and press to change the icon.

means turning the coolant on. The CNC outputs the M8 function to the PLC.

means turning 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.

4.1.2 DEFINITION OF MACHINING CONDITIONS

Some cycles maintain the same machining conditions during the whole execution process (positioning cycle, drilling cycle ...)

Other cycles use machining cycles for roughing and other conditions for finishing ,(turning cycle, rounding cycle, ...)

This section shows how all this data has to be defined.

Axis feedrate (F)

Move onto this item, key in the value required and press ENTER

Spindle turning speed (S)

Move onto this item, key in the value required and press ENTER

Tool for machining (T)

Move onto this item, key in the value required and press ENTER

The CNC updates the "D" offset and the adjacent icon, displaying the graphic representation for the location code associated with the new tool.

Tool calibration may also be accessed to consult or change the data corresponding to the selected tool. To do this, position over the "T" field and press

To exit the tool calibration mode and return to the cycle, press ESC

Roughing Pass (Δ)

Move onto this item, key in the value required and press ENTER

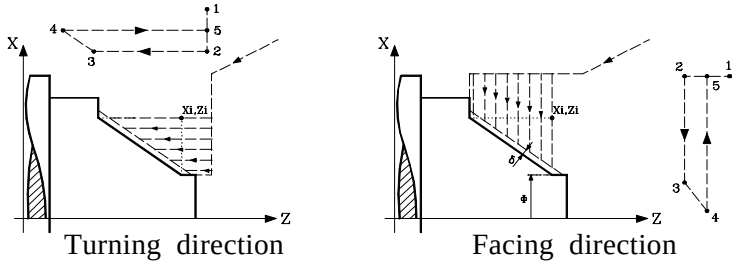
Finishing Stock (δ)

Move onto this item, key in the value required and press ENTER

The finishing stock is always defined in radii.

Machining direction

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



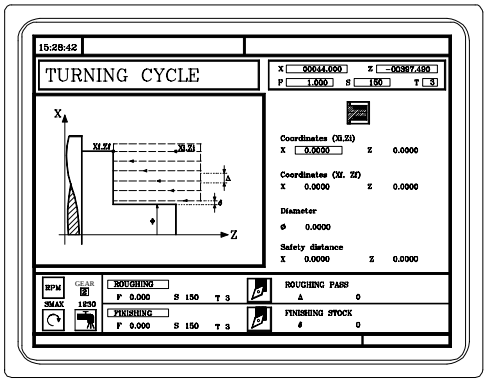
Turning direction Facing direction

To do this, move onto this icon and press

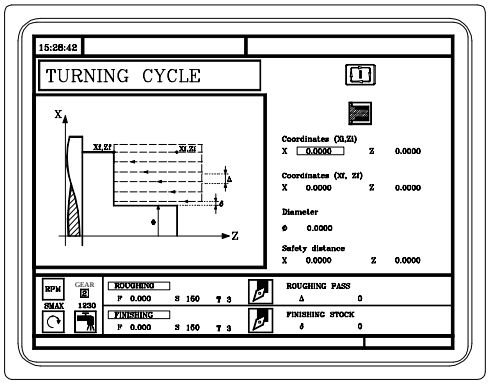
The icon changes and the help graph is updated.

4.2 SIMULATING AND EXECUTING THE CYCLE

There are ways to work with operations or cycles: Editing mode and Execution mode.



Editing mode



Execution mode

To switch from Editing mode to Execution mode, press 

To switch from the Execution mode to the Editing mode, press one of these keys:

An operation or cycle may be simulated in any of these modes. To do that, press 

For further information, refer to the chapter on "Execution and Simulation" of this manual.

To execute an operation or cycle, select the Execution mode and press 

For further information, refer to the chapter on "Execution and Simulation" of this manual.

4.2.1 BACKGROUND EDITING OF CYCLES

It is possible to edit an operation or cycle while running a part-program (background editing).

The new operation edited may be stored as part of the part-program (other than the one being executed).

No operation being edited in the background may be executed or simulated, or the current position of the machine be assigned to a coordinate.

To make a tool inspection or change while background editing, proceed as follows:

Press  => To interrupt the execution of the program and resume background editing.

Press 

=> To quit background editing.

Press 

=> To go into tool inspection.

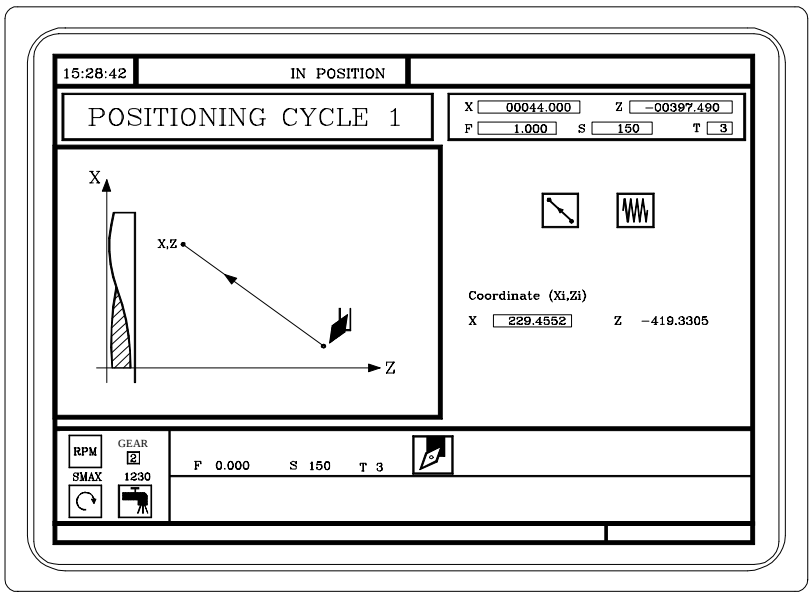
When pressing [T] without quitting background editing, it select the "T" field of the operation or canned cycle being edited.

4.3 POSITIONING CYCLE

To select the Positioning cycle press 

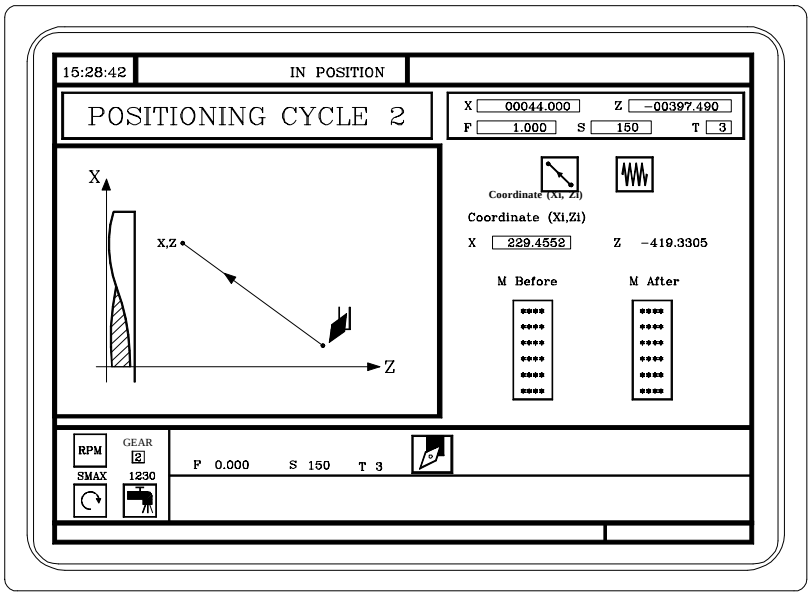
This cycle can be defined in two different ways

Level 1.



The coordinates of the target point have to be defined
The way the positioning is to be done
The type of feedrate, fast or at the F stated

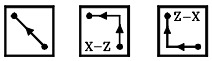
Level 2.

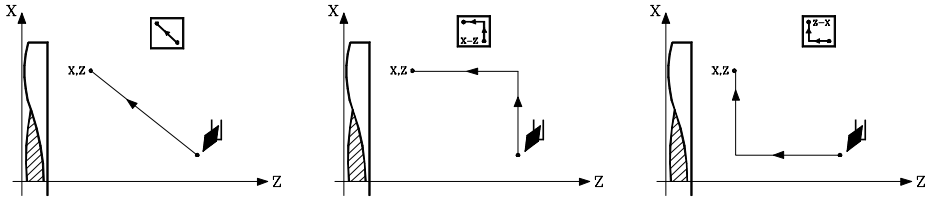


The coordinates of the target point have to be defined
The way the positioning is to be done
The type of feedrate, fast or at the F stated
The auxiliary functions to be executed before and after positioning

To change the level, that is, to go from one level to the next, LEVEL CYCLE should be pressed.

4.3.1 DEFINITION OF DATA

Type of positioning 



To select the type of positioning move onto this icon and press 

The icon changes and the help graphics are updated.

Type of feedrate

 Feedrate at te selected F

 In rapid traverse

To select the type of feedrate move onto this icon and press 

Coordinates of the target point (X,Z)

The coordinates are defined one by one. After moving onto the coordinates for the axis required for definition, one can:

a) Manually enter the value. Enter the value required and press 

b) Assign the present position of the machine.

Move the axis, by means of the handwheel or the JOG keys, to the point required. The top right-hand window shows the tool coordinate at all times.

Press  for the data item selected to take on the value displayed in the top right-hand window.

Press 

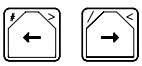
The auxiliary functions "M" which will be executed before and after positioning

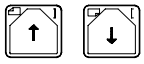
Auxiliary function "M" is the name given to the functions determined by the manufacturer which allow the different machine devices to be governed.

There are auxiliary functions "M" for activating a program stop, for selecting the spindle turning direction, for controlling the coolant, for controlling the spindle gearbox, etc..

The Programming manual states how these functions should be programmed and the Installation manual explains how the system should be set to operate with them.

To define the auxiliary functions to be executed before and after positioning:

a) Move into the relevant window by means of the  keys

To move around within the window use the  keys

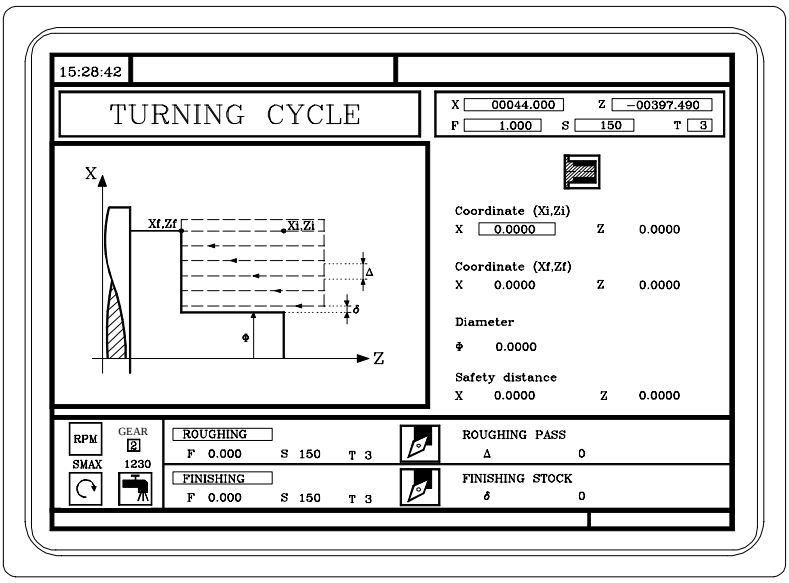
b) Define the auxiliary functions required.

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

To erase a function, select this and press 

4.4 TURNING CYCLE

To select the turning cycle press 



4.4.1 DEFINITION OF GEOMETRY

Type of turning: internal or external

 External turning

 Internal turning

To modify the type of turning move onto this icon and press 

Each time the turning type is changed the CNC modifies the icon and displays the relevant geometrical help screen.

Coordinates of the starting point (Xi, Zi) and coordinates of the end point (Xf, Zf)

The coordinates are defined one by one. After moving onto the coordinates of the axis to be defined, one can:

a) Manually enter the value. Key in the value required and press 

b) Assign the present position of the machine.

Move the axis, by means of the handwheel or the JOG keys, to the point required. The top right-hand window displays the tool coordinate at all times.

Press  for the data item selected to take on the value displayed in the top right-hand window.

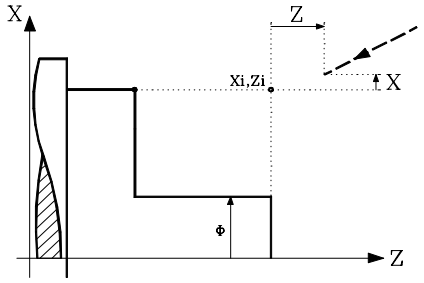
Press 

Final diameter (Φ)

Move onto this item, key in the required value and press 

Safety distance

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



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

To modify one of these values move over the relevant data item, key in the required value and press ENTER.

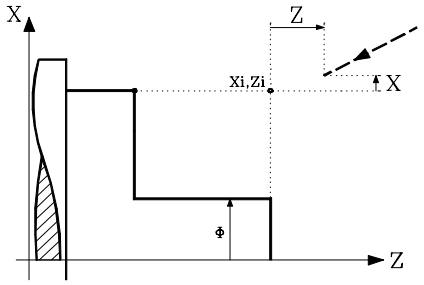
4.4.2 BASIC OPERATION

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 requires this.

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

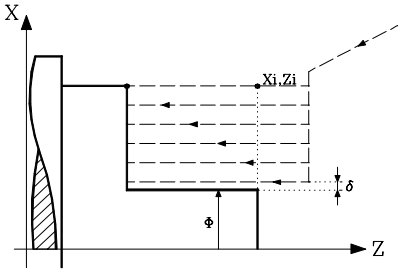
3.- The tool approaches the starting point (Xi, Zi) at fast feedrate speed, keeping the selected safety distance according to axes X and Z.

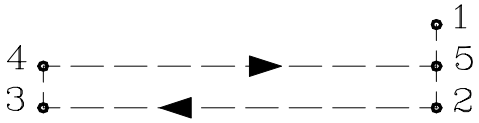


4.- Roughing operation, by successive turning passes, to a distance from the final diameter selected equal to the finishing excess material.

This operation is done with the conditions set for the roughing operation; nevertheless, the CNC calculates the real pass for all the turning passes to be equal. 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”.

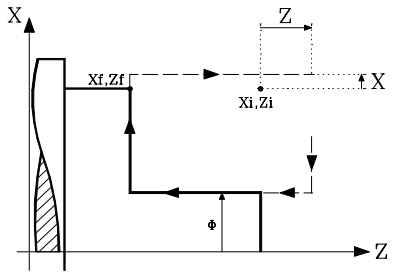




5.- Finishing Operation

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

The part finishing is done with the machining conditions set for finishing; axis feedrate (F), spindle speed (S), turning direction.



6.- Once the operation or cycle has ended, the tool will return to the position it occupied when the cycle was called upon, that is, the point where [I] was pressed.

Obviously, when executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.

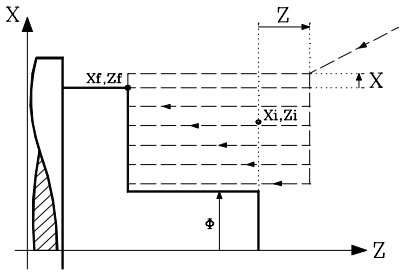
7.- The CNC will stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

Some points to consider:

If T0 is selected as roughing tool, the cycle does not execute the roughing operation. This means that after approaching the finishing operation will be carried out.

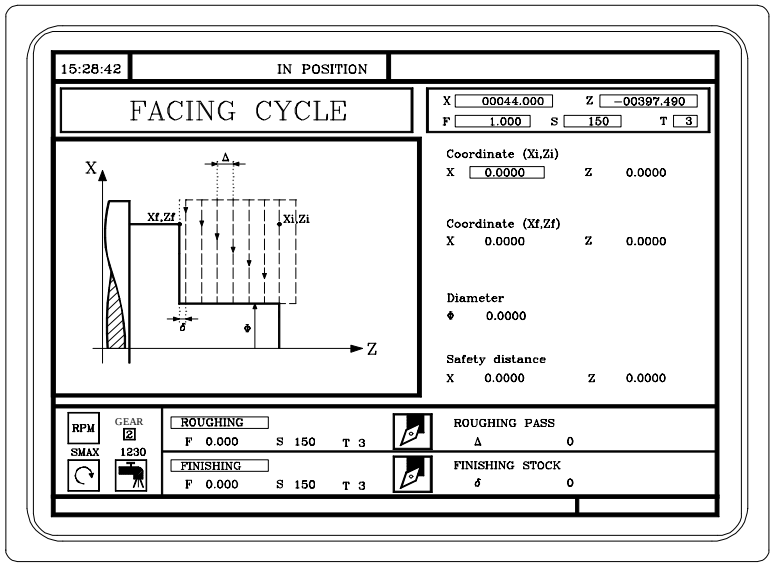
If T0 is selected as finishing tool, the cycle does not execute the finishing operation. This means that, after the roughing operation, the tool will move to the approach point, keeping the safety distance with respect to the starting point (Xi, Zi).

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



4.5 FACING CYCLE

To select the Facing cycle press 



4.5.1 DEFINITION OF GEOMETRY

Coordinates of the starting point (Xi, Zi) and coordinates of the end point (Xf, Zf)

The coordinates are defined one by one. After moving over the coordinates for the axis to be defined, one can:

a) Manually enter the value. Key in the value required and press 

b) Assign the present machine position.

Move the axis by means of the handwheel or the JOG keys to the point required. The top right-hand window displays the machine position at all times.

Press 

for the data item selected to assume the value displayed in the top right-hand window.

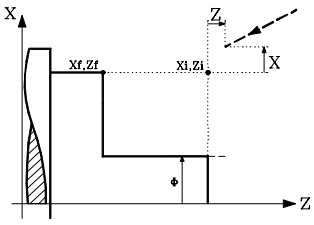
Press 

Final diameter (Φ)

Move onto this item, key in the value required and press 

Safety Distance

In order to prevent collisions with the part, the CNC enables an approach point for the part to be established. The safety distance indicates the position of the approach point with respect to the starting point (Xi, Zi).



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

To modify any of these values move onto the corresponding data item, key in the value required and press 

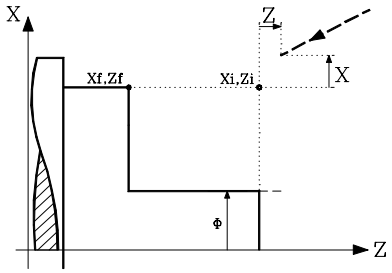
4.5.2 BASIC OPERATION

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 requires it.

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

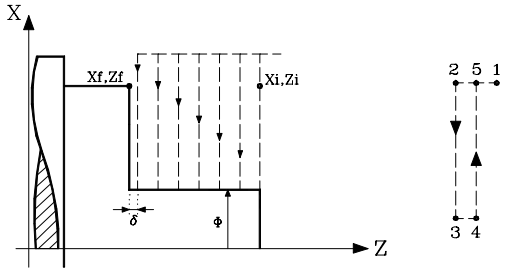
3.- The tool approaches the starting point (Xi, Zi) at fast feedrate speed, keeping the selected safety distance according to axes X and Z.



4.- Roughing Operation, by means of successive facing passes, up to a distance from the final Z coordinate (Zf) equal to the finishing stock.

This operation is done with the conditions set for the roughing operation; nevertheless, the CNC calculates the real pass for all the facing passes to be equal. This pass will be equal to or under the defined value **Δ**.

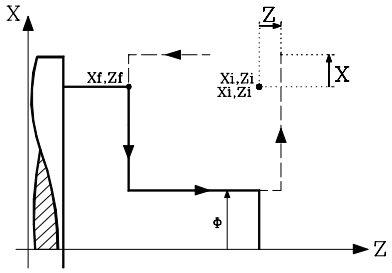
Each facing pass is done as shown in the figure, starting at point “1” and after going through points “2”, “3” and “4”, ending at point “5”.



5.- Finishing Operation

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

The part finishing is done with the machining conditions set for finishing; axis feedrate (F), spindle speed (S), turning direction.



6.- Once the operation or cycle has ended, the tool will return to the position it occupied when the cycle was called upon, that is, the point where [i] was pressed.

Obviously, when executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.

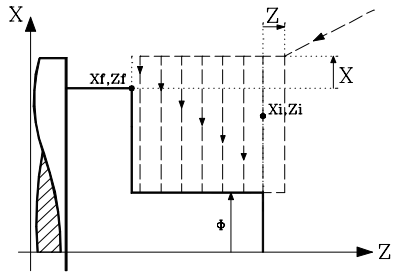
7.- The CNC will stop the spindle but keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and spindle speed (S).

Some points to consider:

If T0 is selected as roughing tool, the cycle does not execute the roughing operation. This means that, after approach, the finishing operation will be carried out.

If T0 is selected as finishing tool, the cycle does not execute the finishing operation. This means that, after the roughing operation, the tool will move to the approach point, keeping the safety distance with respect to the starting point (Xi, Zi).

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

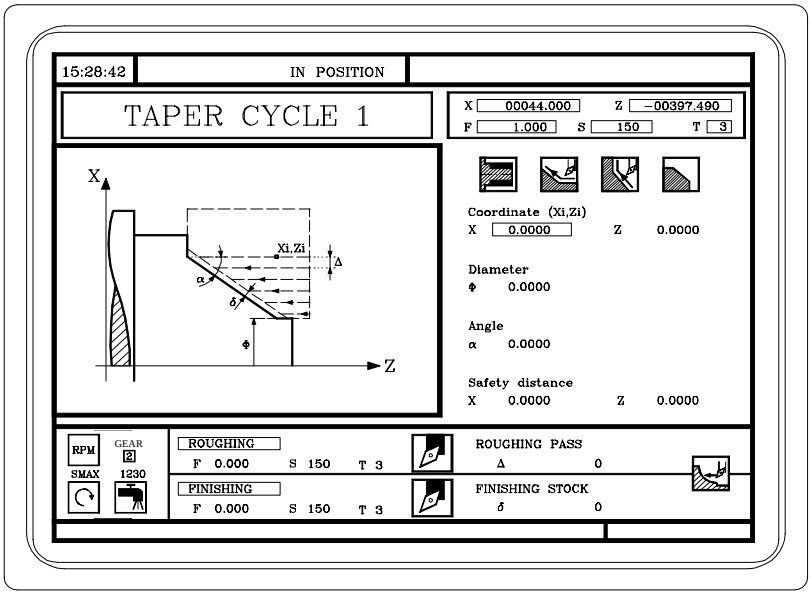


4.6 TAPER CYCLE

To select the taper cycle press 

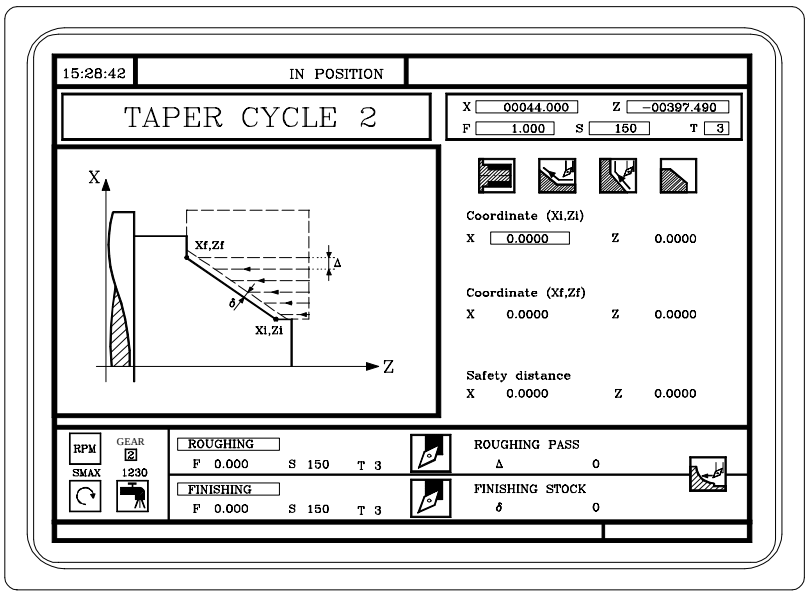
This cycle can be defined in two different ways:

Level 1.



A definition is needed for The coordinates of the theoretical corner.
The taper angle and final diameter.

Level 2.



A definition is needed for The coordinates of the starting point.
The coordinates of the end point.

To change the levels press 

4.6.1 DEFINITION OF GEOMETRY

Type of taper: internal or external

 External taper

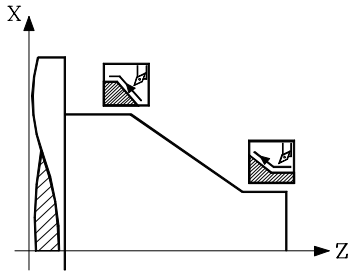
 Internal taper

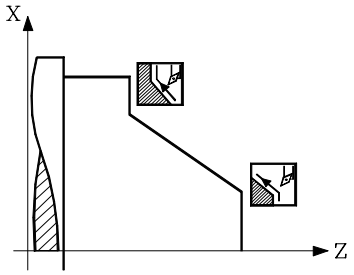
To modify the type of taper move over this icon and press 

Whenever the type of taper is changed the CNC modifies the icon and displays the relevant geometrical help screen.

Shape of the part before and after the tapered section

The icons  and  define the type of section before and after the tapered section.



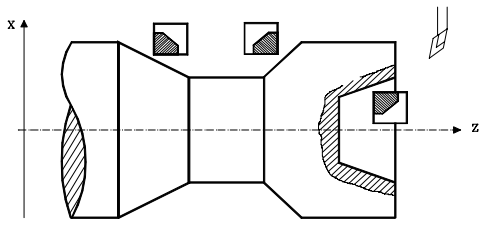


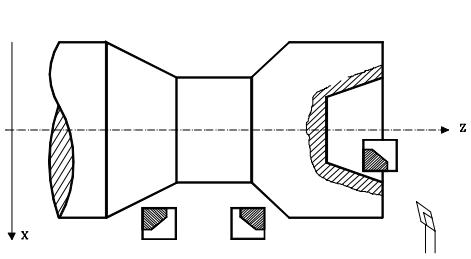
To modify the type of section move over the relevant icon and press 

Whenever one of these is changed the CNC modifies the icon and displays the relevant geometrical help screen.

Work quadrant

The icon  defines the type of corner required to be machined.





To modify the work quadrant move over the icon and press  . The CNC will display the next icon available.

Coordinates of the theoretical corner or starting point (Xi, Zi) and coordinates of the end point (Xf, Zf)

The coordinates are defined one by one. After moving over the coordinates of the axis to be defined, one can either:

a) Manually enter the value. Type in the required value and press ENTER

b) Assign the present position of the machine

Move the axis, by means of the handwheel or the JOG keys, to the point required. The top right-hand window displays the tool position at all times.

Press RECALL or the data item selected to take on the value displayed in the top right-hand window.

Press ENTER

Final Diameter (Φ)

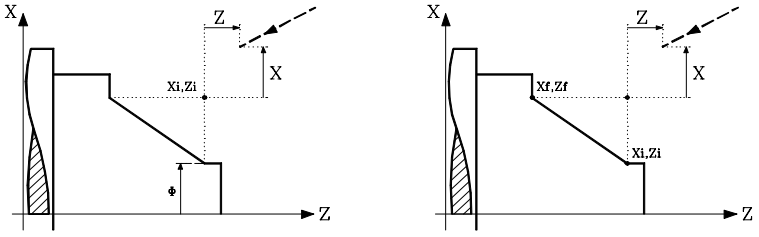
Move onto this data item, type in the value required and press ENTER

Angle (α)

Move onto this data item, type in the value required and press ENTER

Safety Distance

In order to avoid collisions with the part, the CNC allows a point for approach to the part to be defined. The Safety Distance indicates the position of the approach point with respect to the theoretical corner.



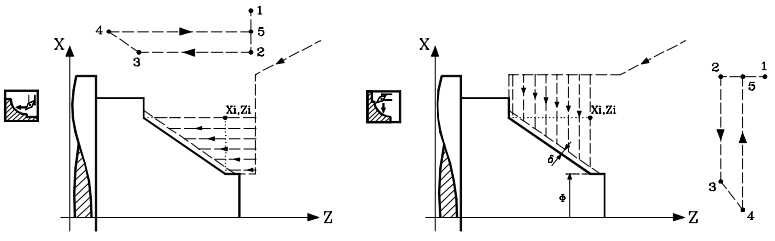
The value of the safety distance on X is always defined in radii.

To modify one of these values move onto the relevant data item, type in the required value and press ENTER key.

Machining direction

To select the machining direction (taper direction or facing direction) move onto the icon for the data zone for roughing and finishing and press key.

The icon changes and the help graphic is updated.



Taper direction Facing direction

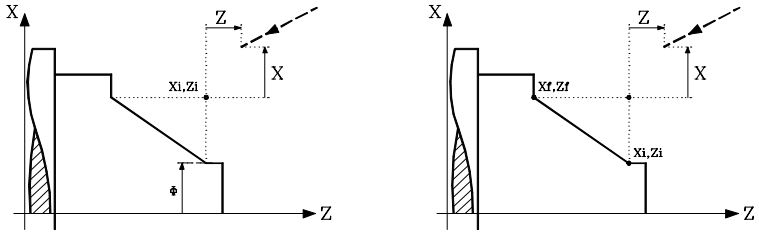
4.6.2 BASIC OPERATION

The machining stages in this cycle are as follows:

1.- If the roughing operation was programmed with another tool the CNC will make a tool change, moving to the change point if this is required by the machine.

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

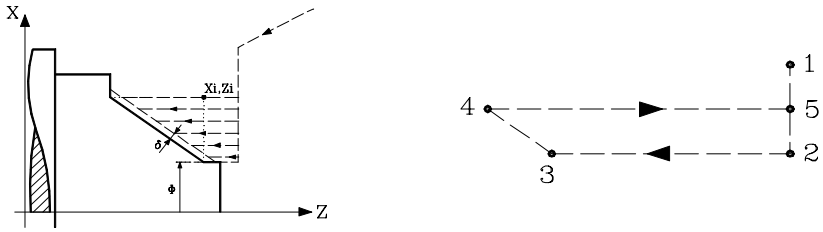
3.- The tool approaches the theoretical corner at fast feedrate, keeping the safety distance selected according to axes X and Z.



4.- Roughing operation, by means of successive passes, up to a distance equal to the excess stock for finishing the profile selected.

This operation is done with the conditions set for the roughing operation.
Nevertheless, the CNC calculates the real pass so that all the passes are equal. This pass will be equal to or under the one defined **Δ**.

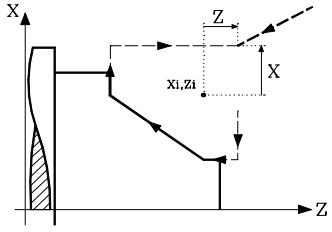
Each taper pass is done as shown in the figure, starting at point “1” and after going through points “2”, “3” and “4”, ending at point “5”.



5.- Finishing operation

If the finishing operation was programmed with another tool, the CNC makes a tool change, moving to the change point if this is required by the machine.

The finishing of the part is done with the machining conditions set for finishing: axis feedrate (F), spindle speed (S), turning direction.



6.- Once the operation or cycle has ended, the tool will return to the position it occupied when the cycle was called upon, that is, the point where  was pressed.

Obviously, when executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.

7.- The CNC will stop the spindle but keeps the machining conditions set for finishing selected: tool (T), axis feedrate (F) and spindle speed (S).

Some points to consider:

If T0 is selected as a roughing tool, the cycle does not execute the roughing operation. This means that after approach the finishing operation will be carried out.

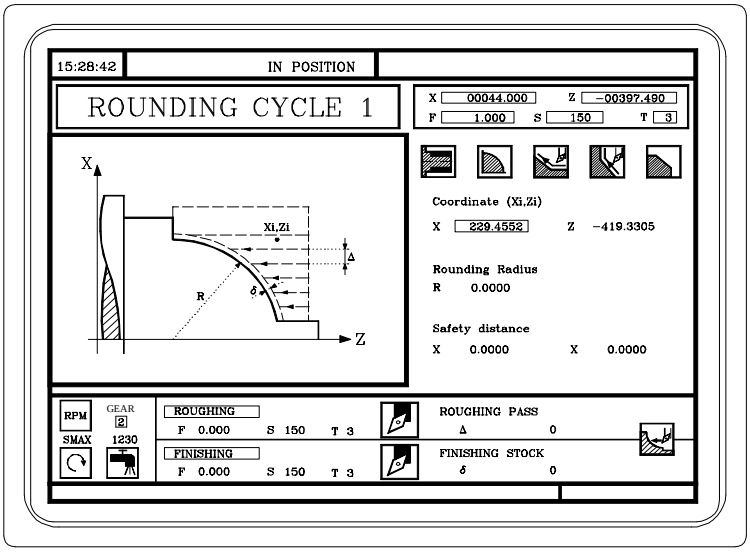
If T0 is selected as a finishing tool, the cycle does not execute the finishing operation. This means that after the roughing operation the tool will move to the approach point, keeping the safety distance with respect to the starting point (Xi, Zi).

4.7 ROUNDING CYCLE

To select the Rounding Cycle press 

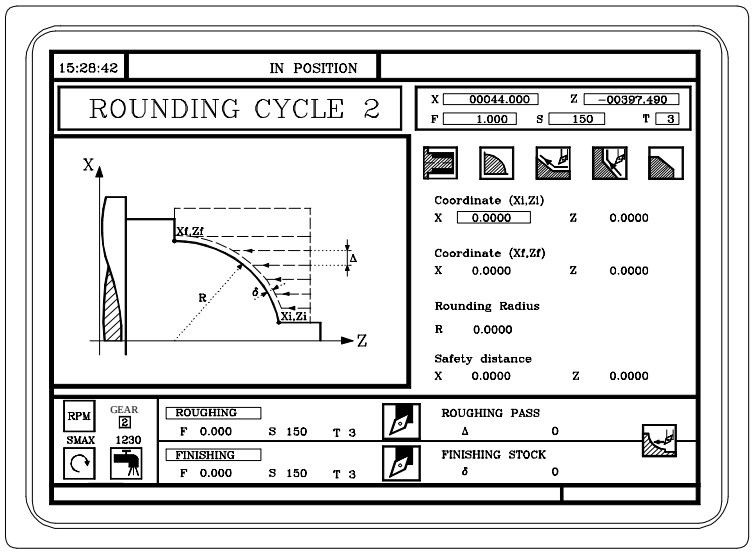
This cycle can be defined in two different ways:

Level 1.



A definition is needed for The coordinates of the theoretical corner
The rounding radius

Level 2.



A definition is needed for The coordinates of the starting point
The coordinates of the end point.
The rounding radius.

To change the levels press 

4.7.1 DEFINITION OF GEOMETRY

Type of rounding: internal or external

External rounding Internal rounding.

To modify the type of rounding move onto this icon and press

Whenever the type of rounding is changed the CNC modifies the icon and displays the relevant geometrical help screen.

Concave and convex rounding

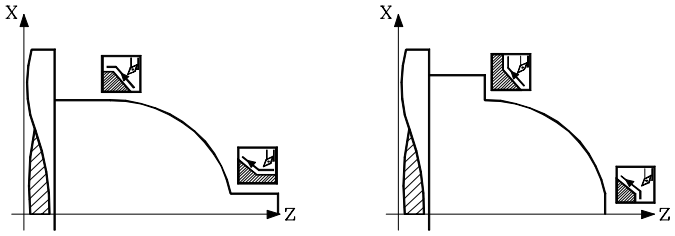
The icons and define the type of rounding required to be done.

To modify the type of rounding move onto the relevant icon and press

Whenever any of these is changed the CNC modifies the icon and displays the relevant geometrical help screen.

Shape of the part before and after the rounding section

The icons and define the type of section before and after the rounding section.

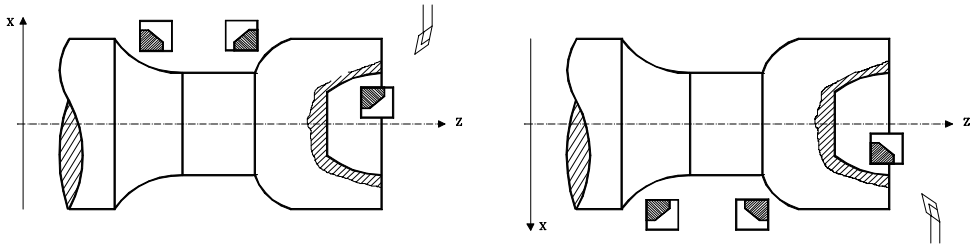


To modify the type of section move onto the relevant icon and press

Whenever any of these is changed the CNC modifies the icon and displays the relevant geometrical help screen.

Work quadrant

The icon defines the type of corner required for machining.



To modify the work quadrant move onto the icon and press . The CNC will display the next icon available.

Coordinates of the theoretical corner or the starting point (Xi, Zi) and coordinates of the end point (Xf, Zf)

The coordinates are defined one by one. After moving onto the coordinates of the axis required to be defined, one can:

a) Manually enter the value. Type in the value required and press ENTER

b) Assign the present machine position

Move the axis, by means of the handwheel or the JOG keys, to the point required. The top right-hand window displays the tool position at all times.

Press RECALL for the data item selected to take on the value displayed in the top right-hand window.

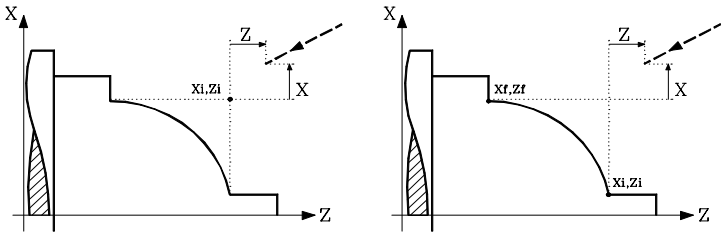
Press ENTER

Rounding radius (R)

Move over this data item, type in the value required and press ENTER

Safety Distance

In order to prevent collisions with the part, the CNC allows an approach point for the part to be established. The safety distance indicates the position of the approach point with respect to the theoretical corner.



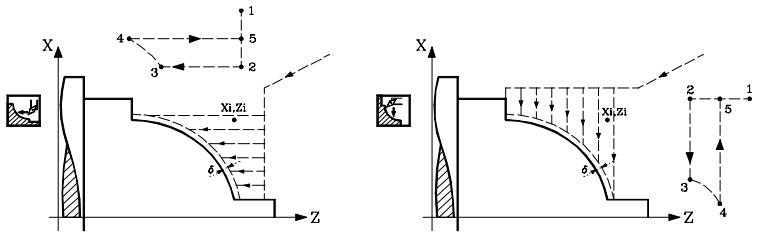
The value of the safety distance on X is always defined in radii.

To modify one of these values move over the corresponding data item, type in the required value and press ENTER .

Machining direction

To select the machining direction (turning direction or facing direction), move onto the icon for the zone for Roughing and Finishing data and press

The icon changes and the help graphic is updated.



Turning direction Facing direction

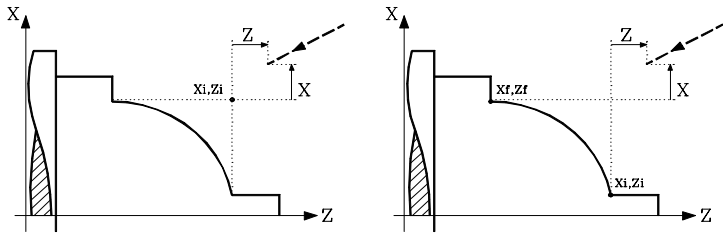
4.7.2 BASIC OPERATION

The machining stages in this cycle are as follows:

1.- If the roughing operation was programmed with another tool the CNC makes the tool change, moving to the change point if this is required by the machine.

2.- The spindle starts at the speed selected and in the direction indicated.

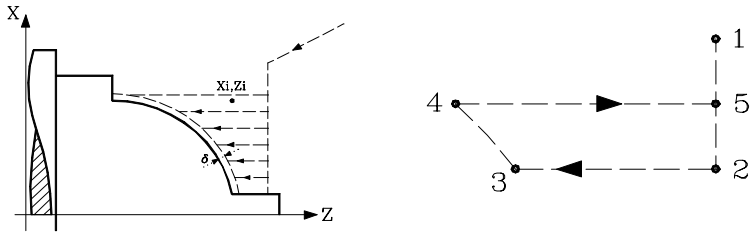
3.- The tool approaches the theoretical corner at fast feedrate, keeping the selected safety distance according to axes X and Z.



4.- Roughing operation, by successive passes, up to a distance equal to the finishing stock of the selected profile.

This operation is done with the conditions set for the roughing operation.
Nevertheless, the CNC calculates the real pass so that all the passes are equal. This pass will be equal to or under the one defined **Δ**.

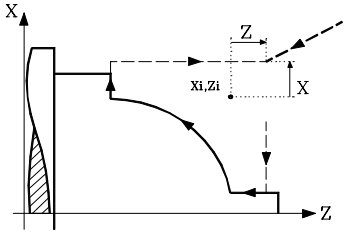
Each taper pass is done as shown in the figure, starting at point “1” and after going through points “2”, “3” and “4”, ending at point “5”.



5.- Finishing operation.

If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if this is required by the machine.

The part finish is done with the machining conditions set for finishing: axis feedrate (F), spindle speed (S), turning direction.



6.-Once the operation or cycle has ended, the tool will return to the position it occupied when the cycle was called upon, that is, the point where  was pressed.

Obviously, when executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.

7.- The CNC will stop the spindle but keep the machining conditions set for finishing; tool (T), axis feedrate (F) and spindle speed (S).

Some points to consider:

If T0 is selected as roughing tool, the cycle will not execute the roughing operation. This means that after approach the finishing operation will be carried out.

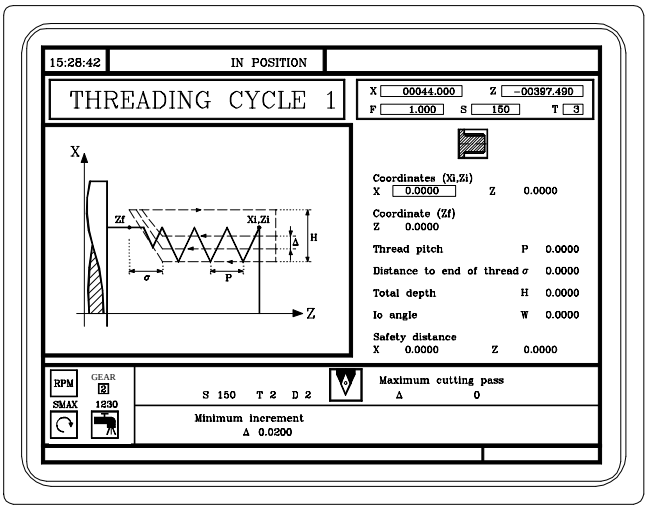
If T0 is selected as finishing tool, the cycle will not execute the finishing operation. This means that after the roughing operation the tool will move to the approach point, keeping the safety distance with respect to the starting point (Xi, Zi).

4.8 THREADING CYCLE

To select the Positioning cycle press 

This cycle can be defined in four different ways:

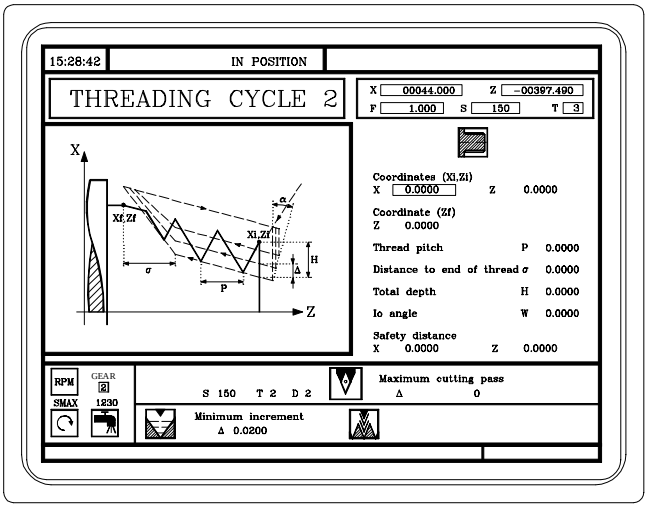
Level 1. Longitudinal threading



A definition is needed for:

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

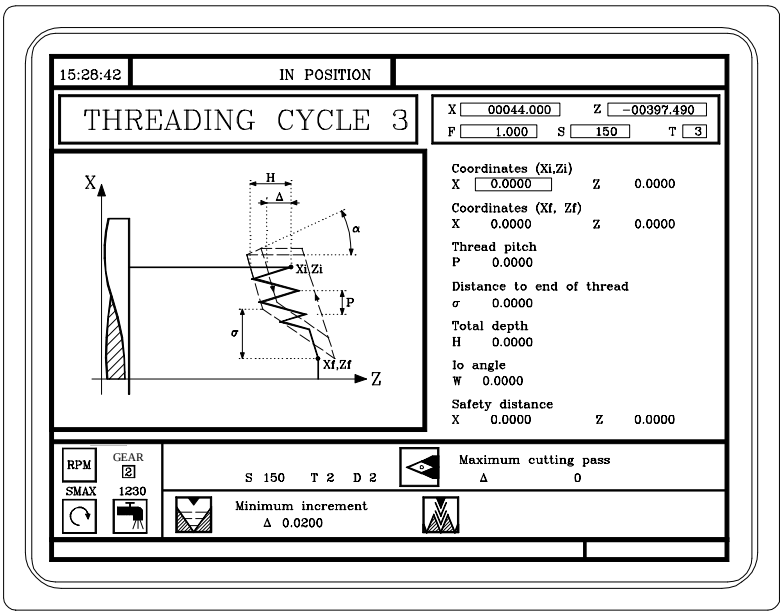
Level 2. Taper threading



Definition must be made of

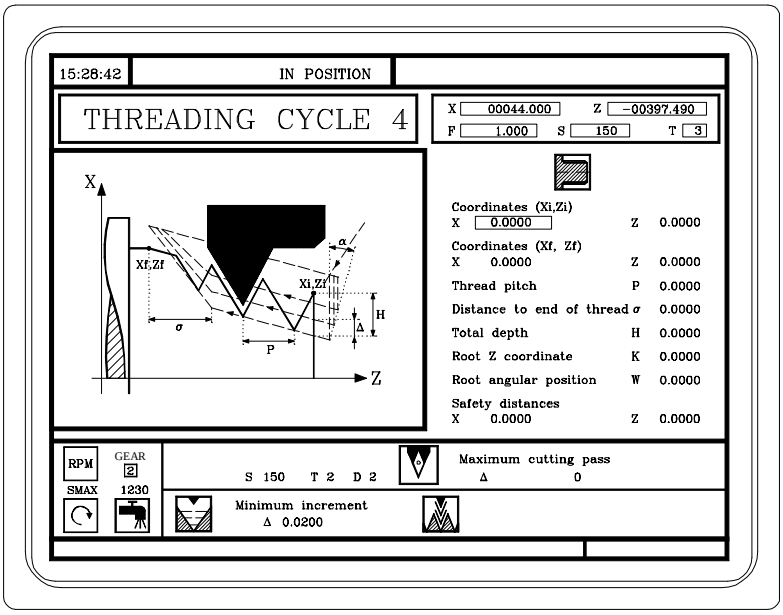
- The coordinates of the starting point
- The coordinates of the end point
- The thread pitch
- The distance to the end of thread
- The total depth

Level 3. Face threading



A definition is needed for:
- The coordinates of the starting point
- The coordinates of the end point
- The thread pitch
- The distance to the end of thread
- The total depth
- The spindle angular position

Level 4. Thread repair. Available when spindle machine parameter "M19TYPE (P43) = 1"



A definition is needed for:
- The coordinates of the starting point
- The coordinates of the end point
- The thread pitch
- The distance to the end of thread
- The total depth
- Z coordinate of the root of the thread
- Spindle angular position at the root of the thread

To change levels press LEVEL CYCLE

4.8.1 DEFINITION OF GEOMETRY

Type of threading: internal or external

 External Threading  Internal threading.

To modify the type of threading move onto this icon and press 

Whenever one changes the type of threading the CNC modifies the icon and displays the relevant geometrical help screen.

Starting point coordinates (Xi, Zi) and end point coordinates (Xf, Zf)

The coordinates are defined one by one. After moving over the coordinates for the axis required for defining, one can:

a) Manually enter the value. Key in the value required and press 

b) Assign the present machine position

Move the axis by means of the handwheel or the JOG keys, to the required point. The top right hand window shows the position of the tool at all times.

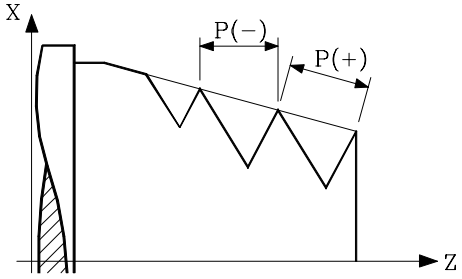
Press  for the data item selected to take on the value shown in the top right-hand window

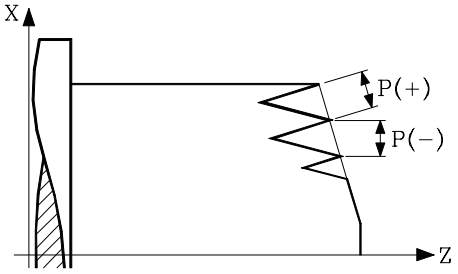
Press 

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" must be used, but with different signs:
«P» with positive sign. When programming the pitch along the taper of the thread.
«P» with negative sign. When programming the pitch along the associated axis.





To set the the pitch, move over this data item, key in the value required and press 

Distance to end of Thread (σ)

Indicates at what distance from the end of the thread the thread starts to be reduced. In this withdrawal movement threading is continued.

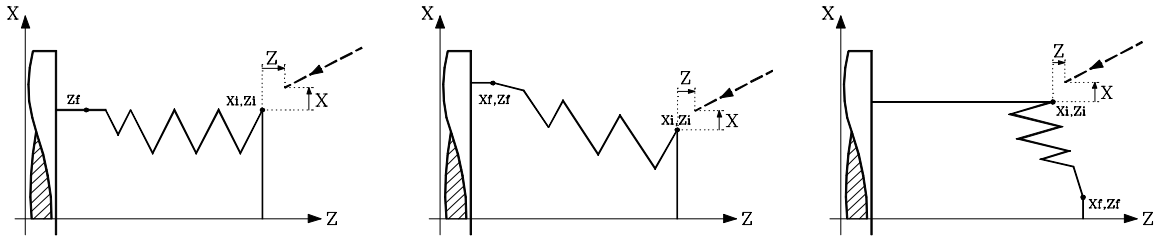
Move on to this data item, key in the required value and press 

Total thread depth (H)

The total depth of the thread should be programmed in radii and with positive value. To define this value, move onto this data item, key in the required value and press ENTER.

Safety distance

In order to prevent collisions with the part, the CNC allows an approach point to the part to be established. The safety distance indicates the position of the approach point with respect to the theoretical corner.



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

To modify one of these values, move over the corresponding data item, key in the required value and press ENTER.

Spindle angular position

In longitudinal threading (level 1), taper threading (level 2) and face threading (level 3), the "Io Angle" indicates the angular position of the spindle where the thread must start, referred to home.

With this option, it is possible to make multiple entry threads without withdrawing to the starting point.

In the Thread Repair Cycle (level 4), it indicates the angular position of the spindle at the root of the thread and it must be programmed together with the "Root Z coordinate" as indicated later on.

4.8.2 DEFINITION OF THE TYPE OF MACHINING

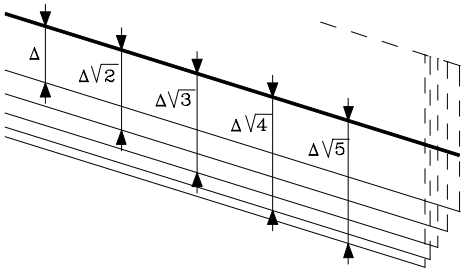
Threading levels 2 and 3 enable selection by means of the icons located in the lower left-hand window of the way the threading passes are to be made.

Depth of the successive threading passes (Δ)

The data item **Δ** fixes the maximum pass of depth and icons and define how the successive machining passes are made.

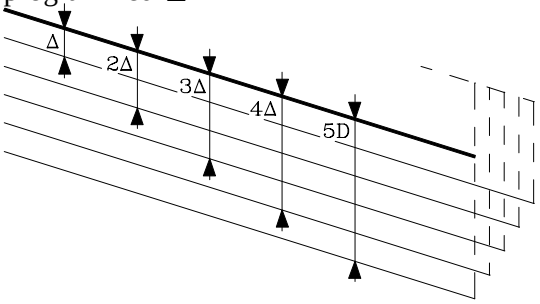
If icon is selected the depth of each pass will be a function of the corresponding pass number

The depths are: **Δ, Δ√2, Δ√3, Δ√4, ...**



If the penetrating increment (difference between penetrations) calculated by the CNC is smaller than the minimum penetrating increment, the CNC assumes this latter value.

If icon is selected the increase in deepening remains constant between passes, with a value identical to the one programmed **Δ**



To define how the successive threading passes are made, move over this icon and press

In the case of threading cycle Level 1, the depth of each pass will depend on the corresponding pass number. **Δ, Δ√2, Δ√3, Δ√4, ...**

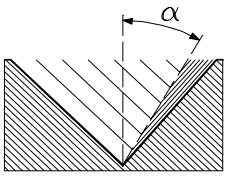
Type of tool penetration

The penetration of the tool may be:

Radial Per flank Zigzag

To define the type of penetration, move onto this icon and press

If penetration type «Per flank» or «zigzag» is selected, the CNC will ask for the angle (**α**) of penetration of the cutter.



In the case of threading cycle Level 1, the type of penetration is always radial.

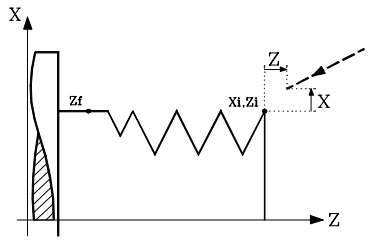
4.8.3 LONGITUDINAL THREADING. BASIC OPERATION.

The machining passes in this cycle are as follows:

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

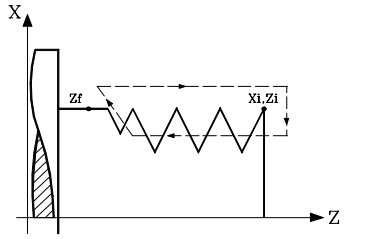
2.- The spindle starts at the speed selected and in the direction indicated.
Depending on the spindle turning direction, the thread will be clockwise or counterclockwise.

3.- The tool approaches at fast feedrate up to the starting point, keeping the safety distance selected according to axes X and Z.



4.- Threading is done with radial penetration radial and by successive passes, until the full depth is reached. The depth of each pass will be a function of the corresponding pass number **Δ, Δ√2, Δ√3, Δ√4,.....**

Each of the threading passes is done in the following way:



> Fast positioning to the corresponding depth coordinate

> Threading of the programmed section, first according to axis Z up to the end of thread distance (**σ**) and then withdrawal threading up to the final coordinate.

During threading it is not possible to vary the feedrate (F) nor the spindle speed (S), and these values are kept 100% stable.

> Fast backwards to the approach point.

5.- Once the operation or cycle has ended, the tool will return to the position it occupied when the cycle was called upon, that is, the point where [I] was pressed.

Obviously, when executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.

6.- The CNC will stop the spindle but keep the machining conditions set for finishing selected: tool (T), axis feedrate (F) and spindle speed (S).

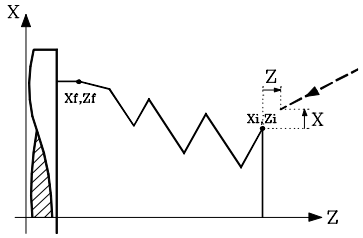
4.8.4 TAPER THREADING. BASIC OPERATION.

The machining passes in this cycle are as follows:

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

2.- The spindle starts at the speed selected and in the direction indicated.
Depending on the spindle turning direction, the thread will be clockwise or counterclockwise.

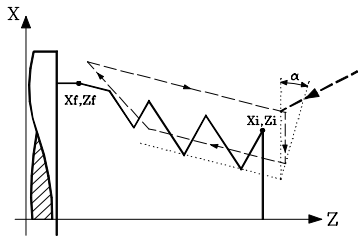
3.- The tool approaches at fast feedrate up to the starting point, keeping the safety distance selected according to axes X and Z.



4.- Threading is done is successive passes until the full depth is reached. The depth of each pass depends on the model selected.

a) as a function of the corresponding pass number **Δ, Δ√2, Δ√3, Δ√4, ...**
b) keeping the increment between passes constant **Δ**

Each one of the threading passes is done as follows:



> Fast positioning up to the corresponding depth coordinate.

This positioning is done according to the tool penetration angle (**α**) selected.

> Threading of the programmed section, first according to the profile defined up to the end of thread distance (**σ**) and then withdrawal threading up to the final coordinate.

During threading it is not possible to vary the feedrate (F) nor the spindle speed (S), and these values are kept 100% stable.

> Fast backwards to the approach point.

5.- The threading operation always ends at the approach point, that is, keeping the safety distance with respect to the starting point (Xi, Zi).

6.- The CNC will stop the spindle but keep the machining conditions set for finishing selected: tool (T), axis feedrate (F) and spindle speed (S).

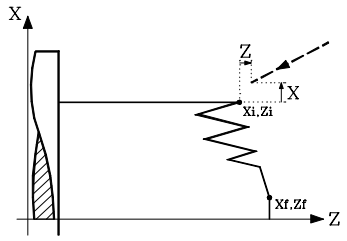
4.8.5 FACE THREADING. BASIC OPERATION

The machining passes in this cycle are as follows:

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

2.- The spindle starts at the speed selected and in the direction indicated.
Depending on the spindle turning direction, the thread will be clockwise or counterclockwise.

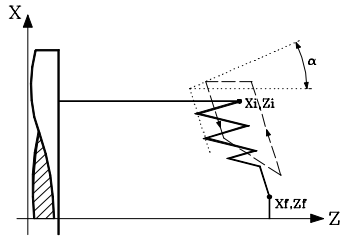
3.- The tool approaches at fast feedrate up to the starting point, keeping the safety distance selected according to axes X and Z.



4.- Threading is done in successive passes until full depth is reached. The depth of each pass depends on the model selected.

a) as a function of the corresponding pass number **Δ, Δ√2, Δ√3, Δ√4,.....**
b) keeping the increment between passes constant **Δ**

Each one of the threading passes is done as follows :



> Fast positioning up to the corresponding depth coordinate.

This positioning will be done according to the tool penetration angle (**α**) selected.

> Threading of the section programmed, first according to the profile defined up to the end of threa **(σ)** and then withdrawal threading up to the final coordinate.

During threading it is not possible to vary the feedrate (F) nor the spindle speed (S), and these values are kept 100% stable.

> Fast backwards to the approach point.

5.- Once the operation or cycle has ended, the tool will return to the position it occupied when the cycle was called upon, that is, the point where [I] was pressed.

Obviously, when executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.

6.- The CNC will stop the spindle but keep the machining conditions set for finishing selected: tool (T), axis feedrate (F) and spindle speed (S).

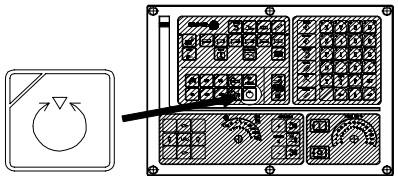
4.8.6 THREAD REPAIR. BASIC OPERATION

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 spindle position.

It is enough to orient the spindle once after power-up for the CNC to know the spindle position. Key:



With the spindle stopped, take the tool to be used for thread repair to one of the roots of the thread.

Once at this point, take these two values:

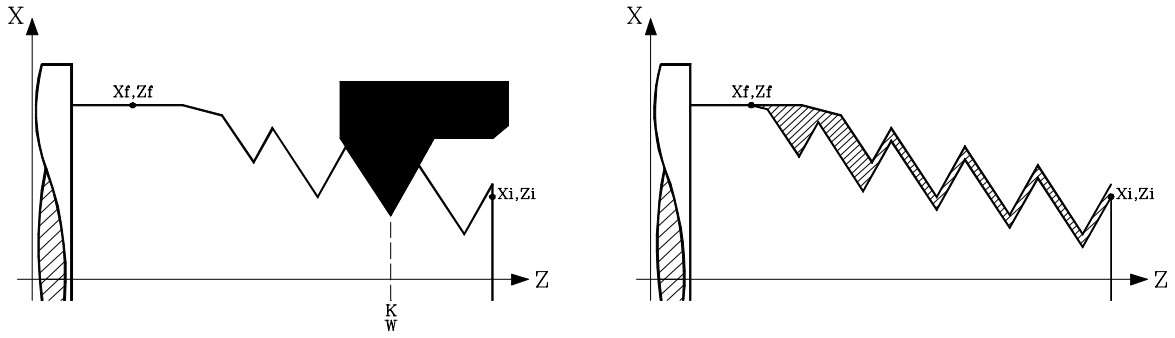
Z coordinate at the root. Position the cursor on the data and press RECALL ENTER

Spindle angular position at the root. Position the cursor on the data and press RECALL ENTER

The CNC assumes these two data necessary to carry out the thread repair.

The machining steps for this cycle are identical to those for taper threading described earlier.

The CNC makes a new thread on the existing thread, but maintaining the roots and inclinations of the current thread as shown in the figure.



Note: To perform the thread repair, proceed as follows:

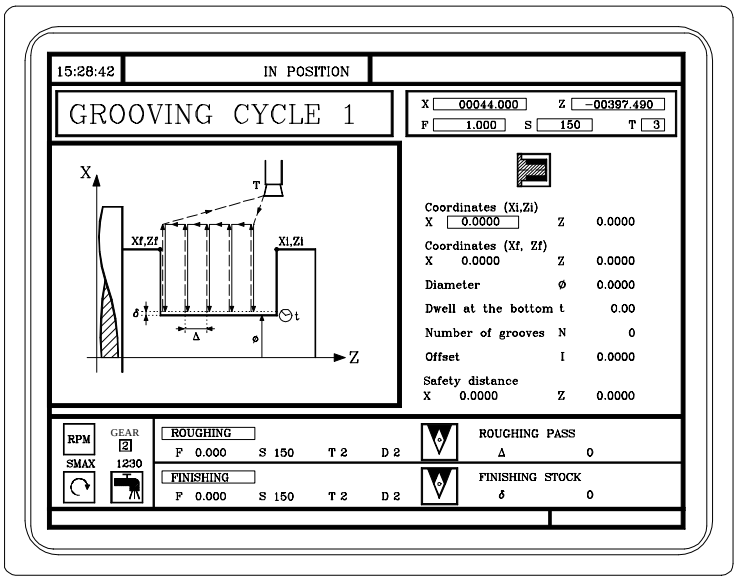
1- The spindle must have been referenced (homed - M19) at least once since the unit was turned on.
2- Take the Z coordinate values (teach-in) and the angular position of the spindle at the root of the thread, parameters K W, while the tool is positioned at one of the roots of the thread to be repaired.
3- Define the thread repair cycle.
4- Execute the cycle.

4.9 GROOVING CYCLE

To select grooving cycle press 

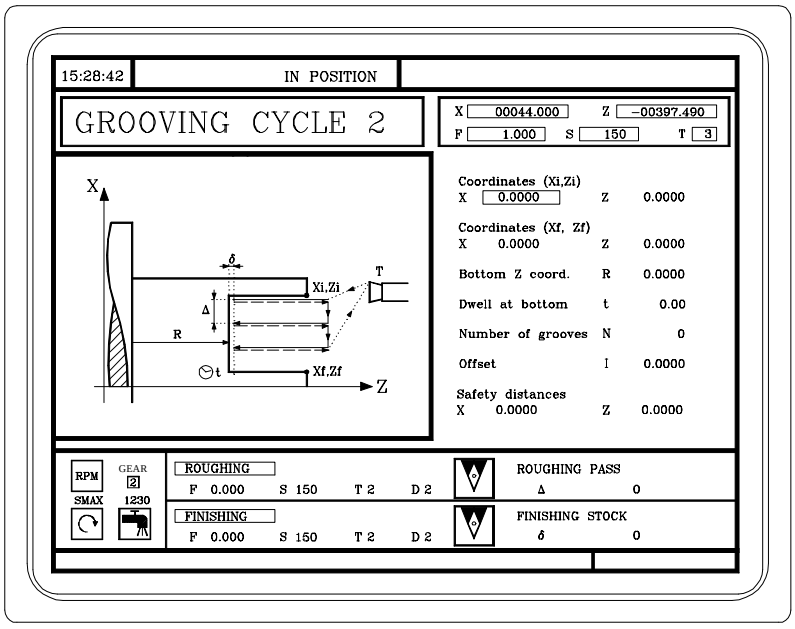
Cylindrical grooves and frontal grooves can be made with vertical and incline walls:

Level 1. Longitudinal grooving



One must define: The coordinates of the starting point and those of the end point
The final diameter
The dwell at the bottom
Number of grooves and Offset

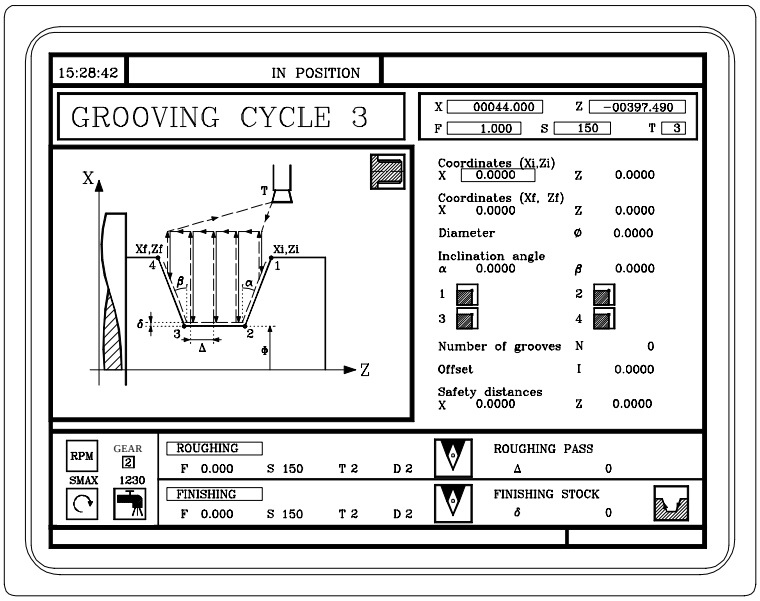
Level 2. Face grooving



Definition must be made of The coordinates of the starting point and of the end point
The coordinate of the groove bottom
The dwell at the bottom
Number of grooves and Offset

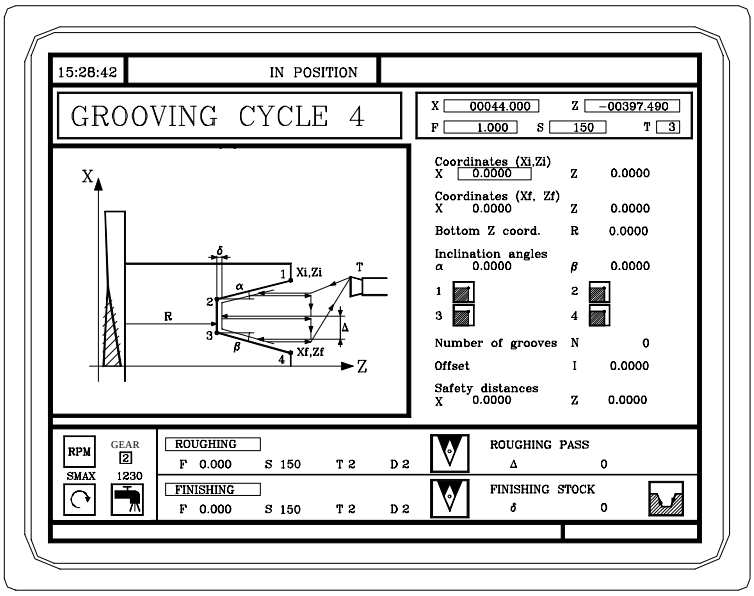
To change levels press 

Level 3. Longitudinal grooving with incline walls.



One must define: the coordinates of the starting and end points
the final diameter
the inclination angles of the incline walls
the number of grooves and Offset

Level 4. Face grooving with incline walls.



One must define: the coordinates of the starting and end points
the coordinate of the bottom of the groove
the inclination angles of the incline walls
the number of grooves and Offset

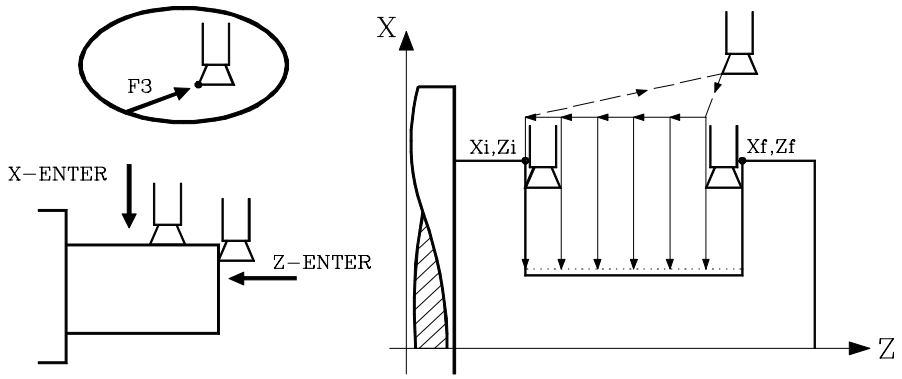
To change levels, press LEVEL CYCLE

4.9.1 CALIBRATION OF THE GROOVING TOOL

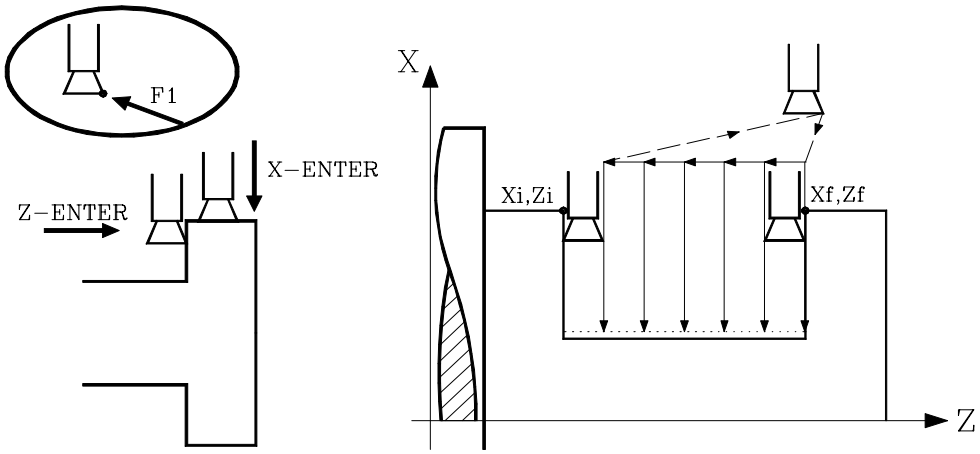
When calibrating the grooving tool proper indication should be made of the location codes for the corner that has been calibrated.

One same tool can thus be calibrated in three different ways, as shown below:

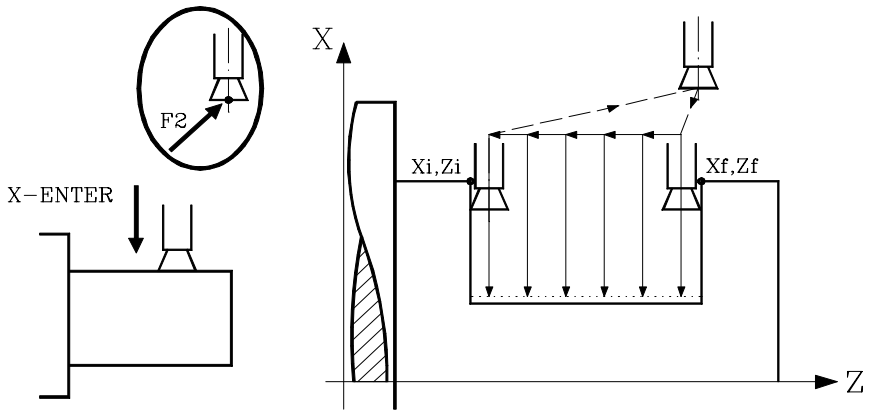
The left-hand corner of the cutter is calibrated. Location code F3



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



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



4.9.2 DEFINITION OF GEOMETRY

Type of grooving: internal or external

 External grooving  Internal grooving

To modify the type of grooving move onto this icon and press 

Whenever the type of threading is changed the CNC modifies the icon and shows the corresponding geometrical help screen.

Coordinates of the starting point (Xi, Zi) and coordinates of the end point (Xf, Zf)

The coordinates are defined one by one. After moving over the coordinates for the axis to be defined, one can:

a) Manually enter the value. Key in the required value and press ENTER

b) Assign the present position of the machine.

Move the axis by means of the handwheel or JOG keys, to the point required. The top right-hand window shows the position of the tool at all times.

Press RECALL for the data item selected to assume the value shown in the top right-hand window.

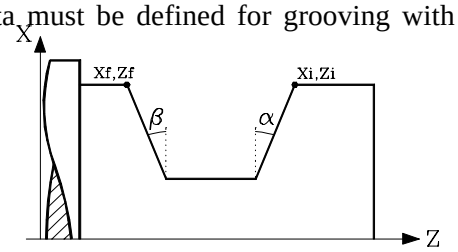
Press ENTER

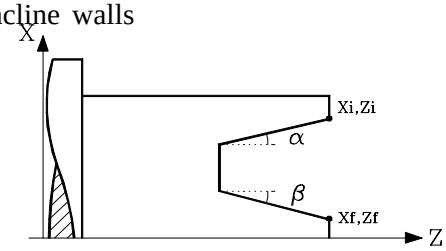
Final diameter (Φ) and coordinate of the bottom of the groove (R)

Depending on the type of grooving selected one of these two data items must be defined. To do this, move on to this data item, key in the required value and press ENTER

Inclination angles (α, β)

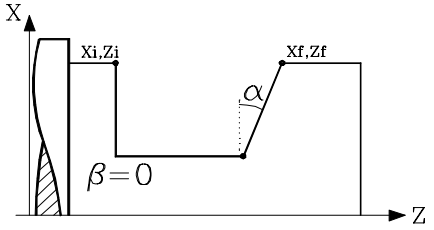
This data must be defined for grooving with incline walls

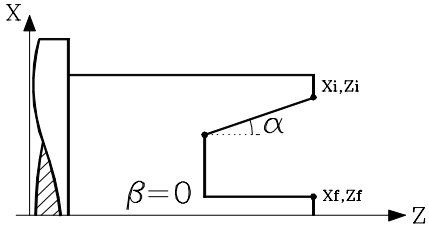




To do that, position over the data, key in the desired value and press ENTER

The next example shows grooves with α=20° and β=0°



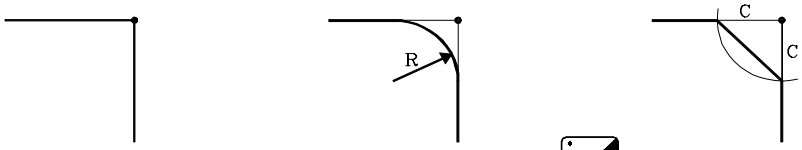


Type of machining to be carried out on each corner.

This data must be defined for grooving with incline walls.

The type of machining must be defined for all four corners of the groove.

A square corner.  Radius blend.  A chamfer. 



To change the type of machining, position over this icon and press 

For the rounded corner (radius blend), the rounding radius (R) must be defined and, for the chamfer, the distance from the theoretical corner to the chamfer point (C).

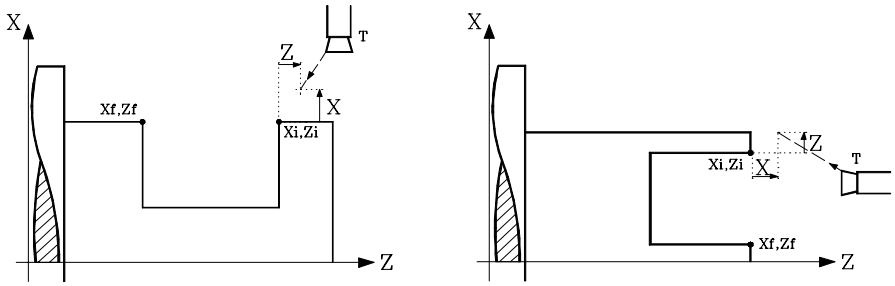
Dwell at the bottom (t)

Define the waiting time in seconds, after each deepening, until the withdrawal movement starts.

To define this, move on to the data item, key in the required value and press 

Safety distance

In order to prevent collisions with the part, the CNC allows a point of approach to the part to be defined. The safety distance indicates the position of the approach point with respect to the initial corner.

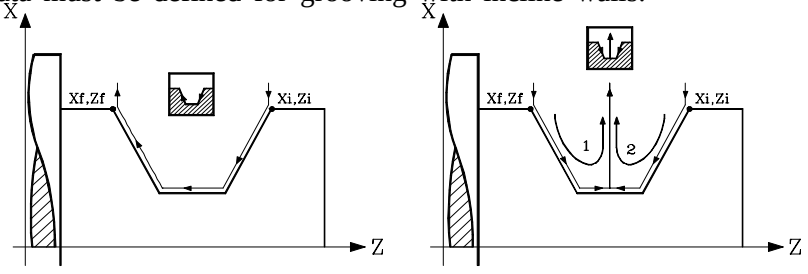


The value of the safety distance on X is always defined in radii.

To modify these values move over the relevant data, key in the required value and press 

Types of machining for the finishing pass 

This data must be defined for grooving with incline walls.

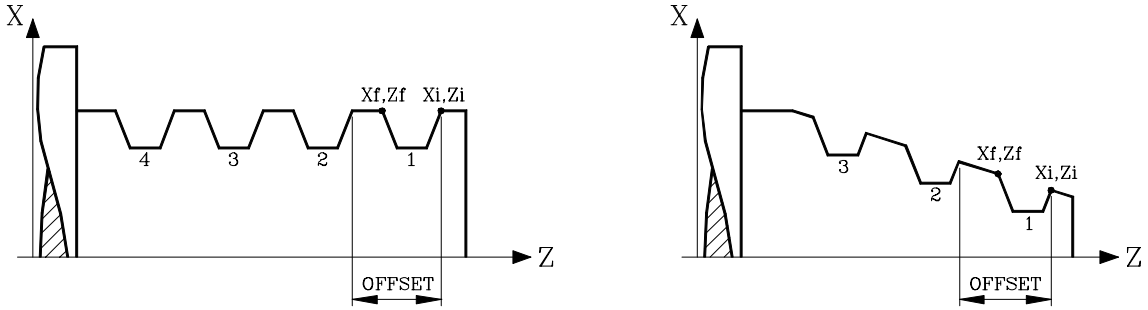


To change the type of machining, position over the icon and press 

Groove repetition

With the «Number of Grooves» and «Offset» data, the same groove may be repeated along the Z axis on longitudinal grooving or along the X axis on face grooving.

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



If «Number of grooves» is equal to "0" or "1", only one groove will be made.

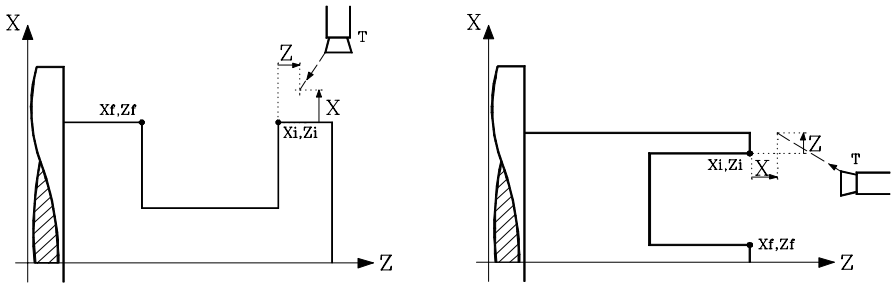
4.9.3 BASIC OPERATION

The machining passes in this cycle are as follows:

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

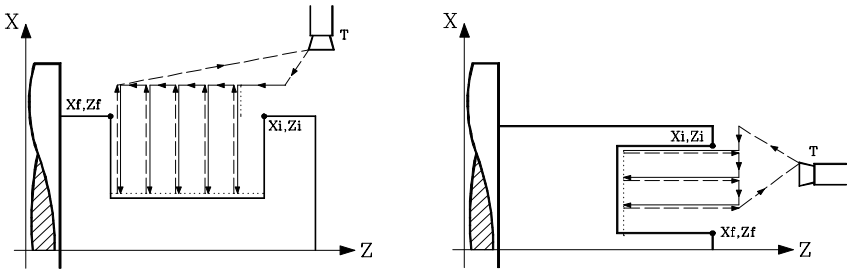
2.- The spindle starts at the speed selected and in the direction indicated.

3.- The tool approaches at fast feedrate to the starting point, (Xi, Zi) keeping the safety distance selected according to axes X and Z.



4.- Roughing Operation by means of successive grooving passes, up to a distance from the final depth selected equal to the finishing excess material.

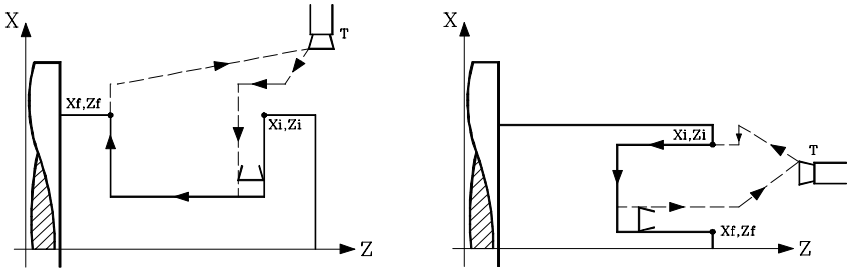
This operation is done with the conditions set for the roughing operation; nevertheless, the CNC calculates the real pass so that all the grooving passes are equal. This pass will be equal to or less than the one defined **Δ.**



5.- Finishing Operation

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

The part finishing is done with the machining conditions set for finishing; axis feed (F), spindle speed (S), turning direction.



6.- Once the operation or cycle has ended, the tool will return to the position it occupied when the cycle was called upon, that is, the point where [i] was pressed.

Obviously, when executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.

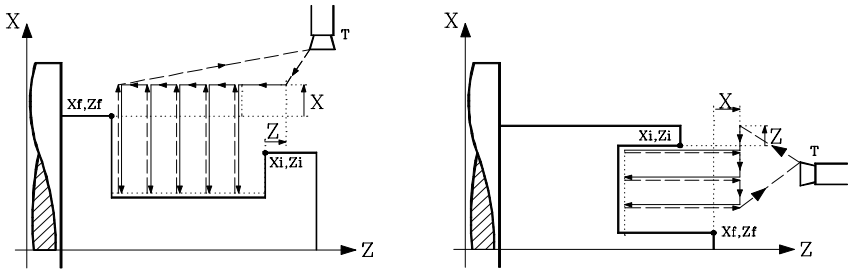
7.- The CNC will stop the spindle but keep the machining conditions set for finishing selected; tool (T), axis feed (F) and spindle speed (S).

Some points to consider:

If T0 is selected as roughing tool, the cycle does not execute the roughing operation. This means that after approaching the finishing operation is carried out.

If T0 is selected as finishing tool, the cycle does not execute the finishing operation. This means that after the roughing operation the tool moves to the approach point, keeping the safety distance with respect to the starting point (Xi, Zi).

When the surface required for machining is not totally cylindrical, the CNC analyzes the coordinates of the initial and end points and takes as starting point the outermost coordinate from the end depth.



4.10 DRILLING AND TAPPING CYCLES

To select the Drilling cycle and the Tapping cycle press 

Depending on the type of machine and how the CNC machine parameters have been set, up to 5 cycles magy be available:

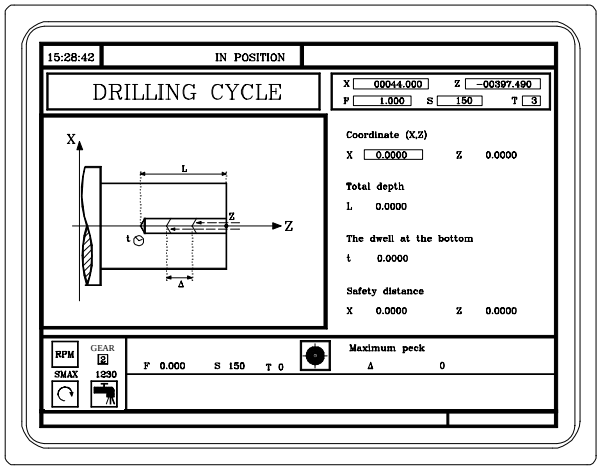
Drilling Cycle.	Multiple Drilling Cycle
Tapping Cycle.	Multiple Tapping Cycle
	Multiple Slot milling Cycle

The drilling and the tapping cycles are always available.
The other three cycles are only available when having live tool and spindle orientation.

Live tool	General parameters P0...P9 = 13
Spindle orientation	Main spindle parameter REFEED1 (P34) other than 0

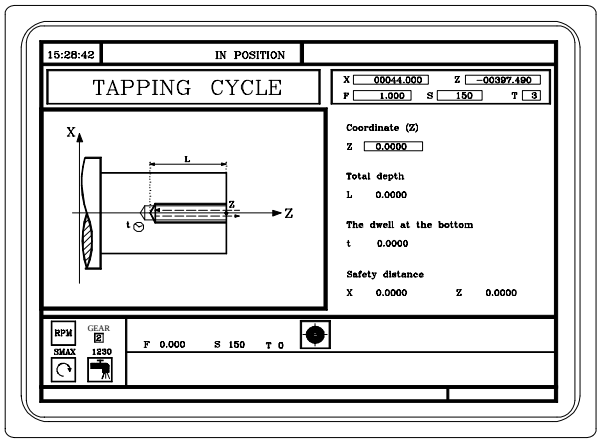
Every time LEVEL CYCLE is pressed, the CNC shows the next available cycle.

Level 1. Drilling cycle



Definition is needed for
- The coordinates of the drilling point
- The total depth
- The dwell at the bottom

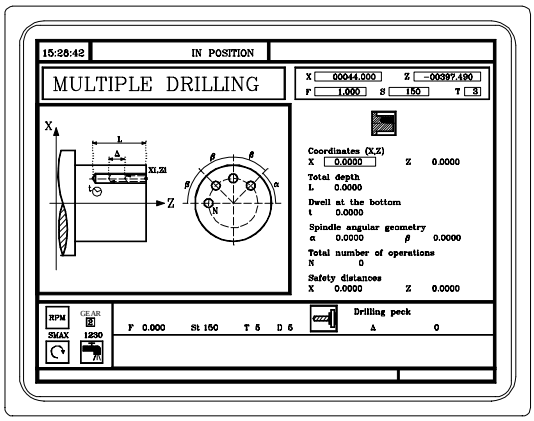
Level 2. Tapping cycle.

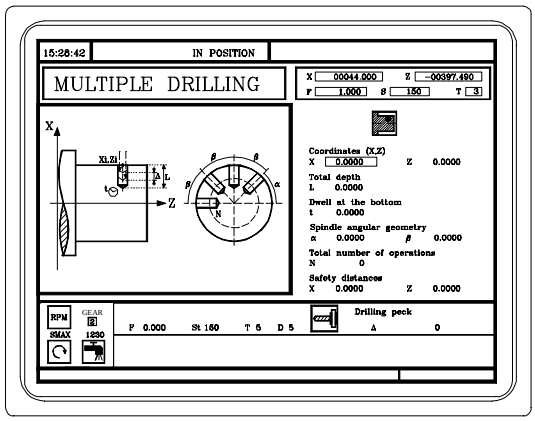


A definition is needed for:
- The Z coordinate of the tapping point
- The total depth
- The dwell at the bottom

To change levels press LEVEL CYCLE

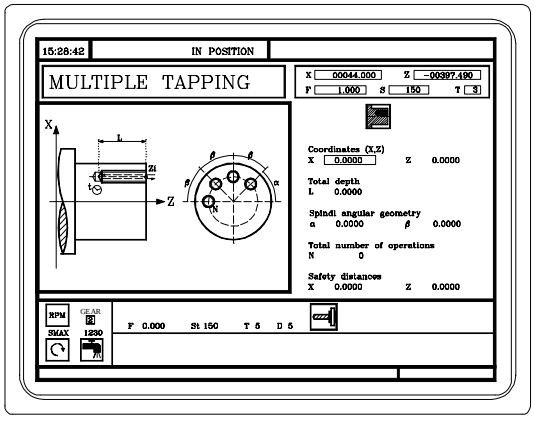
Level 3. Multiple drilling cycle.
Multiple drilling is possible on the side of the part or on its face

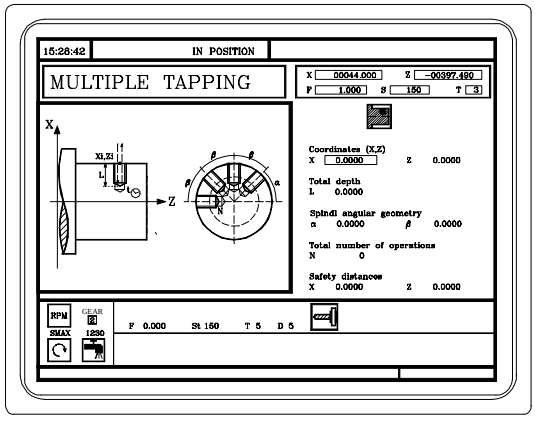




A definition is needed for: The coordinates of the first point — The total depth
The dwell at the bottom — The angular position of the holes
The total number of holes

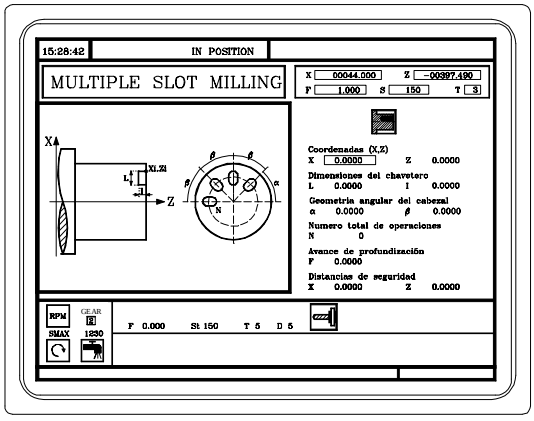
Level 4. Multiple tapping cycle.
Multiple tapping is possible on the side of the part and on its face.

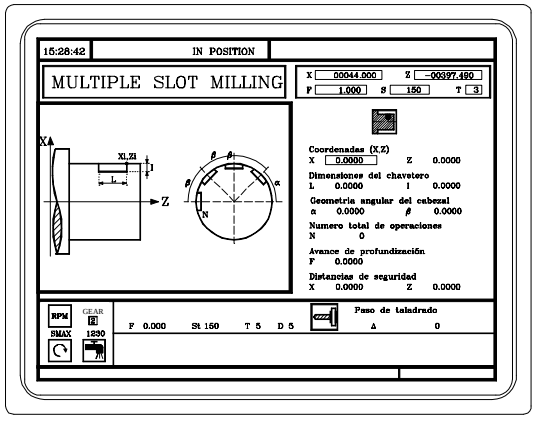




A definition is needed for: The coordinates of the first point — The total depth
The angular positions of the holes — The total number of holes

Level 5. Multiple slot milling Cycle.
Multiple slots may be milled out on the side of the part or on its face.





A definition is needed for The coordinates of the first point — The dimensions of the slot
The angular position of the slots — The total number of slots

4.10.1 DEFINITION OF GEOMETRY

Machining on the face or side of the part:

 Machinig on the face of the part  Machining on the side of the part:

To change the type of machining, position over this icon and press 

Every time the type of tapping is changed, the CNC modifies the icon and displays the corresponding geometry aide on the screen.

Coordinates of the starting point (X, Z)

The coordinates are defined one by one. After moving over the coordinates for the axis to be defined, one can:

a) Manually enter the value. Key in the required value and press 

b) Assign the present position of the machine.

Move the axis by means of the handwheel or the JOG key up to the point required. The top right-hand window shows the tool position at all times.

Press  for the data item selected to assume the value shown in the top-right hand window

Press 

Tapping must always be axial, in the rotation center (X0).

Though the drilling is normally done in the rotation center, the CNC enables X to be defined with a value different to X0 and make grooves in the front face of the part.

Total depth (L)

Move onto this data item, key in the required value and press 

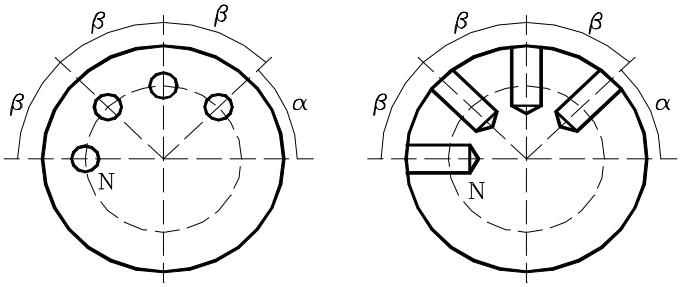
Dwell at the bottom (t)

Define the waiting time in seconds, after each deepening, until the withdrawal movement starts.

To define this, move onto this data item, key in the required value and press 

Angular positions (a, b)

This data indicates the angular position of the first machining operation and β indicates the angular increment between machining operations.



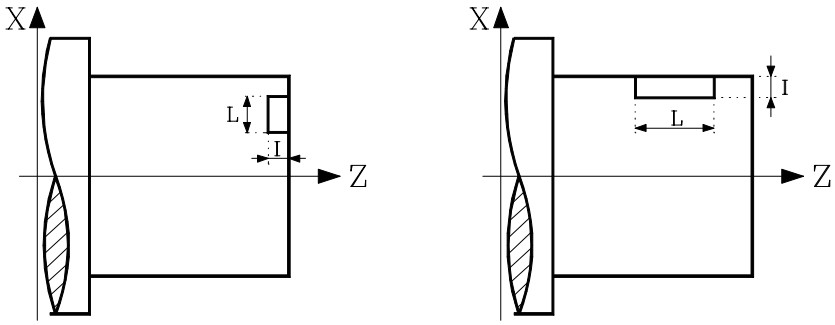
To defined them, position over the corresponding data (α or β), key in the value and press 

Number of operations (N)

Position 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.



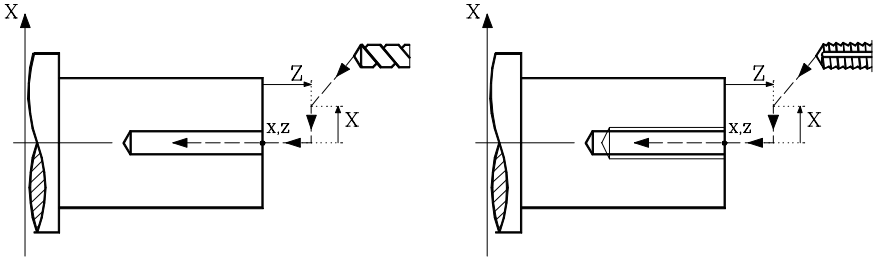
To define them, position over the corresponding data (L or I), key in the value and press ENTER.

Penetration feedrate (F)

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

Safety distance

In order to prevent collisions with the part, the CNC allows an approach point towards the part to be defined. The Safety distance indicates the position of the approach point with respect to the drilling or threading point.



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

To modify one of these values move onto the relevant data item, key in the required value and press ENTER.

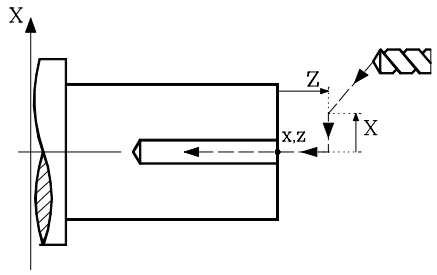
4.10.2 BASIC OPERATION. DRILLING CYCLE

The machining passes in this cycle are as follows:

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

2.- The spindle starts at the speed selected and in the direction indicated.

3.- The tool approaches at fast feedrate up to the starting point, keeping the safety distance selected according to axes X and Z.



4.- Drilling loop. The following steps are repeated with the amount D being deepened each time until depth L is reached.

> Fast approach up to 1 mm before the previous deepening.

> Drilling up to the next deepening.

> Fast withdrawal to the approach point.

5.- Delay time **t** at the bottom of the drilling.

6.- Fast withdrawal to the approach point.

7.- Once the operation or cycle has ended, the tool will return to the position it occupied when the cycle was called upon, that is, the point where [i] was pressed.

Obviously, when executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.

8.- The CNC will stop the spindle but keeps the machining conditions set for the drilling selected; tool (T), axis feedrate (F) and spindle speed (S).

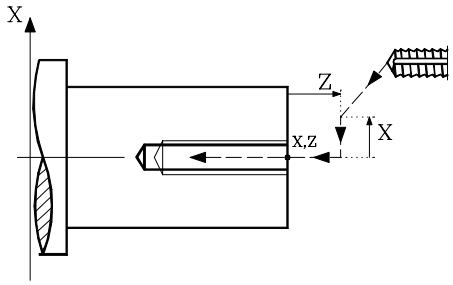
4.10.3 BASIC OPERATION. TAPPING CYCLE

The machining passes in this cycle are as follows:

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

2.- The spindle starts at the speed selected and in the direction indicated.

3.- The tool approaches at fast feedrate up to the starting point, keeping the safety distance selected according to axes X and Z.



4.- Tapping of the part in work feedrate F, until the depth L is reached.

5.- Reversal of the spindle turning direction.

If a dwell time at the bottom has been defined, the spindle stops, and after this time the spindle starts in the other direction.

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 the cycle was called upon, that is, the point where [I] was pressed.

Obviously, when executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.

8.- The CNC will stop the spindle but keeps the machining conditions set for the tapping selected; tool (T), axis feedrate (F) and spindle speed (S).

4.10.4 BASIC OPERATION. MULTIPLE DRILLING CYCLE

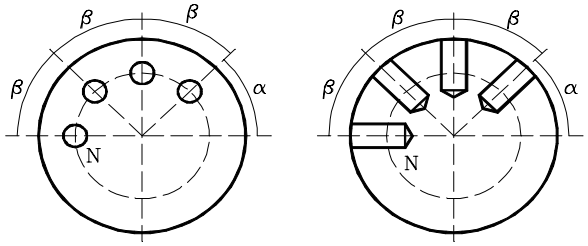
The machining steps for this cycle are:

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

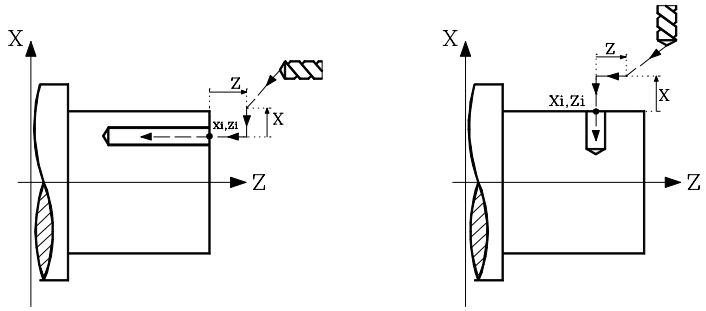
2.- If the operation was programmed with another tool, the CNC performs a tool change moving the axes to the tool change position if so required by the machine.

3.- Turns the live tool at the indicated rpm.

4.- Orients the spindle to the first angular position (indicated by α)



5.- The tool approaches in rapid to the first point staying at the selected safety distance along the X and Z axes.



6.- Drilling loop. The following steps are repeated by penetrating an amount Δ at each peck until the total depth "L" is reached.

> Rapid approach up to 1 mm (0.03937 inch) from the previous peck.
> Drill up to next pecking depth.
> Rapid withdraw to the approach point.

7.- Dwell "t" at the bottom of the hole.

8.- Rapid withdraw to the approach point.

9.- Depending on the value assigned to parameter N (number of holes).

> The spindle turns (orients) to the next hole position (angular increment β)
> Repeats drilling steps 6, 7 and 8

10.- Once the operation or cycle has ended, the tool will return to the position it occupied when the cycle was called upon, that is, the point where [I] was pressed.

Obviously, when executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.

11.- The CNC stops the live tool but maintains the selected machining conditions, tool (T), Feedrate (F) and spindle speed value (S).

4.10.5 BASIC OPERATION. MULTIPLE TAPPING CYCLE

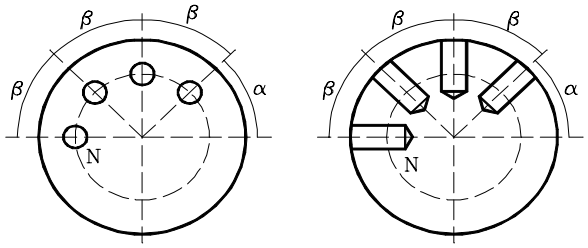
The machining steps for this cycle are:

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

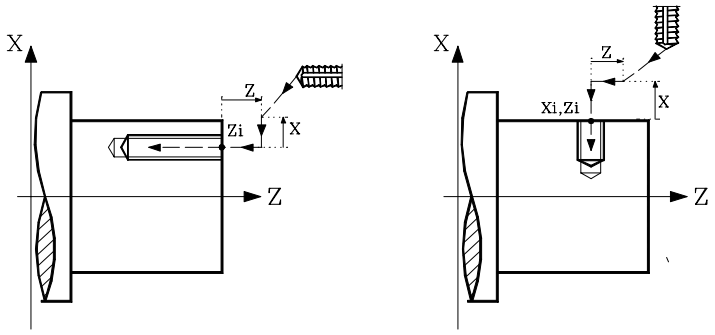
2.- If the operation was programmed with another tool, the CNC performs a tool change moving the axes to the tool change position if so required by the machine.

3.- Turns the live tool at the indicated rpm.

4.- Orients the spindle to the first angular position (indicated by α)



5.- The tool approaches in rapid to the first point staying at the selected safety distance along the X and Z axes.



6.- It taps the hole at the feedrate F, until the depth "L" is reached.

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

8.- Withdrawal at "F" up to the approach point.

9.- Depending on the value assigned to parameter N (number of holes)

> The spindle turns (orients) to the next hole position (angular increment β)
> Repeats drilling steps 6, 7 and 8

10.- Once the operation or cycle has ended, the tool will return to the position it occupied when the cycle was called upon, that is, the point where [I] was pressed.

Obviously, when executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.

11.- The CNC stops the live tool but maintains the selected machining conditions, tool (T), Feedrate (F) and spindle speed value (S).

4.10.6 BASIC OPERATION. MULTIPLE SLOT MILLING CYCLE

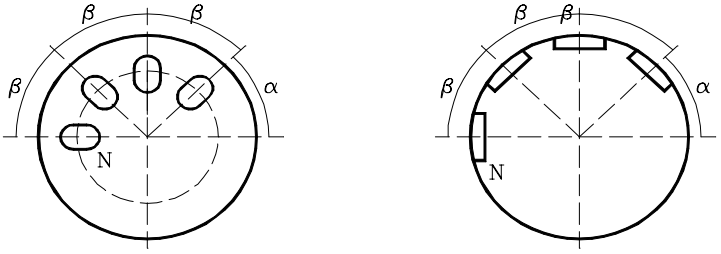
The machining steps for this cycle are:

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

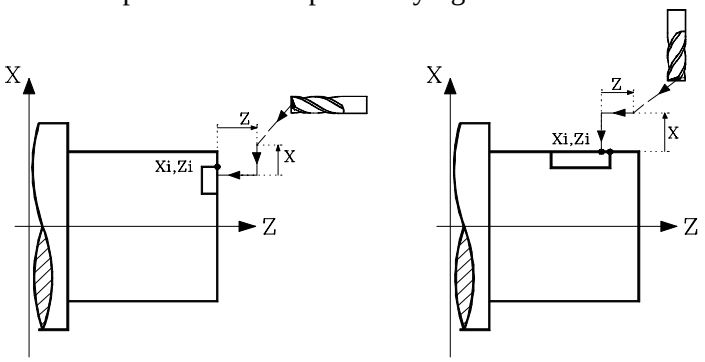
2.- If the operation was programmed with another tool, the CNC performs a tool change moving the axes to the tool change position if so required by the machine.

3.- Turns the live tool at the indicated rpm.

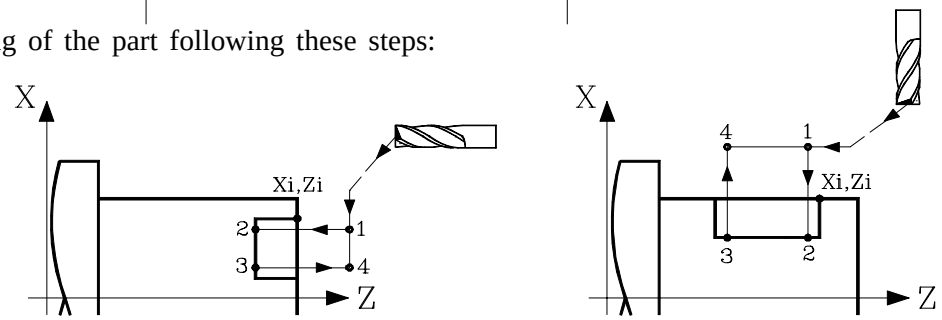
4.- Orients the spindle to the first angular position (indicated by α)



5.- The tool approaches in rapid to the first point staying at the selected safety distance along the X and Z axes.



6.- Milling of the part following these steps:



> Penetration at the programmed "F" all the way to the bottom of the slot (section 1-2)
> Makes the slot by moving the X or the Z axis (as it corresponds) at the programmed feedrate "F" (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 spindle orients to the next point (angular increment β)
> Mills another slot as described in paragraph 6.

8.- Once the operation or cycle has ended, the tool will return to the position it occupied when the cycle was called upon, that is, the point where [I] was pressed.

Obviously, when executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.

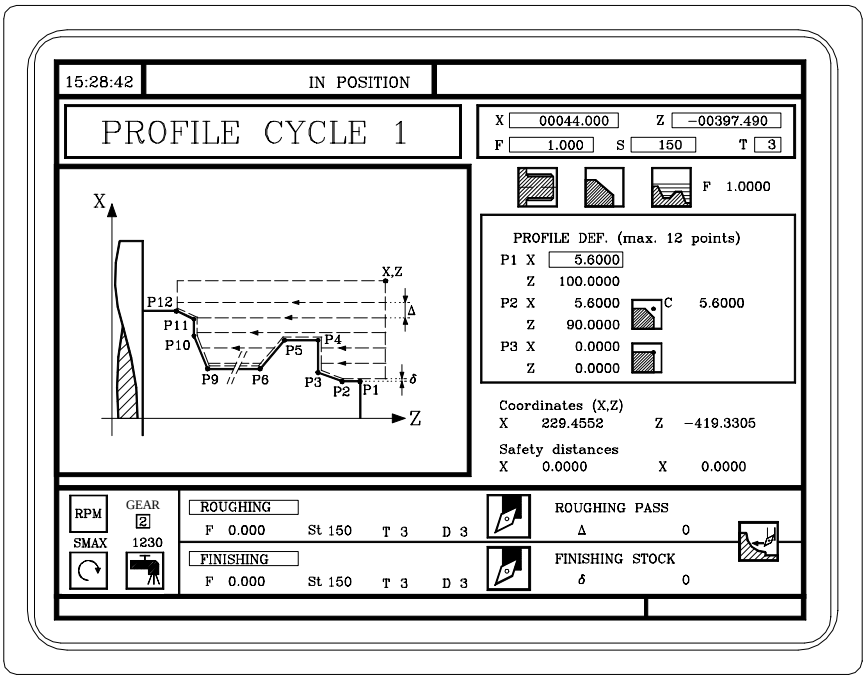
9.- The CNC stops the live tool but maintains the selected machining conditions, tool (T), Feedrate (F) and spindle speed value (S).

4.11 PROFILE CYCLE

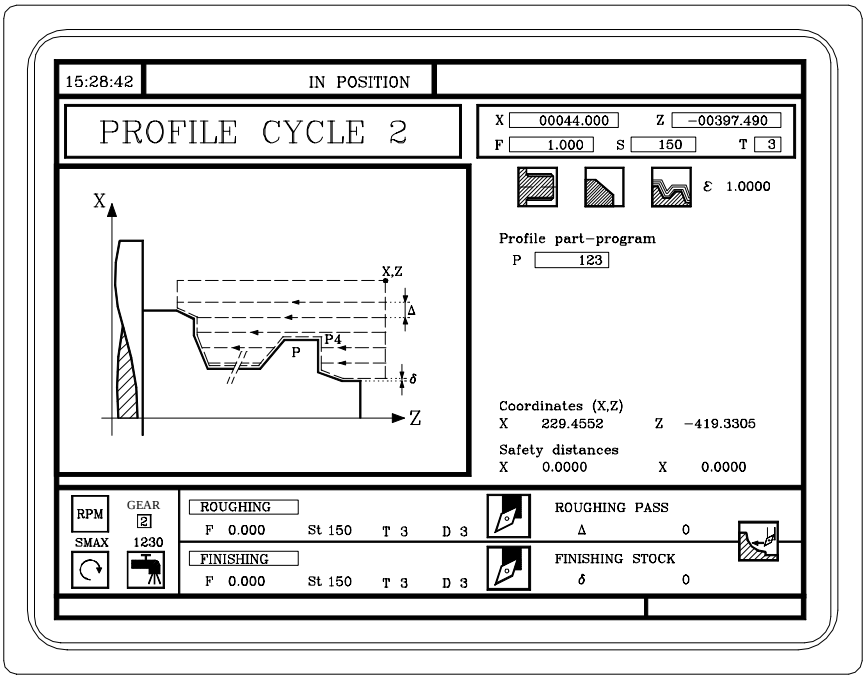
To select the profile cycle press 

This cycle can be defined in four ways:

Level 1. Defining all the points of the profile.

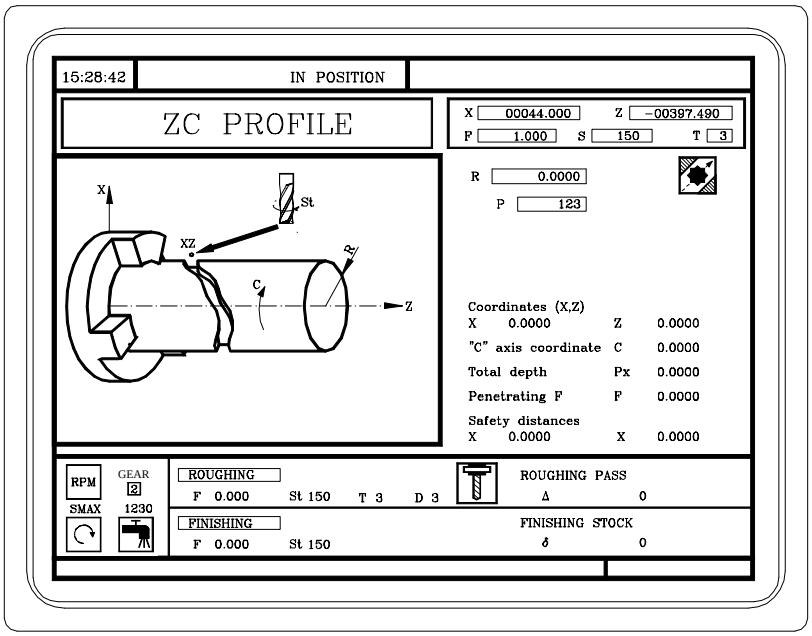


Level 2. Using a part-program which contains the profile.

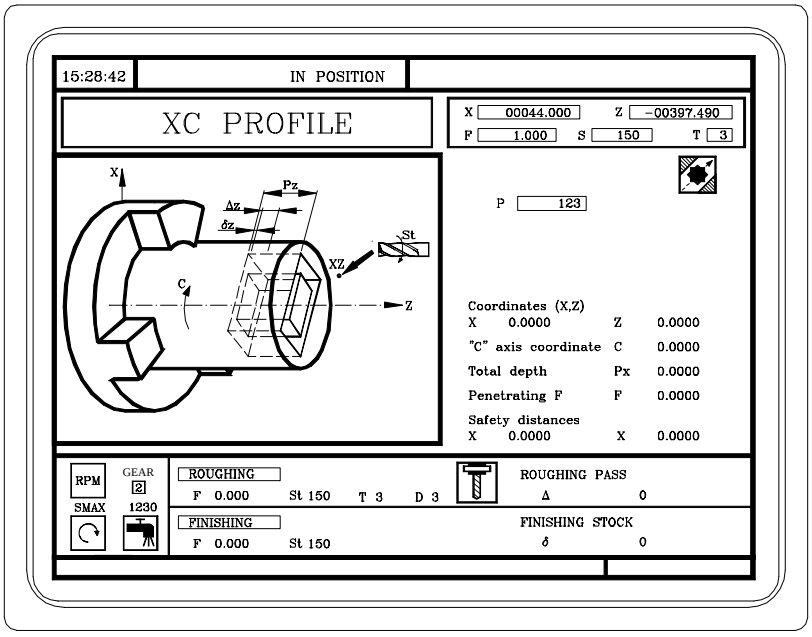


To change level press 

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



Level 4. XC Profile. Available when having a "C" axis.

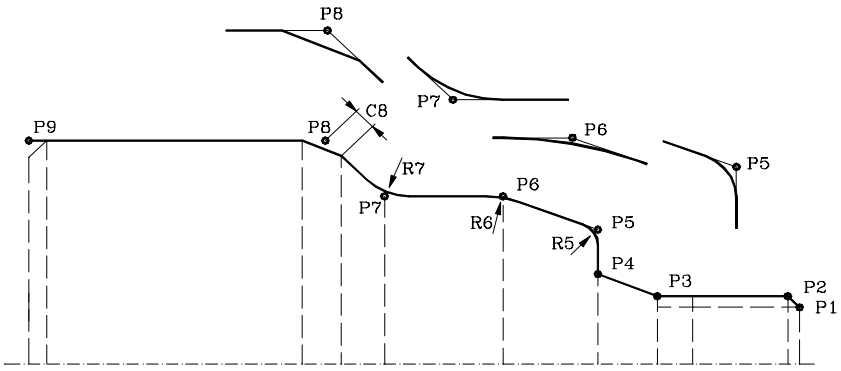


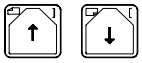
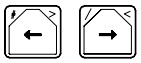
To change levels, press LEVEL CYCLE

4.11.1 LEVEL 1. PROFILE DEFINITION

This mode allows the profile to be defined by means of the description of its theoretical corners.

Up to 12 points can be used for defining said corners. Point P1 is the profile starting point. The remaining points must be correlative.



Keys  should be used for selecting and leaving the window containing the profile definition points and keys  for defining said points.

The coordinates for each point are defined one by one. For this purpose, after moving onto the coordinate required for definition one can:

a) Manually enter the value. Key in the required value and press 

b) Assign the present position of the machine.

Move the axis, by means of the handwheel or the JOG keys, up to the point required. The top right-hand window displays the tool position at all times.

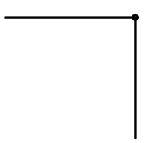
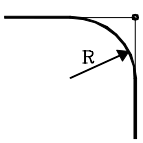
Press  for the data item selected to assume the value displayed in the top right-hand window

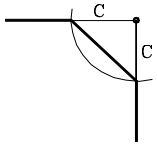
Press 

At all the intermediate points of the profile the type of machining required to be done on the corner must be defined.

A square corner.  A rounding. 

A chamfer. 



To modify the type of machining move onto this icon and press 

In the case of the rounded corner the rounding radius (R) has to be defined and in the case of the chamfer the distance from the theoretical corner up to the point where the chamfer is to be made (C).

When the 12 definition points are not used the following conditions have to be complied with:

* The CNC does not take into account the type of machining of the last profile point.

* The first point not used must be defined with the same coordinates as the last point of the profile. In the example shown in the figure above P10 = P9 must be defined.

4.11.2 LEVELS 2, 3 AND 4. PROFILE DEFINITION

To define the «Profile Program» move onto the "Profile part-program" or "P" window.

After selecting this window one can:

Key in the required «Profile Program» required.

If the "Profile Program" is already known, key in the number and press ENTER

Access the «Profile programs directory».

Press ↓ . The cycle will display a window with the currently defined profile programs at the selected level.

To move up and down in this window use the ↑ ↓ keys.

Position the cursor over the desired program and press ENTER

To exit this window without selecting a program, use the ← → keys.

Edit a new «Profile Program».

To edit a new "Program", key in the program number (between 0 and 999) and press RECALL
The CNC will display the window for the profile editor (to see how to use it, see the CNC 8055T, Operation manual, chapter 4 «Editing», in the «Profile Editor» section.

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 to added, press ESC

Modify an already existing «Profile program».

To edit an existing «Profile program» key in the program number and press RECALL
The CNC will display the profile currently defined in the profile editor window

One can:
- Add new items at the end of the present profile.
- Modify any item.
- Modify or include chamfers, rounded corners, etc..
- Erase elements of the profile.

An intermediary element of the profile cannot be erased. To erase it, all the elements have to be erased one by one from the last one defined, until the one required is reached.

Delete an existing "Profile Program"

Position the cursor over the program number and press CLEAR The CNC will request confirmation.

Notes: The profile programs can be accessed in the "T" mode because the CNC stores it internally as:

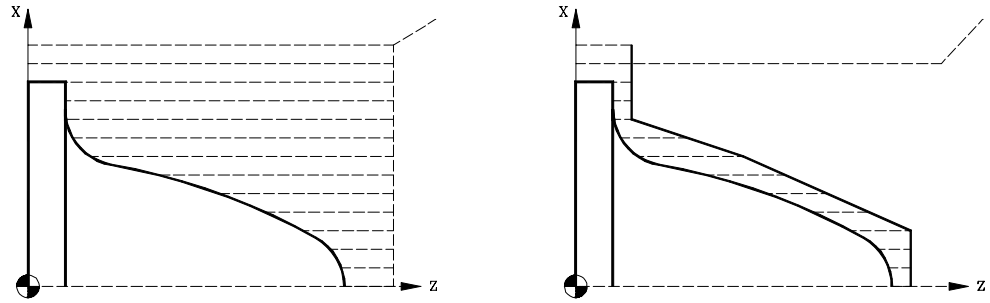
P998xxx (ZX profile, level 2) profile program number 11 is stored as P998011
P997xxx (ZC profile, level 3) profile program number 22 is stored as P997022
P996xxx (XC profile, level 4) profile program number 33 is stored as P996033

This chapter shows several profile editing examples later on.

When saving a part-program containing a profile cycle into an external device, its associated profile cycle (P998xxx, P997xxx, P996xxx) must also be saved.

4.11.3 LEVEL 2. OPTIMIZING OF THE MACHINING OF A PROFILE

When defining the desired profile only, the CNC assumes that the rough part is cylindrical and it machines it as indicated on the left diagram.



When the part profile is known, it is recommended to define both profiles, that of the rough part and the desired final profile. It will be machined faster since it only removes the material delimited by the two profiles.

To define both profiles, proceed as follows:

1 Access the profile editor.
2 Edit the desired final profile.
3 Press the "New Profile" softkey
4 Edit the rough part profile
5 Exit the profile editor saving the profile.

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

4.11.4 GEOMETRY DEFINITION. LEVELS 1, 2. ZX PROFILE

Internal or external profile

 External profile

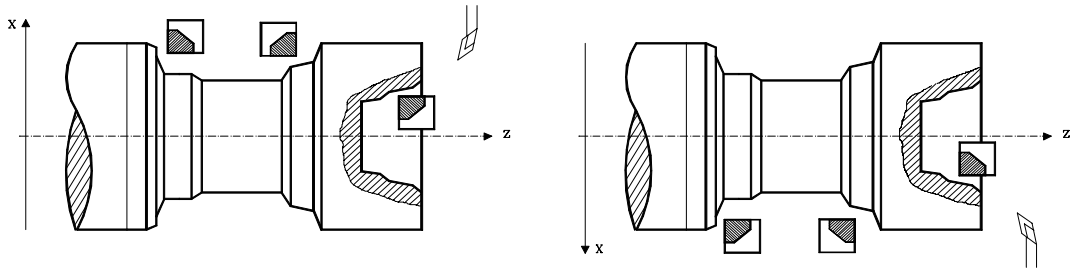
 Internal profile.

To modify the type of profile move onto this icon and press 

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

Work quadrant

The icon  defines the type of corner to be machined.



To modify the work quadrant move onto the icon and press  The CNC will display another icon.

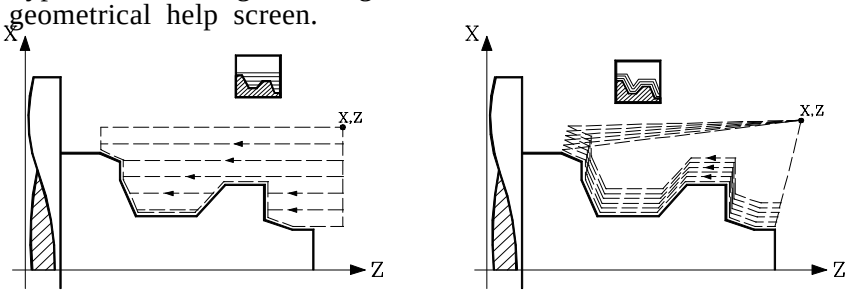
Type of machining

Paraxial .....  F 1.0000

 ε 1.0000

Pattern repeat .

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



To modify the type of machining move onto the icon and press 

In paraxial machining (one axis at a time), the tool penetration feedrate must be defined for the "valleys". The machining feedrate will be the one indicated in the window for roughing and finishing.

In pattern repeat (following the profile) the amount of material required to be removed from the origin part (**ε**) must be defined. Said value is defined in radii.

Coordinates of the starting point (X, Z)

The coordinates are defined one by one. After moving onto the coordinates for the axis to be defined, one can:

a) Manually enter the value. Key in the required value and press ENTER

b) Assign the present position of the machine.

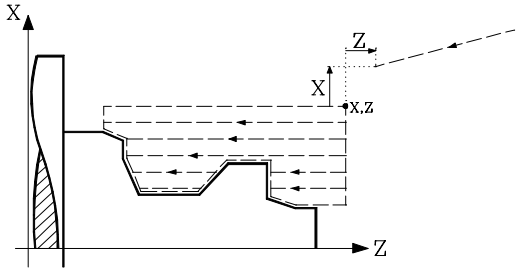
Move the axis by means of the handwheel or the JOG keys up to the point required. The top right-hand window displays the tool position at all times.

Press RECALL for the data item selected to assume the value displayed in the top right-hand window

Press ENTER

Safety distance

In order to prevent collisions with the part, the CNC allows an approach point to the part to be set. The Safety distance indicates the position of the approach point with respect to the initial corner.

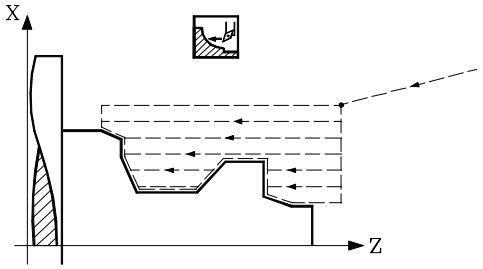


To modify one of these values move onto the corresponding data item, key in the required value and press ENTER

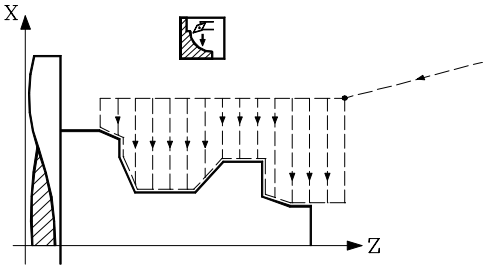
Machining direction

To select the machining direction (turning direction or facing direction), move onto the icon for the Roughing and Finishing data zone and press

The icon changes and the help graphic is updated.



Turning direction



Facing direction

4.11.5 GEOMETRY DEFINITION. LEVELS 3, 4. XC, ZC PROFILES

Milling with or without tool radius compensation

 Without compensation

 With left-hand tool radius compensation.

 With right-hand tool compensation.

To change the type of compensation, place the cursor over this icon and press 

Every time the type of profile is changed, the CNC changes the icon and displays the corresponding geometry assistance screen.

Radius

Indicates the outside radius of the part. Place the cursor over this item, key in the desired value and press 

Coordinates of the starting point (X, Z, C)

These coordinates are defined one by one. Place the cursor over the coordinates of the axis to be defined. One then may:

a) Enter the value manually. Key in the desired value and press 

b) Assign the current machine position.

Move the axis with the handwheel or the JOG keys to the desired point. The upper right-hand window shows the tool position at all times.

Press 

for the selected data to assume the value appearing in the upper right-hand window.

Press 

Total depth (Px)

The total depth of the profile is programmed with a positive value and in radius (ZC profile).

To define that value, place the cursor over this item, key in the desired value and press 

Penetrating feedrate (F)

Place the cursor over this item, key in the desired value and press

Safety distance

In order to avoid colliding into the part, an approach point may be set. The safety distance indicates the position of the approach point referred to the starting point.

To modify any of these values, place the cursor over the corresponding item, key in the desired value and press 

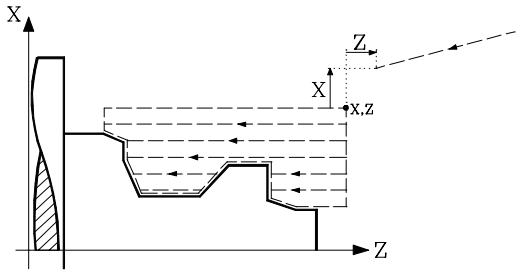
4.11.6 BASIC OPERATION. LEVELS 1,2. ZX PROFILE

The machining steps in these cycles are the following:

1.- If the roughing operation was programmed with another tool the CNC will make a tool change, moving to the change point if the machine requires this.

2.- The spindle starts at the speed selected and in the direction indicated.

3.- The tool approaches at fast feedrate up to the starting point, (X, Z) keeping the safety distance selected as per axes X and Z.



4.- Roughing operation, by means of successive passes, up to a distance from the profile equal to the finishing stock.

This operation is done in the conditions set for the roughing operation.

5.- Finishing operation

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

The part finishing is done in the machining conditions set for finishing: axis feedrate (F), spindle speed (S), tool (T).

6.- Once the operation or cycle has ended, the tool will return to the position it occupied when the cycle was called upon, that is, the point where [I] was pressed.

Obviously, when executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.

7.- The CNC will stop the spindle but keep the machining conditions set for finishing; tool (T), axis feedrate (F) and spindle speed (S).

Some points to consider:

If T0 is selected as roughing tool the cycle does not carry out the roughing operation. That is, after approach the finishing operation is done.

If T0 is selected as finishing tool the cycle does not carry out the finishing operation. This means that after the roughing operation the tool moves to the approach point keeping the safety distance with respect to the starting point (X, Z).

4.11.7 BASIC OPERATION. LEVELS 3,4. XC, ZC PROFILES

The machining steps in these cycles are the following:

1.- If the roughing operation was programmed with another tool the CNC will make a tool change, moving to the change point if the machine requires this.

2.- The tool approaches at fast feedrate 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, by means of successive passes, up to a distance from the profile equal to the finishing stock.

This operation is done in the conditions set for the roughing operation.

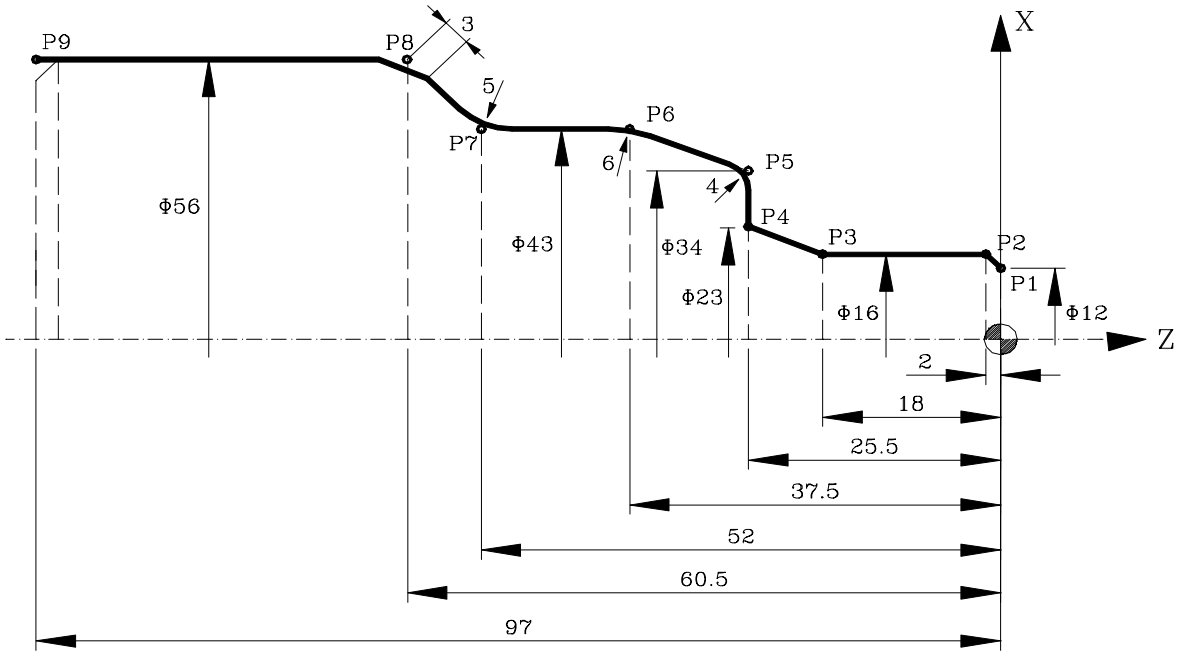
5.- Finishing operation

The part finishing is done in the machining conditions set for finishing: axis feedrate (F), live tool speed (St).

6.- Once the operation or cycle has ended, the tool will return to the position it occupied when the cycle was called upon, that is, the point where [i] was pressed.

Obviously, when executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.

4.11.8 LEVEL 1. EXAMPLE



Geometry definition

External profile  Working quadrant  Type of machining 

Profile definition

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

Coordinates (X, Z) X 80.0000 Z 10.0000

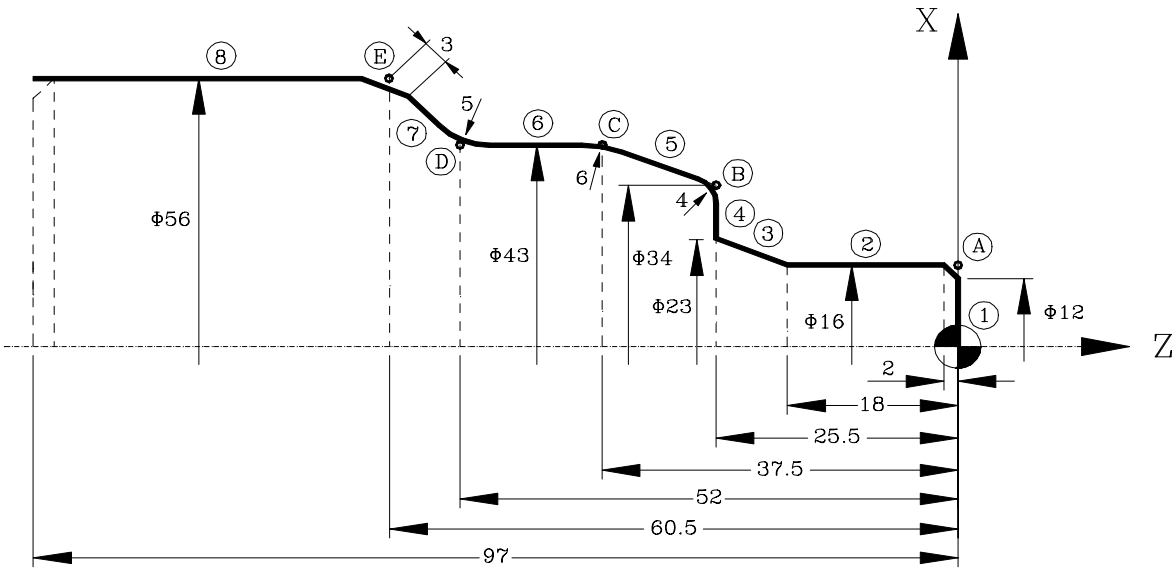
Safety distance X 0.0000 Z 0.0000

Roughing F 1.000 S 1000 T 3 Δ 2 

Finishing F 0.800 S 1000 T 3 δ 0.25

Spindle RPM 

4.11.9 LEVEL 2. EXAMPLES



Geometry definition

External profile  Work quadrant  Type of machining 

Profile definition

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

Modify

Chamfer	Select point "A"	Press ENTER	and assign this Radius = 2
Rounding	Select point "B"	Press ENTER	and assign this Radius = 4
Rounding	Select point "C"	Press ENTER	and assign this Radius = 6
Rounding	Select point "D"	Press ENTER	and assign this Radius = 5
Chamfer	Select point "E"	Press ENTER	and assign this Radius = 3

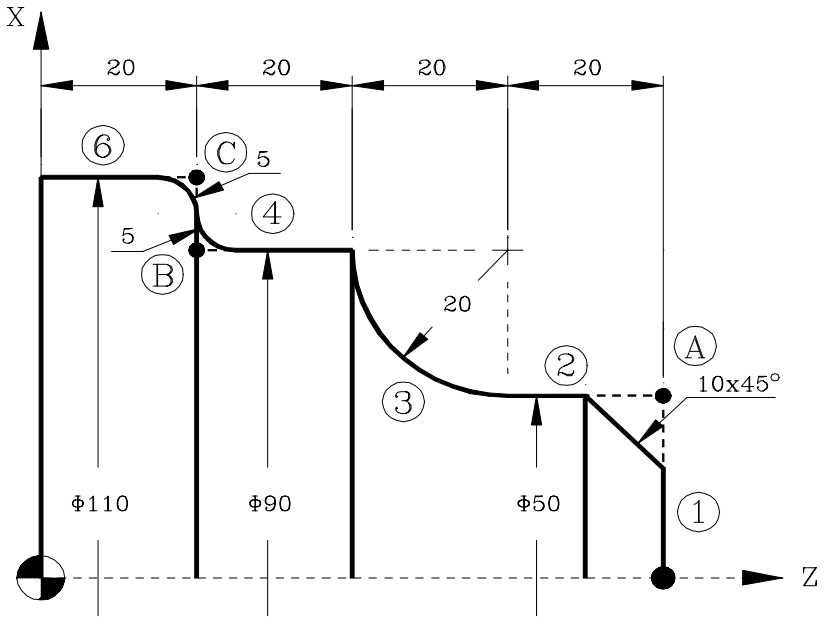
Coordinates (X, Z) X 65.0000 Z 10.0000

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

External profile  Work quadrant  Type of machining 

Profile definition

Abscissa and ordinate of the starting point		Z = 80	X = 0
Section 1	Straight	Z = 80	X = 50
Section 2	Straight	Z = 60	X = 50
Section 3	Clockwise circular pattern	Z = 40	X = 90 Z_{center}=60 X_{center}=90 Radius=20
Section 4	Straight	Z = 20	X = 90
Section 5	Straight	Z = 20	X = 110
Section 6	Straight	Z = 0	X = 110

Modify

Chamfer	Select point “A”	Press ENTER	and assign this Radius = 10
Rounding	Select point “B”	Press ENTER	and assign this Radius = 5
Rounding	Select point “C”	Press ENTER	and assign this Radius = 5

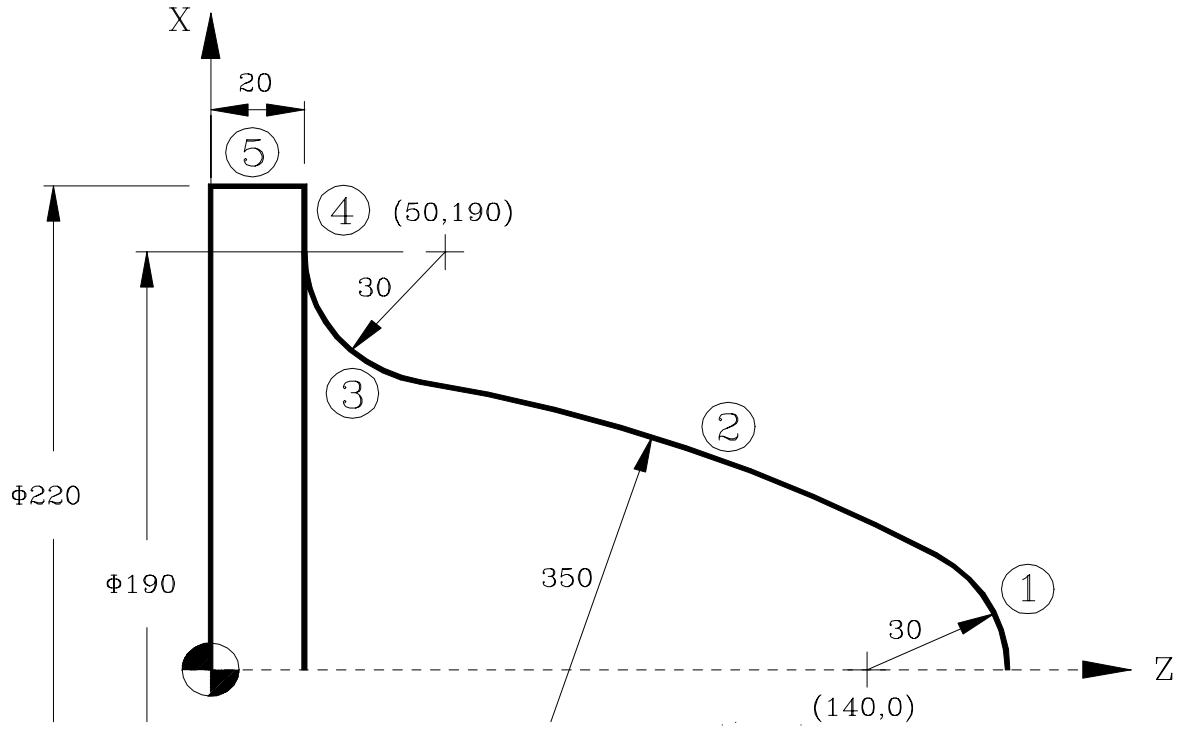
Coordinates (X, Z) X 120.0000 Z 90.0000

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

External profile  Work quadrant  Type of machining 

Profile definition

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

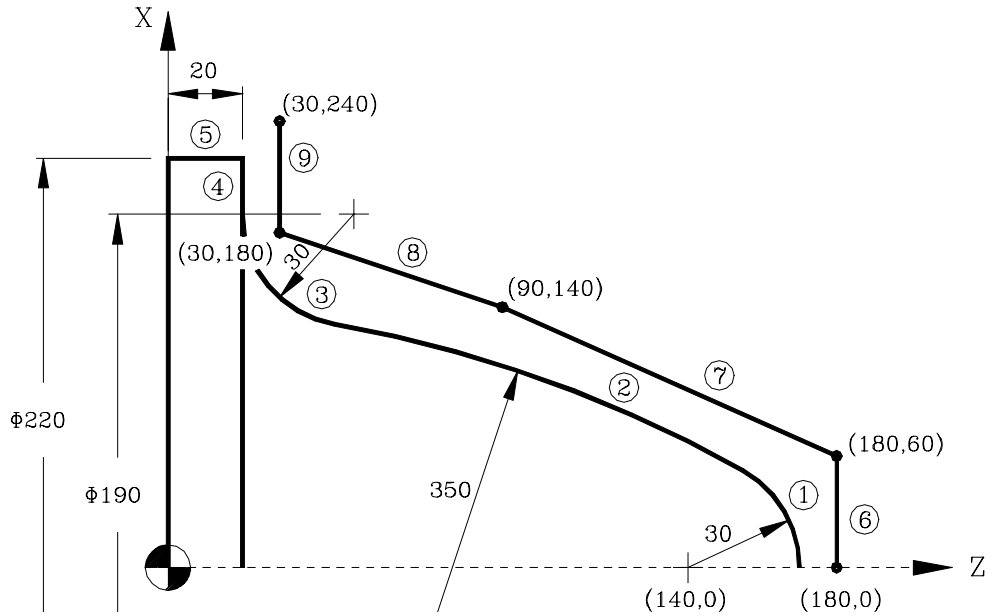
Coordinates (X, Z) X 230.0000 Z 180.0000

Safety distance X 0.0000 Z 0.0000

Roughing F 1.000 S 1000 T 3 Δ 2 

Finishing F 0.800 S 1000 T 3 δ 0.25

Spindle RPM 



Geometry definition

Outside profile  Work quadrant  Type of machining 

Definition of the desired final profile

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

Definition of the rough profile

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

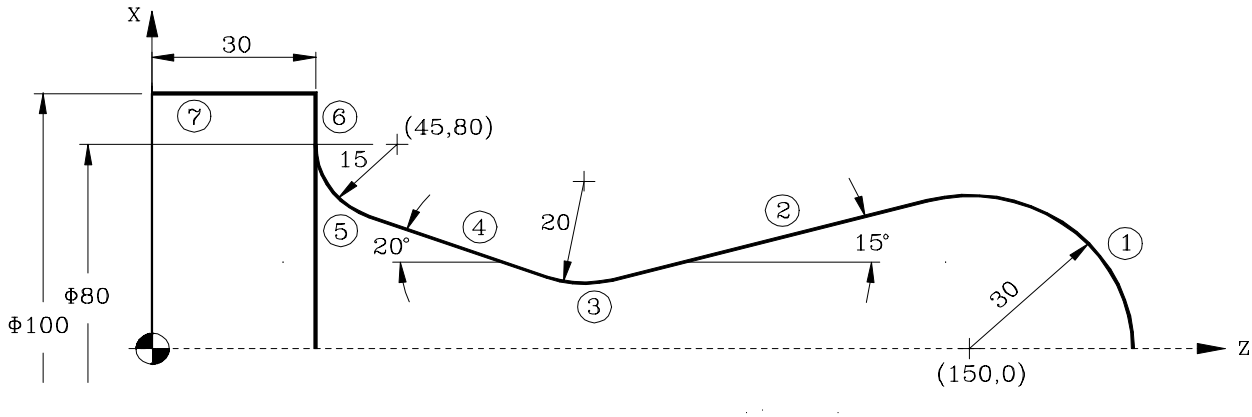
Coordinates (X, Z) X230.0000 Z180.0000

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

External profile  Work quadrant  Type of machining 

Profile definition

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

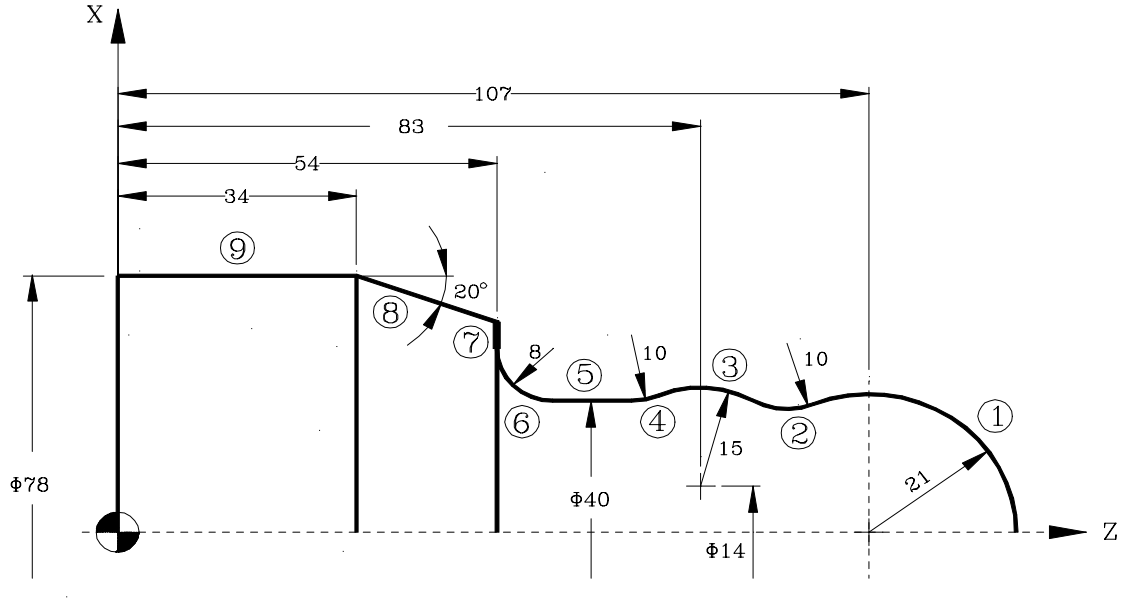
Coordinates (X, Z) X 110.0000 Z 190.0000

Safety distance X 0.0000 Z 0.0000

Roughing F 1.000 S 1000 T 3 Δ 2

Finishing F 0.800 S 1000 T 3 δ 0.25

Spindle RPM 



Geometry definition

External profile  Work quadrant  Type of machining 

Profile definition

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

Coordinates (X, Z) X 85.0000 Z 135.0000

Safety distance X 0.0000 Z 0.0000

Roughing F 1.000 S 1000 T 3 Δ 2

Finishing F 0.800 S 1000 T 3 δ 0.25 

Spindle RPM 

5. PROGRAM STORAGE

This CNC allows the editing, simulating and executing of part-programs.

Each of these programs consists of the interlinking of elementary operations or cycles and/or blocks edited in ISO code. The form of editing or defining said operations or cycles is explained in the previous chapter.

This chapter explains how to operate with these part-programs and has the following sections and subsections for this purpose.

List of programs stored

See the content of a program ..... See one of the operations in detail

Edit a new part-program..... ..... Storage of an operation or cycle

Erase a part-program

Copy one part-program in another

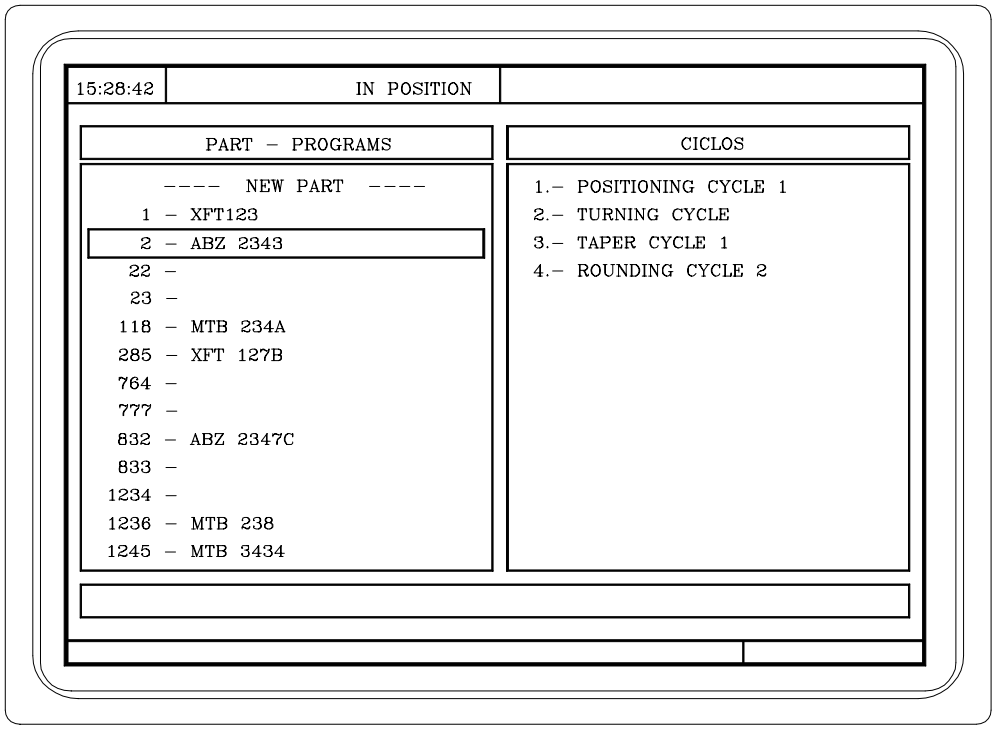
Modify a part-program ..... Erase an operation
Move an operation to another position
Add or insert to a new operation
Modify an already existing operation

5.1 LIST OF STORED PROGRAMS

To access the list of part-programs stored press P.PROG

Note: If the «Tool Calibration» mode is selected you cannot directly access the list of part-programs. This mode must first be left, that is, press ESC and then P.PROG.

The CNC will display the following information:



On the left there is a list of part-programs that are stored in the CNC's memory.

When there are more programs than those displayed in the window, use keys ↑ and ↓ to move the pointer over the list of programs. To go forward or backward page by page use the following combinations of keys SHIFT ↑ and SHIFT ↓.

The right-hand column will display the cycles and/or the ISO blocks that said part consists of.

After selecting the program list, the CNC will let you:

Create a new part-program
See the content of a part-program
Erase a part-program
Copy a part-program in another
Modify a part-program

To leave the directory or list of part-programs press:

the ESC key
the key for an operation
the key ISO

5.2 SEE CONTENT OF A PROGRAM

To see the content of a part-program, select this with the pointer from the left-hand column. To do this use  and  .

The right -hand column will display the cycles which said part consists of:

If you press  or  or  the pointergoes on to the right-hand column.

Now keys  and  let the pointer be moved over the blocks or cycles which make up the part.

To sum up, use keys:

 and  to move up and down in each one of the columns

 and  to change the column

After selecting an operation, the CNC allows:

Seeing the operation in detail
Erasing the operation
Moving the operation to another position
Modifying the operation

5.2.1 SEEING THE OPERATIONS IN DETAIL

After selecting the operation required, with the pointer, press 

The CNC will display all the data for said operation.

Now you can:

Simulate the operation. (See following chapter).
Execute the operation. (See following chapter).
Modify the operation
Store the operation. Replace the previous one or including this as a new one.

5.3 EDIT A NEW PART-PROGRAM

To edit a new part-program the following steps should be taken:

* Press [P.PROG] to access the list of part-programs stored.
* Use the pointer to select the option "--Create new part --2" in the left-hand column.
* Press [P.PROG]. The CNC will ask at the bottom for the number to be given to the new part-program, prompting the first one available.
* Type in the program number required and press [ENTER]

  This must be a number between 1 and 899999, and both numbers can be used.
* The CNC will ask for the comment to be assigned to the part-program.

  A comment does not have to be associated.
* Press [ENTER] or [ESC]

  The CNC includes the new part-program in the list of part-programs (left-hand column).

From this time all the operations required can be stored, and in the required order.

5.3.1 STORAGE OF AN ISO BLOCK OR A CYCLE

An ISO block or a cycle may be added at the end of the program, after the last operation, or inserted between 2 existing operations.

To store an operation the following steps must be taken:

* Define the operation or cycle required, assigning this the relevant data
* Press [P.PROG] to access the list of part-programs stored.
* Use the pointer to select the program number required in the left-hand column and go on to the right-hand column.
* Move over the operation after which the block or cycle is to be stored and press [ENTER]

Example:

You have	You want
1.- Positioning cycle 2	1.- Positioning cycle 2
2.- Facing cycle	2.- Facing cycle
3.- Taper cycle 2	3.- Taper cycle 2
	4.- Turning cycle
4.- Rounding cycle 2	5.- Rounding cycle 2
5.- Taper cycle 1	6.- Taper cycle 1
	7.- Threading cycle 1

4.- Turning cycle After defining the cycle move over the operation "3.- Taper cycle 2" and press [Enter]

7.- Threading cycle 1 After defining the cycle move over the last "Taper cycle 1" operation and press [Enter]

5.4 ERASING A PART-PROGRAM

To erase a part-program follow these steps:

* Press P.PROG to access the list of part-programs stored.
* Use the pointer to select from the left-hand column the part-program to be erased.
* Press CLEAR

  At the bottom the CNC will display a message requesting confirmation of the erasing operation.

  If you press ENTER the CNC will erase the program selected and update the list of part-programs stored.

  If you press ESC the program will not be erased and the erasing operation is left.

5.5 COPY A PART-PROGRAM IN ANOTHER

To copy a part-program in another take the following steps:

* Press P.PROG to access the list of part-programs stored.
* Use the pointer to select in the left-hand column the part-program to be copied.
* Press P.PROG

  At the bottom the CNC will display a message requesting the number to be assigned to the copy.

* Type in the program number required and press ENTER

  This must be a number between 1 and 899999, and both numbers can be used.

* If there is already a part-program with said number, the CNC will display a message at the bottom, asking if this should be replaced or if you wish to cancel the operation.

  If you press ENTER the CNC will ask for a new program number

  If you press ESC the CNC will erase the present program and carry out program copying.

* The CNC requests the comment to be associated with the new part-program (with the copy).

  A comment does not have to be associated.

* Press ENTER or ESC

  The CNC updates the list of part-programs stored.

5.6 MODIFYING A PART-PROGRAM

To modify a part-program the following steps must be taken:

* Press P.PROG to access the list of part-programs stored.
* Use the pointer to select from the left-hand column the part-program you wish to modify.

After selecting the program, the CNC lets you:

Erase an operation
Move an operation to another position
Add or insert a new operation
Modify an already existing operation.

5.6.1 ERASING AN OPERATION

To erase an operation follow these steps:

* Use the pointer to select the operation to be erased, in the right-hand column.
* Press CLEAR

  The CNC will display a message at the bottom, requesting the confirmation of the erasing operation.

  If you press ENTER the CNC will erase the operation selected and update the right-hand column.

  If you press ESC the operation is not erased and the erasing operation is left.

5.6.2 MOVING AN OPERATION TO ANOTHER POSITION

To move an operation to another position take the following steps:

* Use the pointer to select the operation to be moved from the right-hand column.
* Press *

  The CNC will display this operation in highlighted text.

* Place the cursor after the operation which the operation is to be moved to and press ENTER

Example:

You have	You want
1.- Positioning cycle 2	1.- Positioning cycle 2
2.- Facing cycle	2.- Taper cycle 2
3.- Taper cycle 2	3.- Turning cycle
4.- Turning cycle	4.- Rounding cycle 2
5.- Rounding cycle 2	5.- Taper cycle 1
6.- Taper cycle 1	6.- Threading cycle 1
7.- Threading cycle 1	7.- Facing cycle

Select the "Facing cycle" and press *

Move the cursor onto the "Taper cycle 1" and press ENTER

5.6.3 ADDING OR INSERTING A NEW OPERATION

To add or insert an operation take the same steps as to store an operation.

* Define the desired block or cycle, assigning this the relevant data.
* Press P.PROG to access the list of part-programs stored.
* Move over the operation after which the block or cycle is to be stored and press ENTER

5.6.4 MODIFYING AN ALREADY EXISTING OPERATION

To modify an operation take the following steps:

* Use the pointer to select, in the right-hand column, the block or cycle to be modified.
* Press RECALL

  The CNC will display the relevant edition page for this operation.

* Modify all the data required.

  To store the modified operation again:

* Press P.PROG to access the list of part-programs stored.

  The CNC displays the pointer over the same operation.

  To select another position use the ↑ ↓ . The new operation will be inserted after this point.

* Press ENTER

  If one wishes to place the modified operation in its previous location, the CNC will display a message asking if one wishes to replace the previous operation or keep this, inserting the new one after.

  In the following example the "Facing cycle" operation is modified

You have	Option Replace	Option Replace
1.- Facing cycle	***1.- Facing cycle***	***1.- Facing cycle***
2.- Taper cycle 2	2.- Taper cycle 2	***2.- Facing cycle***
		3.- Taper cycle 2

Note: One can select an existing operation, modify this and then insert this somewhere else and even in another part-program.

6. EXECUTION AND SIMULATION

Simulation allows graphic reproduction of a part-program or an operation with the data that has been defined.

By means of simulation, one can thus check the part-program or the operation before executing or storing this and consequently correct or modify the data:

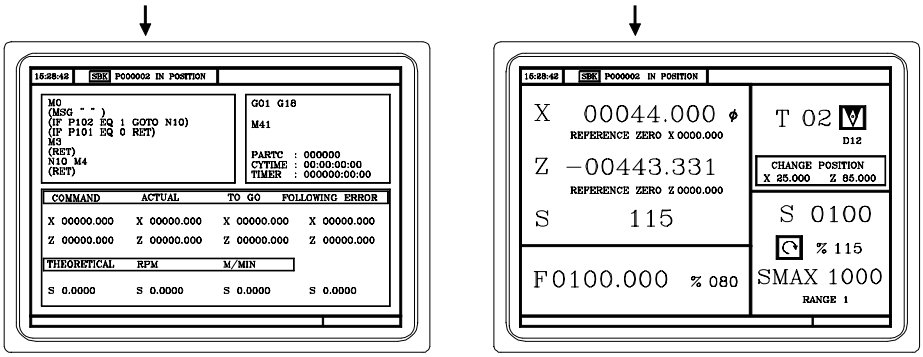
The CNC allows a part-program or any operation to be executed or simulated. This simulation or execution can be done from beginning to end or alternatively press [SINGLE] for this to be executed or simulated step by step.

The CNC enables execution or simulation of:

Any operation or cycle.
A part-program.
An operation stored as part of a part-program.

Warning

Whenever a part-program or an operation stored as part of a part-program is selected for simulation or execution, the CNC selects this part-program in the top center window and highlights it next to the [I] symbol.



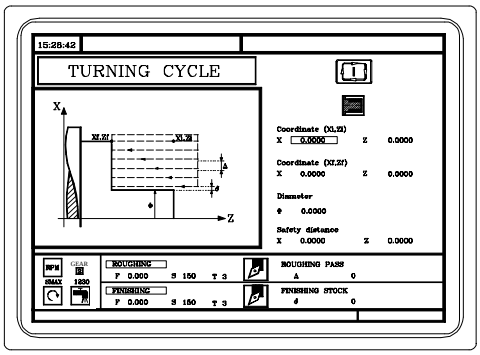
It then acts as follows:

If [I] is pressed, the CNC executes the part-program that is selected.

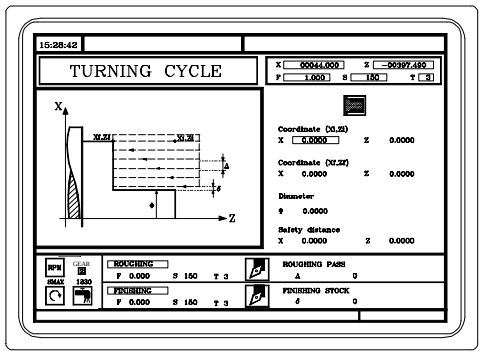
if [CLEAR] is pressed, the part-program is de-selected and the CNC deletes it from the top center window..

6.1 SIMULATING OR EXECUTING AN OPERATION OR CYCLE

All the operations or cycles have 2 operating modes: Execution mode and Edition Mode



Execution mode



Edition Mode

Simulation

The operation or cycle can be simulated in both operating modes. To do this press [GRAPHICS]
The CNC will display the graphic representation page for 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 operation mode is selected.

To exit the edition mode and go on to execution mode press [ESC]

To execute an operation or cycle, press [▷]

6.2 SIMULATING OR EXECUTING A PART-PROGRAM

Whenever you wish to simulate or execute a part-program do the following:

* Press P.PROG to access the list of part-programs stored.
* Select the program to be simulated or executed from the left-hand column.

To simulate the part-program press GRAPHICS and to execute this press START.

6.2.1 SIMULATING OR EXECUTING A SECTION OF A PART-PROGRAM

To simulate or execute a part program, proceed as follows:

* Press P.PROG to access the list of the stored part-programs.
* Select the program in the left column and the first operation to be executed or simulated in the right column.

Press GRAPHICS to simulate the part program, and START to execute it.

Warning

Whenever a section of the part-program is executed, the CNC does not execute the initial subroutine 9998 associated with all part-programs.

6.3 SIMULATING OR EXECUTING A STORED OPERATION

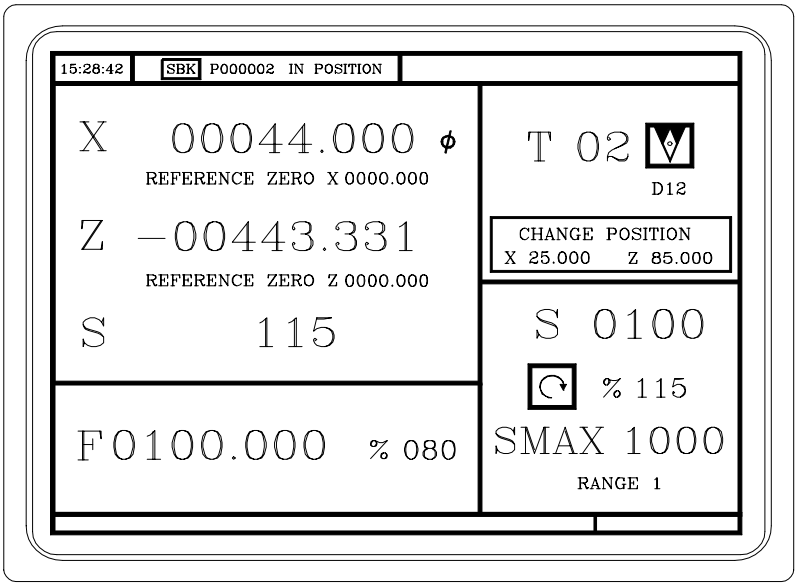
To simulate or execute an operation which is stored as part of a part-program do the following:

* Press P.PROG to access the list of part-programs stored.
* Select the program which contains this from the left-hand column and the operation required to be simulated or executed from the right-hand column.
* Press RECALL

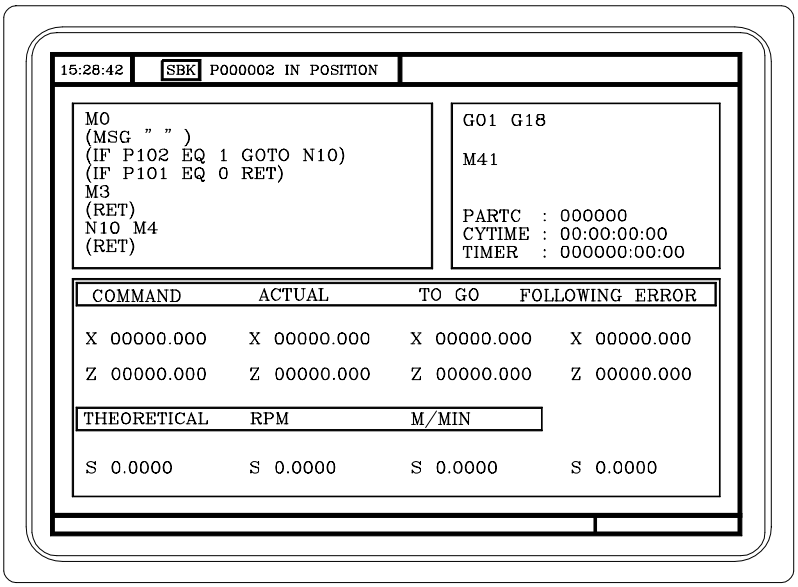
To simulate the operation press GRAPHICS and to execute this press START.

6.4 EXECUTION MODE

When you press  to execute an operation or part-program, the CNC displays the standard TC operating mode screen.



If you press  the CNC displays the special TC operating mode screen.



After selection, the operation or part can be executed as many times as necessary. To do this, after execution once more press 

During execution of the operation or part one can press  to access the graphic representation mode.

To stop execution press 

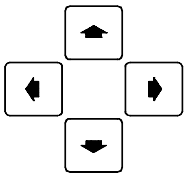
After stopping the execution the CNC allows a tool inspection to be made. See the following section.

6.4.1 TOOL INSPECTION

Depending on how the CNC has been set, tool inspection will be available after interrupting the execution without pressing any key or after pressing [T]

Once tool inspection has been selected, one can:

Jog the axes up to the tool change position

Move the tool to the tool change position by means of the  keys or the handwheels.

Select another tool

In order to make the tool change, the standard screen for the TC mode must be selected.

Press [T] The CNC will highlight the tool number.

Key in the number of the tool to be selected and press [I] for the CNC to select the new tool.

The CNC will carry out the tool change.

Modify the tool values (dimensions and geometry)

Press [ ] The CNC will show the Tool Calibration screen.
Tool dimensions can be modified (I, K offsets for tool wear) or the values for tool geometry.
To exit this screen and return to the previous one (while staying in tool inspection) press [ESC]

Resume program execution

To resume program execution, press [I]
The CNC will reposition the tool, moving it to the point where tool inspection began. There could be two cases:

1.- Only one of the axes has been moved.
The CNC repositions it and resumes execution.

2.- Both axes have been moved.
The CNC will display a window with the following options for selecting the positioning order of the axes.

PLANE	Both axes will be moved at the same time
Z-X	The Z axis moves first and then the X axis
X-Z	The X axis moves first and then the Z axis

6.5 GRAPHIC REPRESENTATION

When you press  the CNC displays the "T" model graphic representation page.

To leave the graphic representation mode press  or 

In the CNC 8055 T Operation Manual, section «Graphics» in the «Execution / Simulation» chapter, there is an explanation of how to operate during graphic representation. Nevertheless, there will now be a brief description of the softkeys.

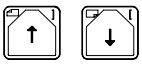
Type of graphics. Can be "X-Z", "X-C", "Z-C", "Solid X-Z", "Solid X-C" and "Solid Z-C"

The "X-Z", "X-C" and "Z-C" graphics are line graphics which use colored lines to describe tool tip movement.

The "Solid X-Z", "Solid X-C" and "Solid Z-C" graphs start from an initial block. During execution or simulation the tool removes material and the form of the resulting part is seen.

Zone to be displayed

Allows modification of the display zone, by defining the maximum and minimum coordinates of each axis.

To select the maximum and minimum coordinates use 

After defining all the data press 

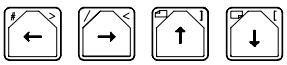
After selecting a new display zone the CNC erases the screen showing the axes or the unmachined part.

The zone displayed cannot be modified during execution or simulation of the piece. In this case stop execution or simulation by pressing 

Zoom

This function allows the graphic representation zone to be increased or reduced in size.

It displays a window superimposed on the graphic represented and another on the figure in the lower right-hand part of the screen. These windows indicate the new zone of graphic representation that is being selected.

To move the window use  to increase or reduce its size use s “+” “-”, and for the CNC to assume these values press 

Each time a new display zone is selected the CNC keeps the present graphic representation. It does not erase this.

When you press  to continue with or restart execution or simulation, the present graphic representation is erased and the next starts with the new values.

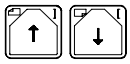
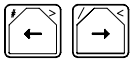
The zoom function cannot be executed during execution or simulation of the part. In this case, interrupt the execution or simulation by pressing 

Graphic parameters

Simulation speed. In the top right-hand of the screen select the percentage of the simulation speed to be applied.

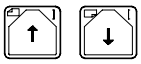
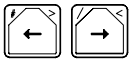
To select the percentage use the  keys and for the CNC to assume said value, press 

Colors of the path. This only applies in line graphics (not solid). It enables selection of colors to represent fast feedrate, path with no compensation, path with compensation and threading.

From the right-hand side of the screen, use the  keys to select the type of path and the  keys to select the color required for application.

For the CNC to assume said values press 

Colors of the solid. This only applies in solid graphics (not in line graphics). It enables selection of colors to represent the cutter, the part, the axes and the clamps.

At the top right-hand side of the screen use keys  to select the type of path and the keys  to select the color to be applied.

For the CNC to assume said values press 

Erase screen

When this option is selected the CNC erases the screen and displays the axes or the unmachined part.

The screen cannot be erased during simulation of the part. In this case stop simulation by pressing the  key

After selecting the types of graphics, the display area, the graphic parameters, etc. press  to start the graphic simulation.

During the graphic simulation, the CNC takes into account the simulation speed and the position of the right Manual Feedrate Override switch (0%-120% FEED).

When selecting a new simulation speed, the CNC applies a 100% of it regardless of the position of the switch.

Once the switch is moved, the CNC starts applying the selected %.

To interrupt the simulation, press 

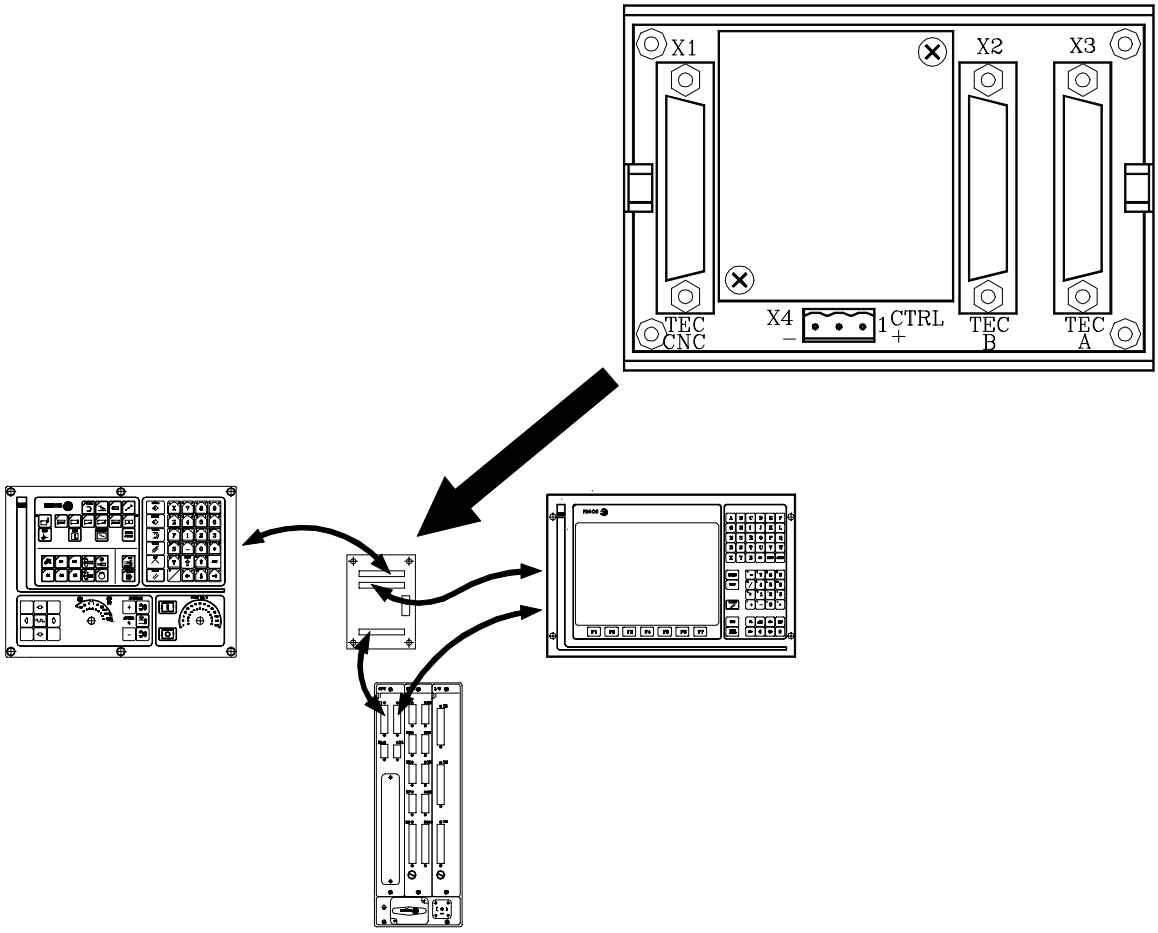
To quit the simulation mode, press 

or 

Appendix

Keyboard Selection

When 2 keyboards are available the keyboard switching board must be used.

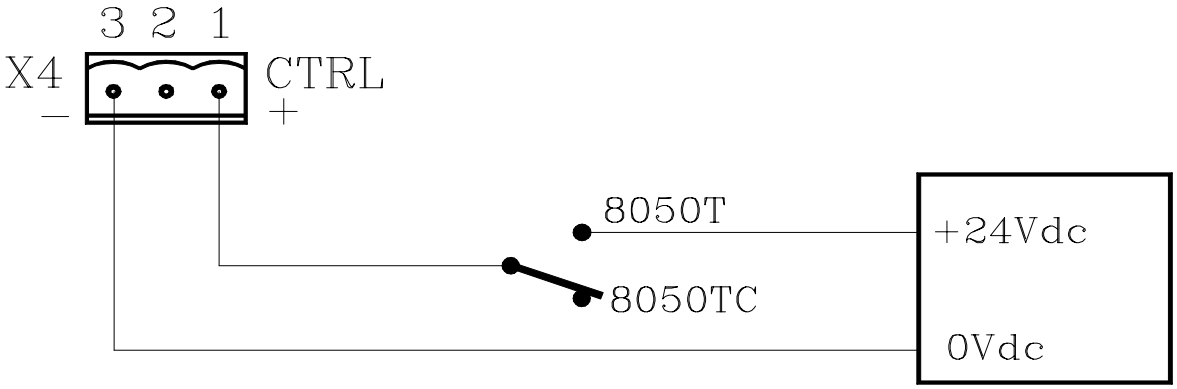


We will now explain some possibilities for keyboard selection by means of the keyboard switching board.

By means of a switch

The switch can be installed anywhere on the machine.

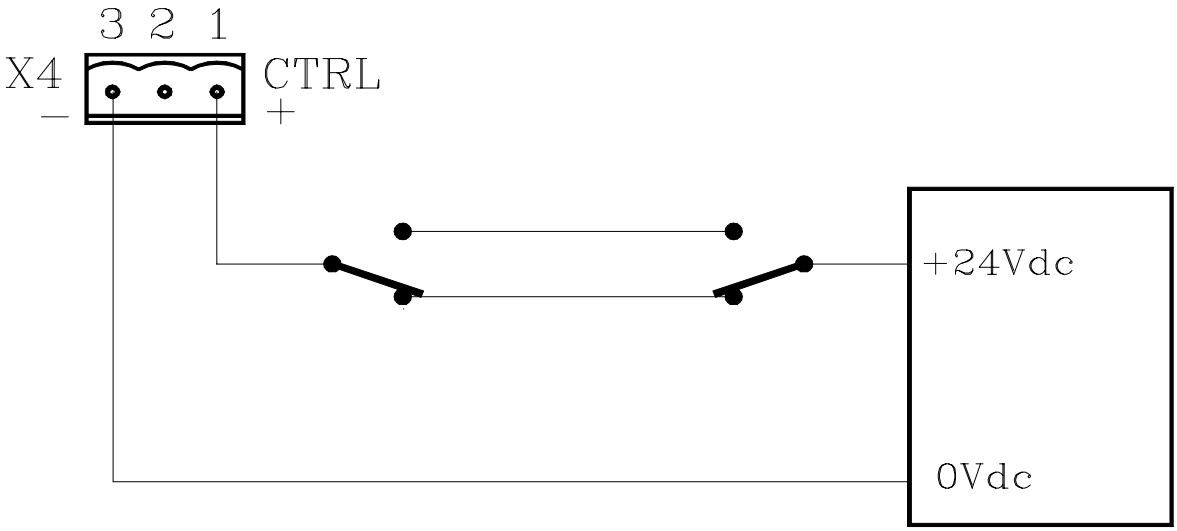
When this is placed in one of the positions, the "TC" keyboard will be selected, and when in the other position, the monitor keyboard.



By means of two switches

Locate each of the switches beside each of the keyboards.

Whenever the position of any switch is moved the keyboard changes, meaning that if the TC keyboard was selected it will select the Monitor keyboard and vice versa.



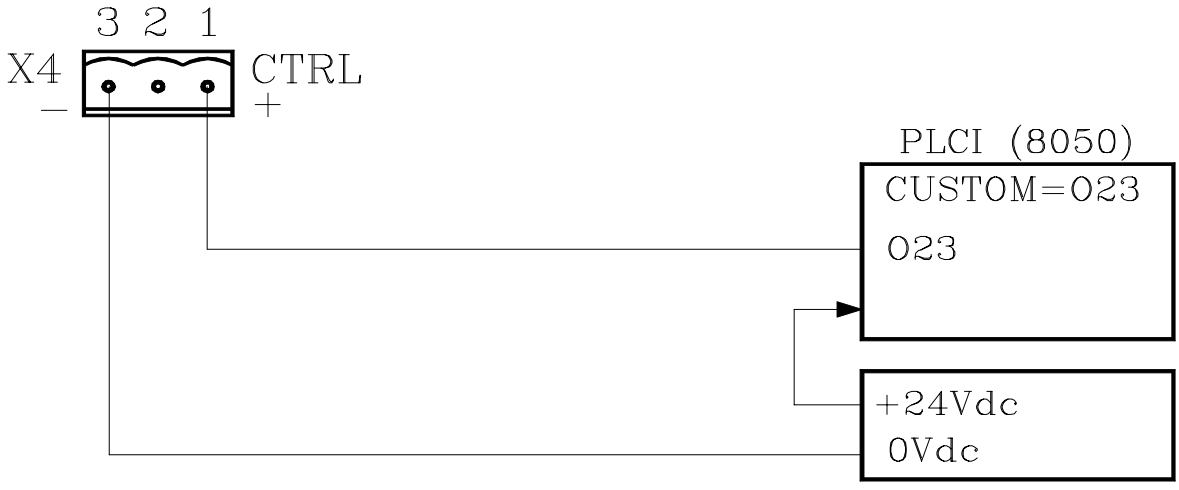
By means of the PLC

The general logic output of the CNC CUSTOM (M5512) tells the PLC the operating mode that is selected.

CUSTOM (M5512) = 0	The "T" operating mode is selected.
CUSTOM (M5512) = 1	The "TC" operating mode is selected.

If the sentence CUSTOM=O23 is programmed in the PLC, output O23 indicates the operating mode selected in the CNC.

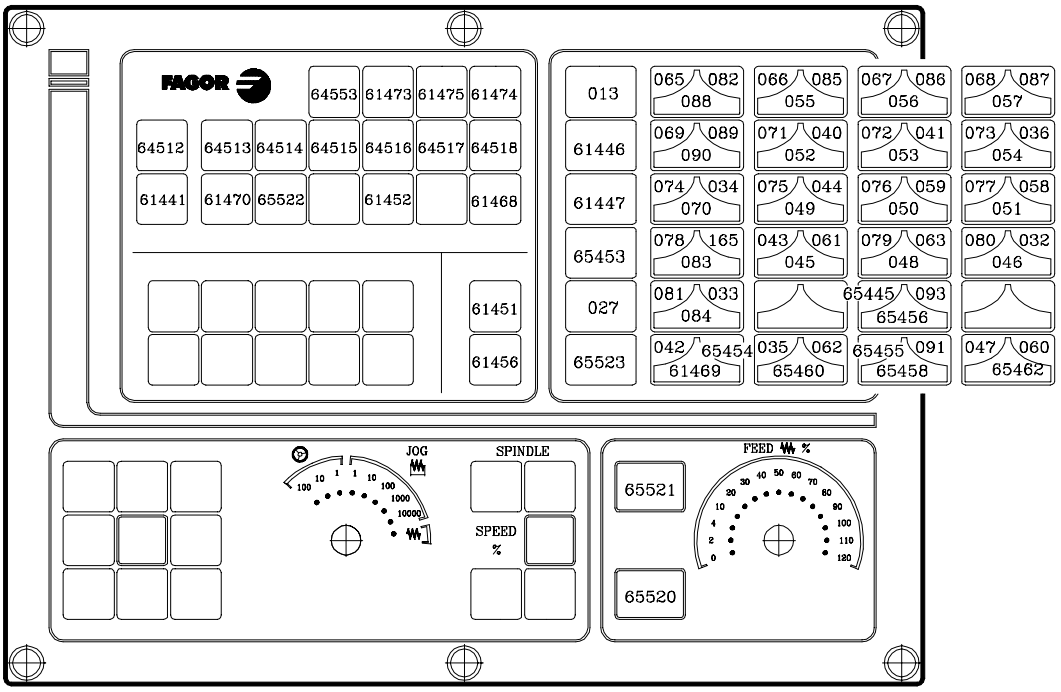
For this reason, if the connections are made as in the figure, every time the operating mode is changed the relevant keyboard will be selected.



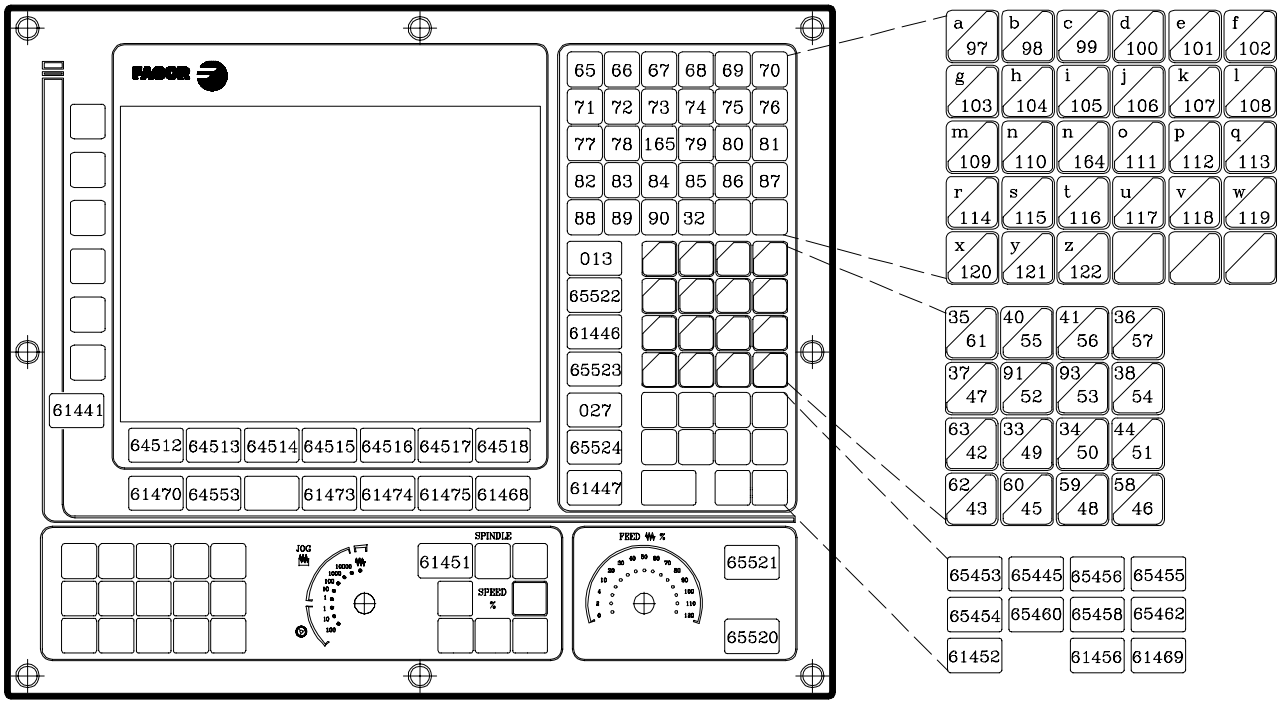
KEY CODES

Key codes returned by the customizing instruction (WKEY) in the KEY variable.

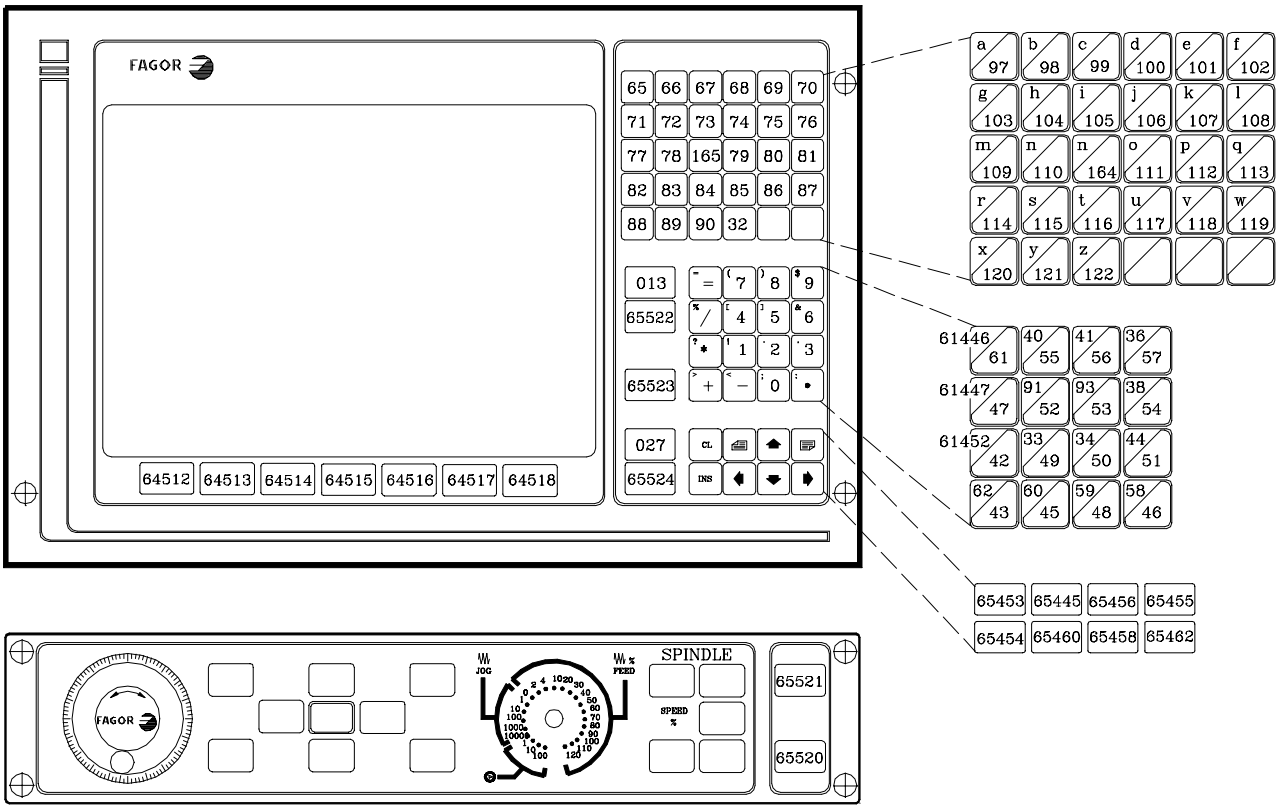
"TC" keyboard



11" LCD Monitor keyboard



14" Color monitor keyboard

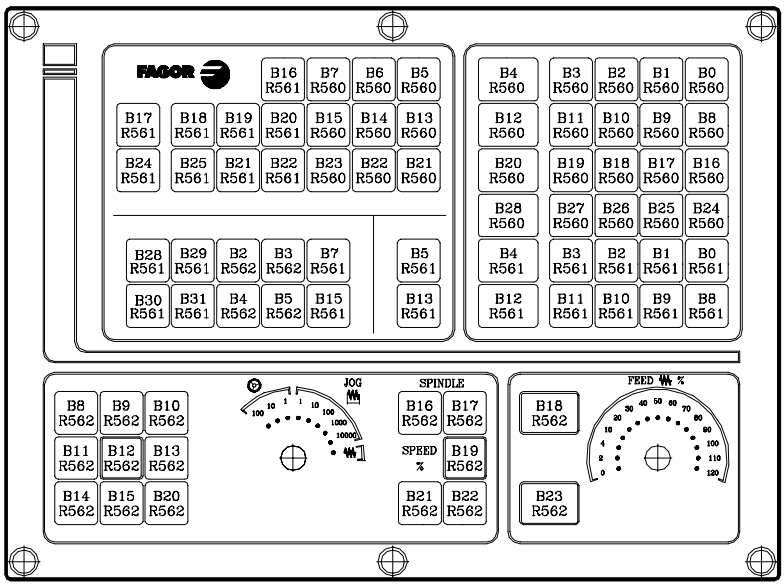


Logic outputs for key status

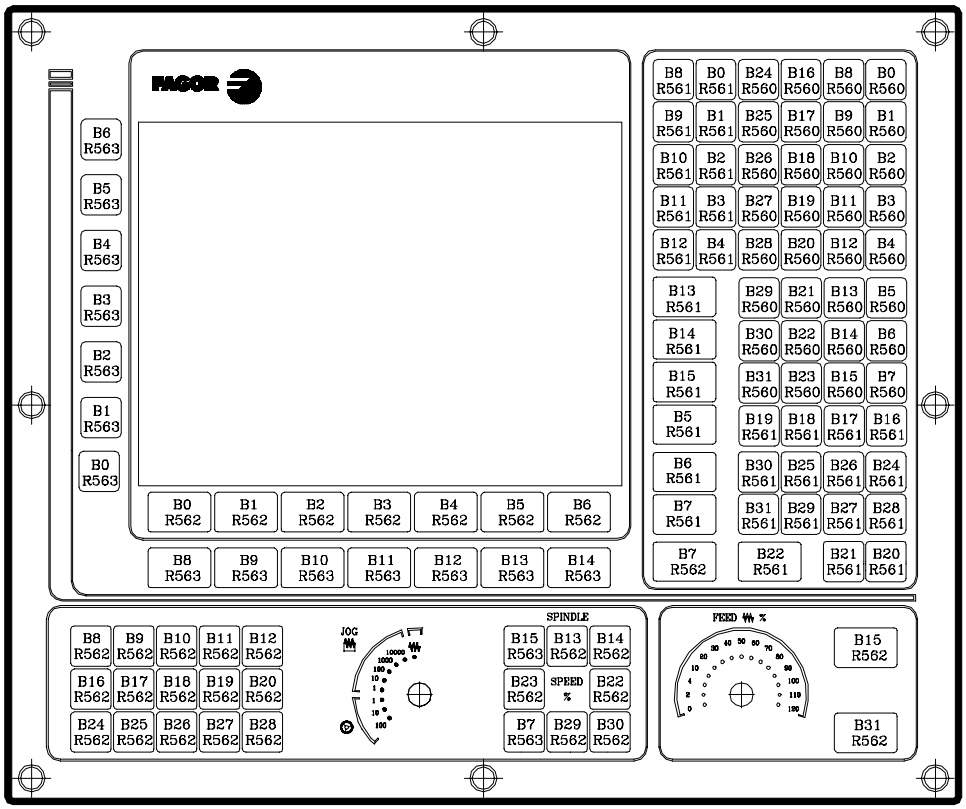
Registers KEYBD1 (R560), KEYBD2 (R561) and KEYBD3 (R562) indicate to the PLC whether any key is pressed.

When a key is pressed, its corresponding bit will be at logic state high (1) and will go back low (0) when the key is released.

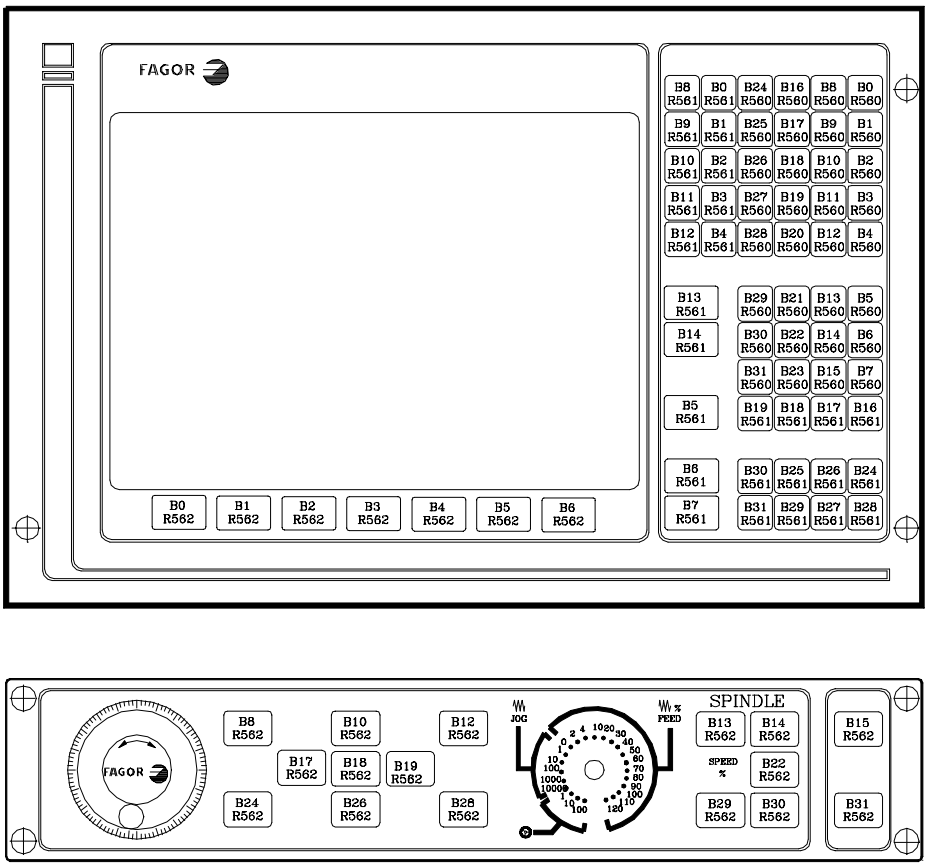
"TC" keyboard



11" LCD Monitor keyboard



14" color monitor keyboard



User cycles 1 through 20 (PCALL cycles) may be selected directly by means of keyboard simulation. The codes to be assigned to the KEY variable are:

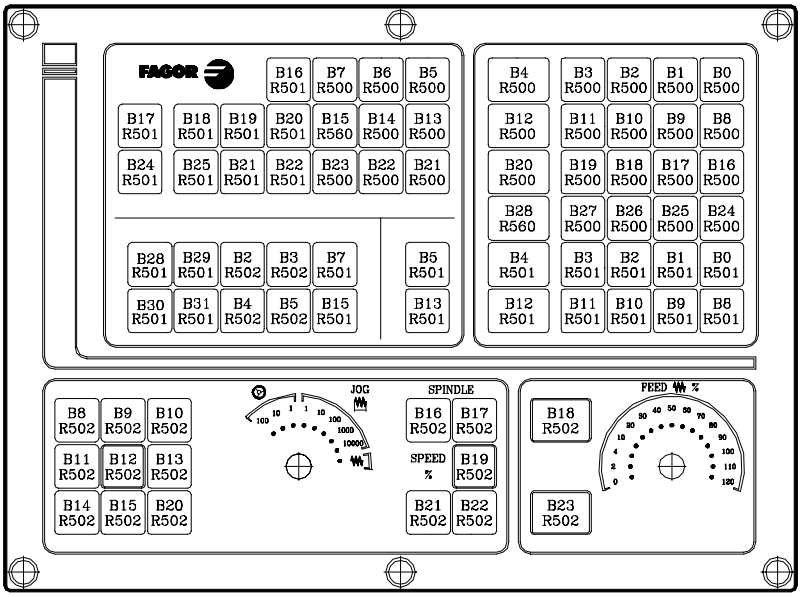
Cycle Nr.	Decimal code	Hexadecimal code
1	61697	0F101
2	61698	0F102
3	61699	0F103
4	61700	0F104
5	61701	0F105
6	61702	0F106
7	61703	0F107
8	61704	0F108
9	61705	0F109
10	61706	0F10A
11	61707	0F10B
12	61708	0F10C
13	61709	0F10D
14	61710	0F10E
15	61711	0F10F
16	61712	0F110
17	61713	0F111
18	61714	0F112
19	61715	0F113
20	61716	0F114

Keys inhibiting codes

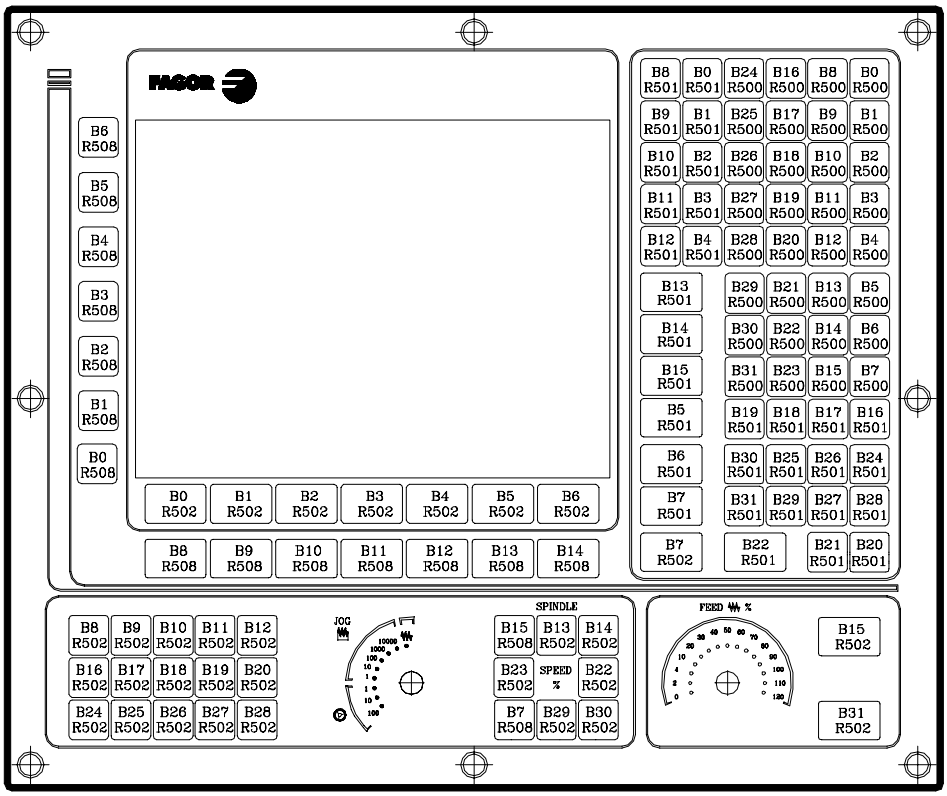
With registers KEYDIS1 (R500), KEYDIS2 (R501) and KEYDIS3 (R502), it is possible to individually inhibit the operation of the keys.

To inhibit a key, set the corresponding register bit high (to 1).

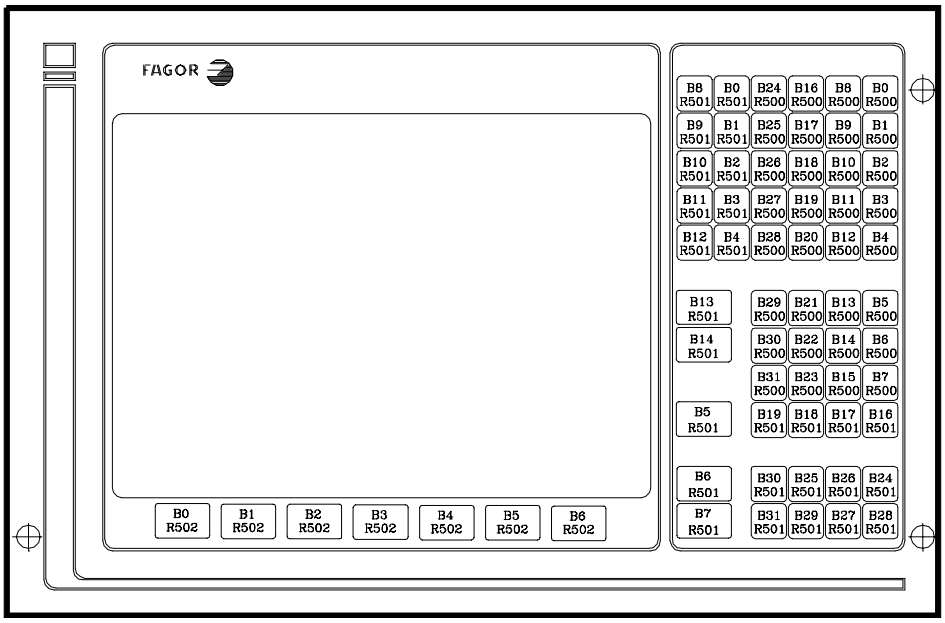
"TC" keyboard

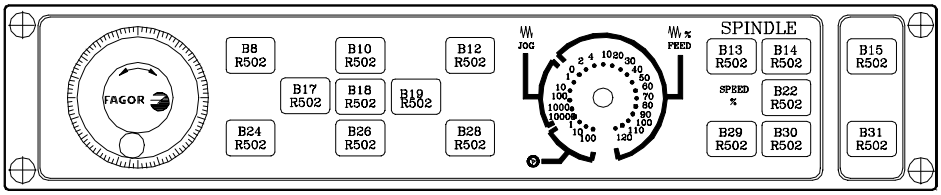


11" LCD Monitor keyboard



14" color monitor keyboard





FAGOR CNC 8055TC

SELF-TEACHING Manual

Ref. 9805 (ing)

INDEX

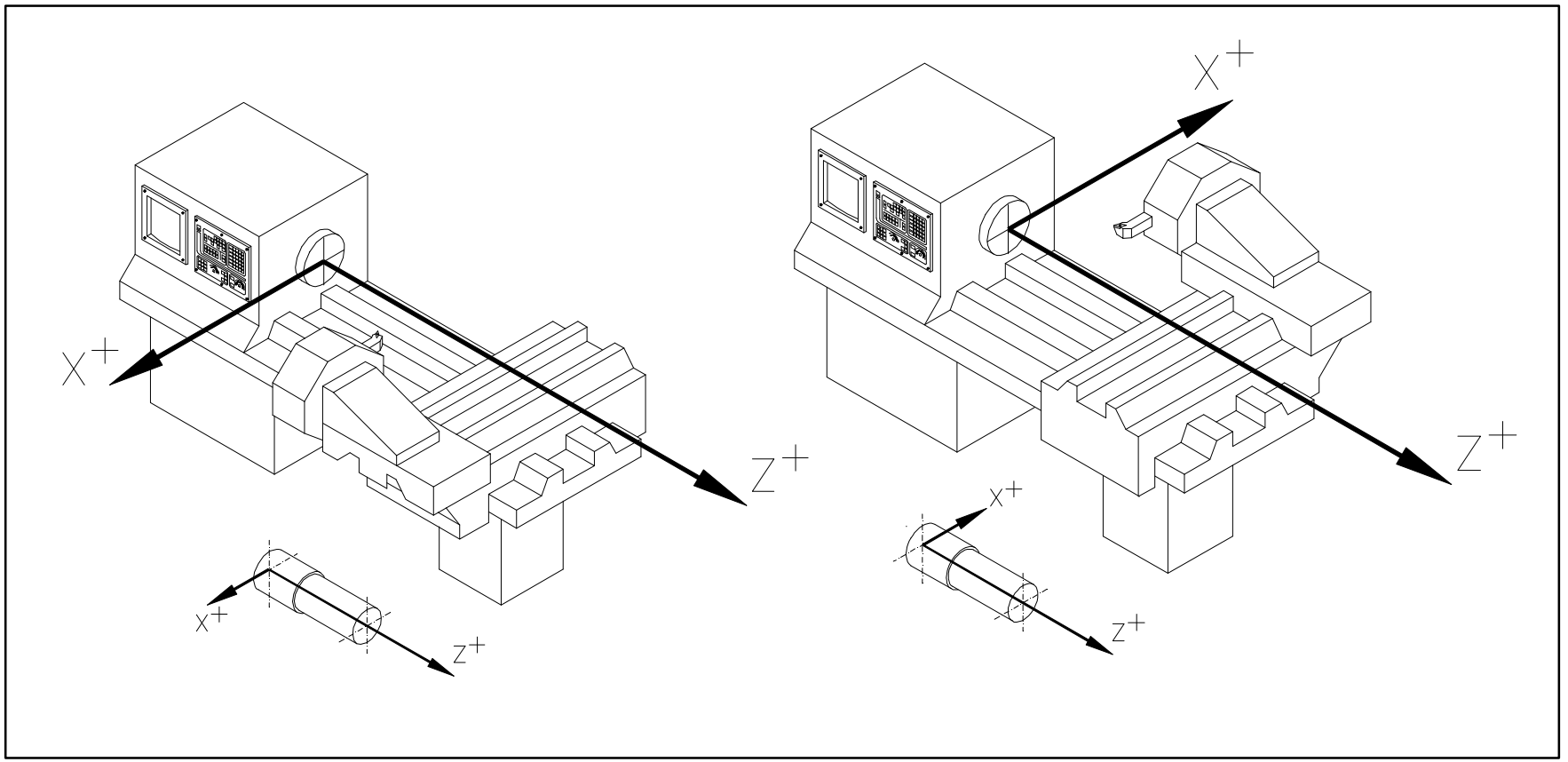
Chapter 1

Theory on CNC machines

This chapter describes:

- How to name the axes of the machine.
- What machine reference zero and part zero are.
- What “Home Search” is.
- What travel limits are.
- How to preset a part zero.
- Which are the programming units.
  - > millimeters/inches.
  - > radius/diameter.
- Ways to operate with the spindle.
  - > RPM/CSS. (Revolutions Per Minute/Constant Surface Speed).
- Ways to move the axes.
  - > mm/min or mm/rev.

1.1 Machine Axes.



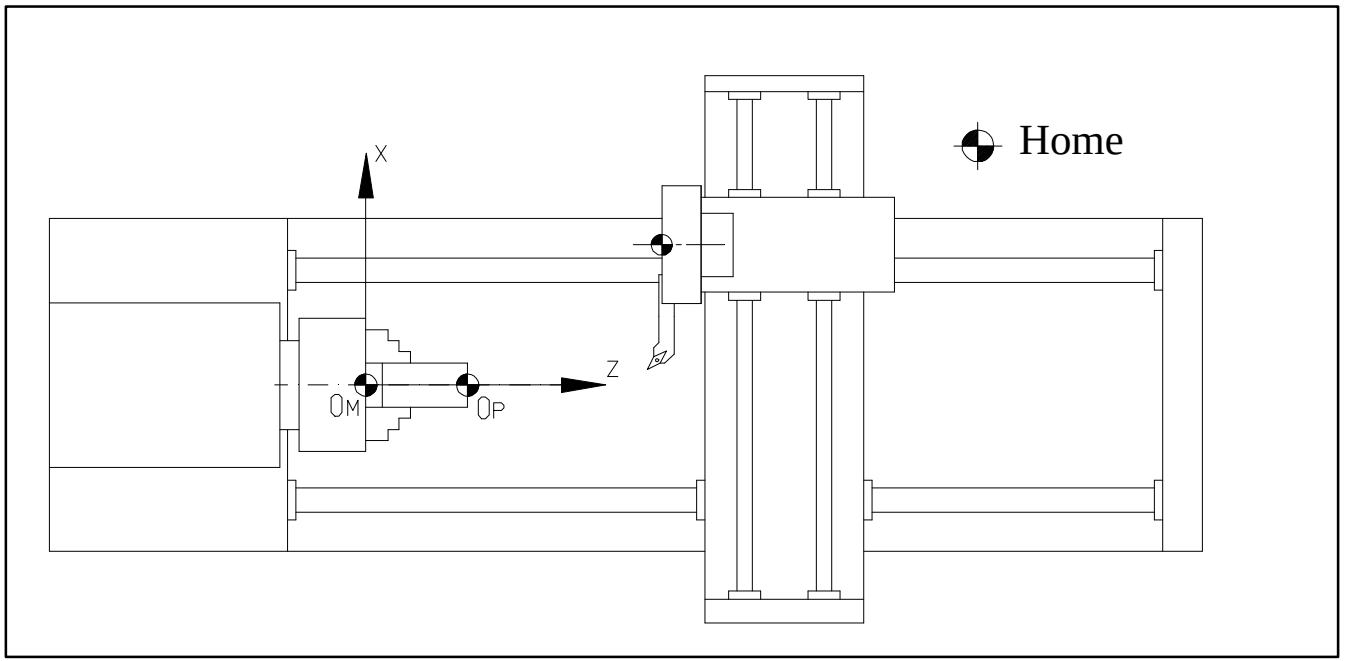
Z axis: Along the machine.
X axis: Across the machine.

1.2 Machine reference zero and part zero.

They are the references the machine needs in order to work:

– Machine ref. zero (O_M): Is set by the manufacturer and it is the origin point for the axes.

– Part zero (O_P): Is set by the operator. It is the part's origin or datum point with respect to which the movements are programmed. It could be set anywhere on the part.

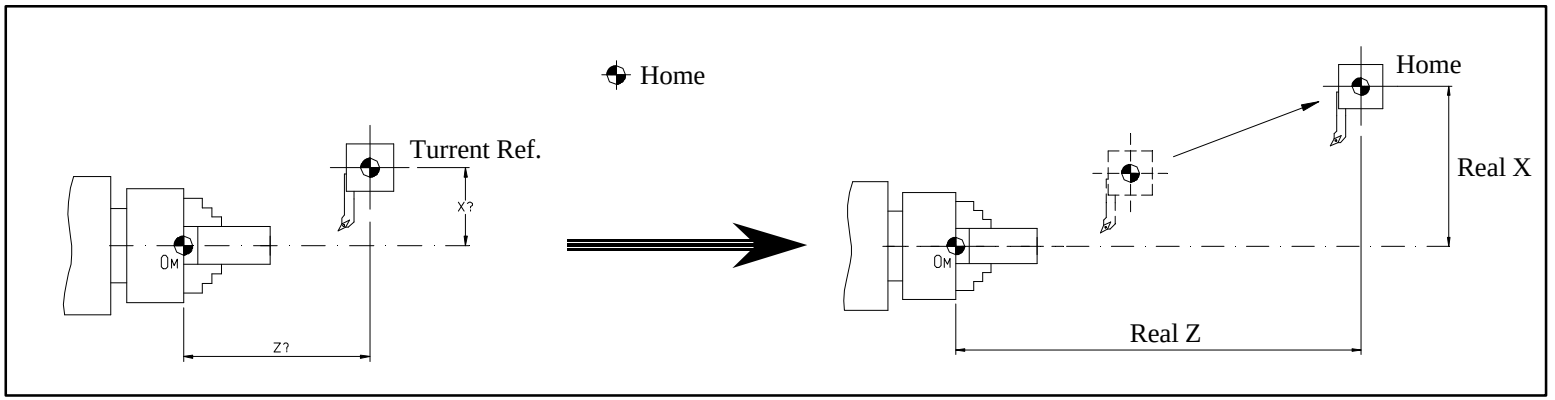


1.3 Home Search.

When the CNC is off, the axes may be moved by hand or by accident.

In these situations, the CNC no longer keeps track of the real position of the axes. That is why a “Home Search” should be carried out on power-up.

When searching home, the axes move to the home point set by the manufacturer and the CNC assumes the value of the coordinates set by the manufacturer for that point.



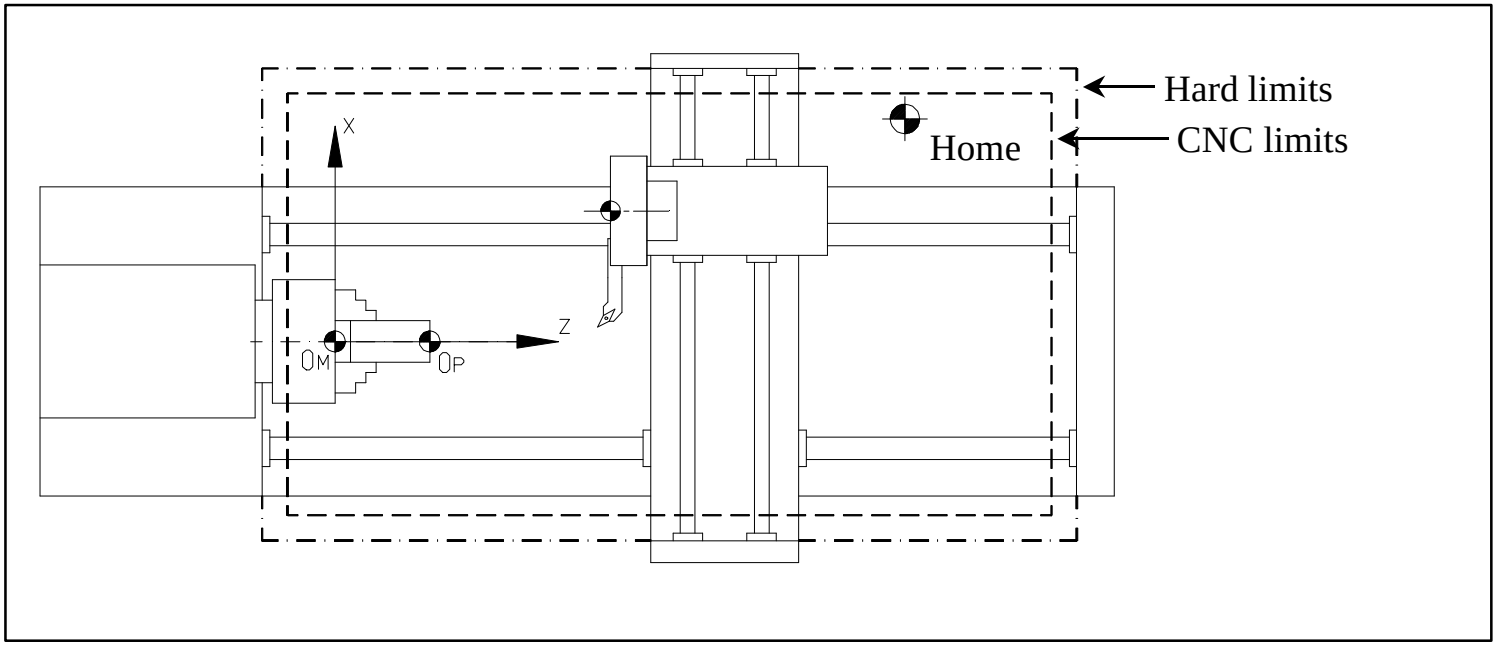
– Home: Set by the manufacturer. It is the point where the axes move during “Home Search”.

– Turret Ref.: Set by the manufacturer. Point moving with the turret. It is the point that moves during “Home Search”.

1.4 Travel limits.

There are two types of limits:

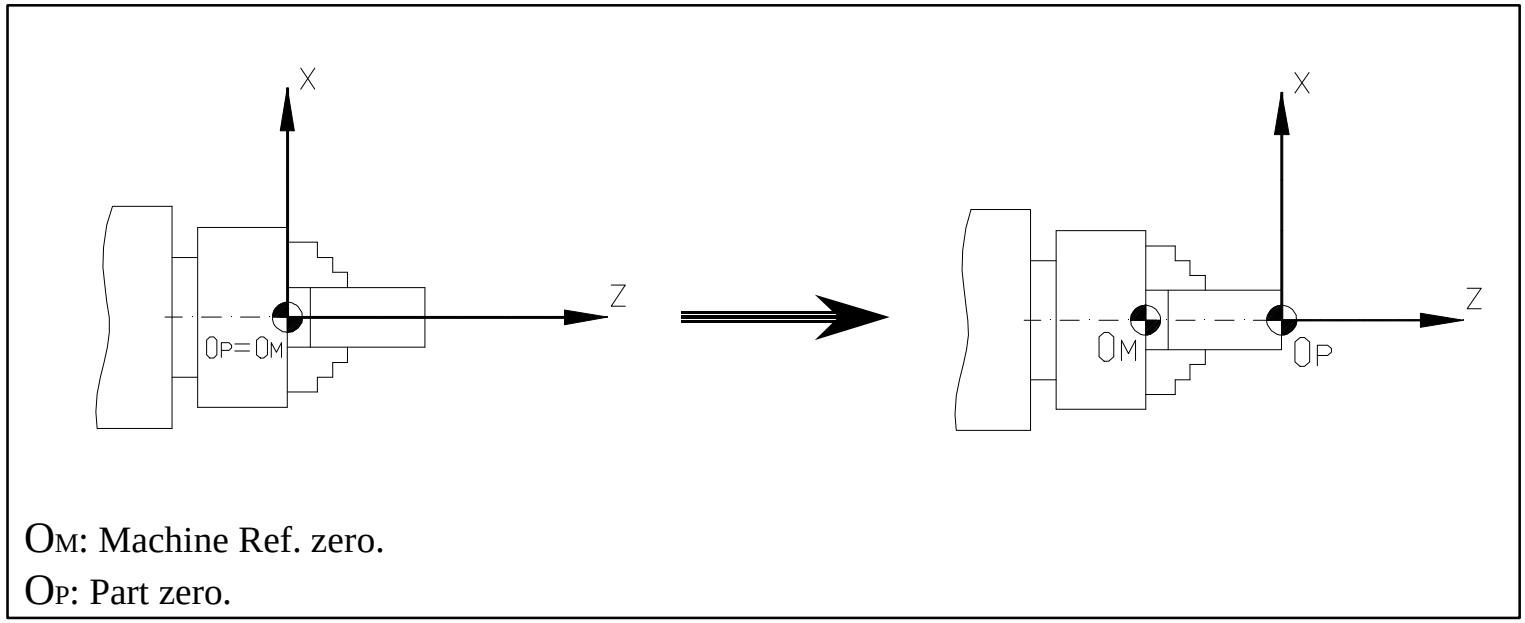
- Hard limits: Mechanical limits set on the machine to prevent the carriage from moving beyond the ways.
- CNC limits: Set at the CNC by the manufacturer to prevent the carriage from running into the machine's hard limits.



1.5 Part zero preset.

It is easier to program movements from a part zero.

The part zero is only set on the Z axis.

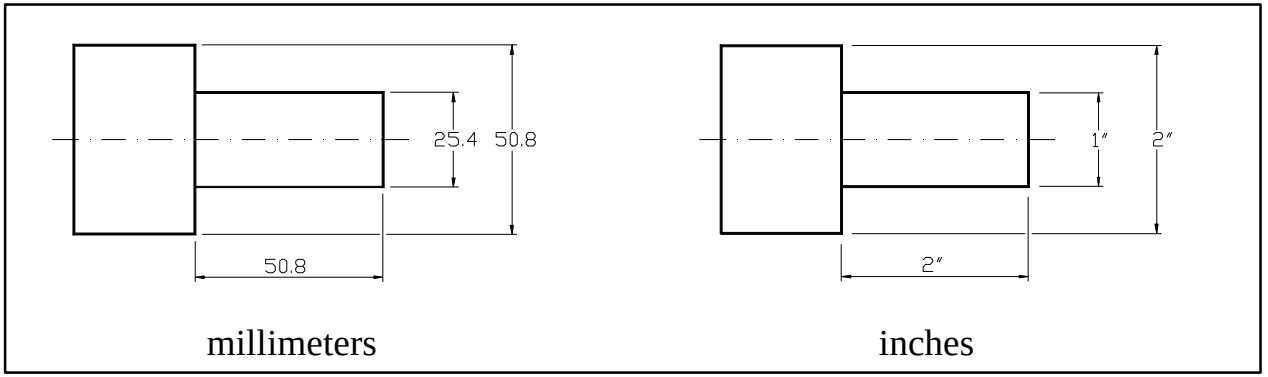


OM: Machine Ref. zero.

OP: Part zero.

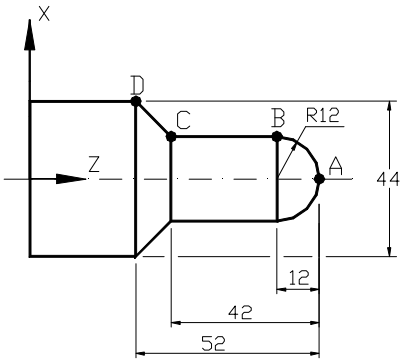
1.6 Programming units.

The movement units are set by the machine manufacturer in mm or inches.



millimeters inches

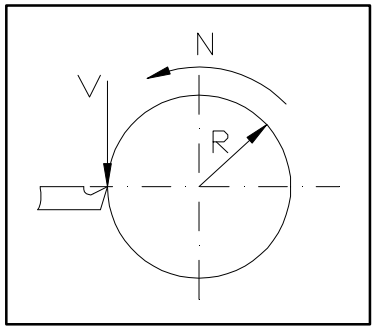
The programming units for the X axis are set by the machine manufacturer in radius or diameter.



	Radius	Diameter
A	X=0 Z=0	X=0 Z=0
B	X=12 Z=-12	X=24 Z=-12
C	X=12 Z=-42	X=24 Z=-42
D	X=22 Z=-52	X=44 Z=-52

1.7 Spindle speed.

It could be defined in two ways:



–Cutting speed (V):

It is the linear speed between the part and the tool at the contact point.

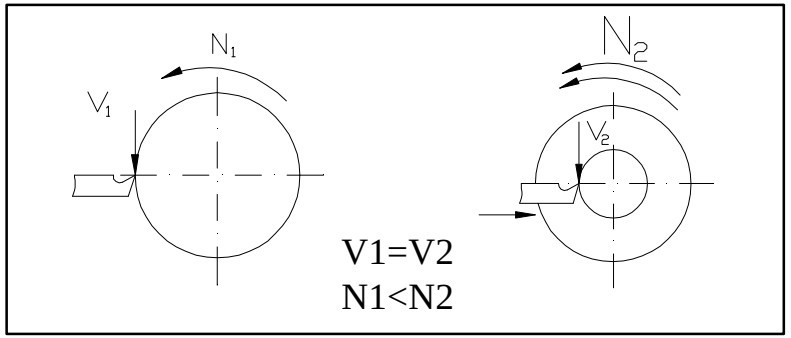
–Turning speed (N):

It is the angular speed of the part.

The relationship between them is: $V=2*\pi*R*N/1000$

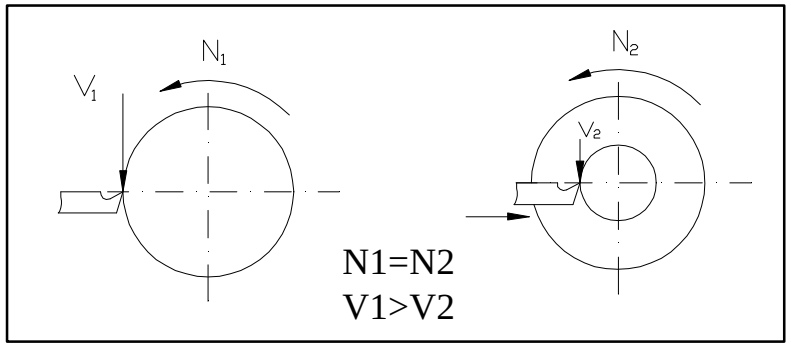
Τηε ΧΝΧ οφφερσ τωο ωαψσ το οπερατε ωιτη τηε σπινδλε:

CSS: Constant Surface Speed.



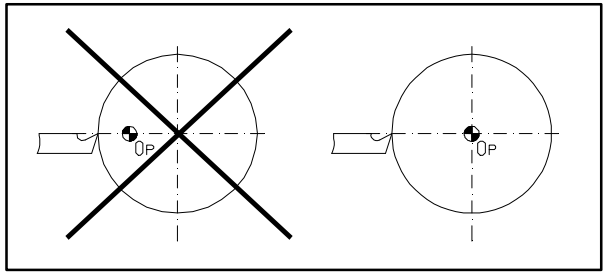
Τηε ΧΝΧ μαινταινσ τηε χυττινγ σπεεδ (ς) χονσταντ ωηιλε ϖαρψινγ τηε τυρνινγ σπεεδ (Ν).

ΡΠΜ: Ρεϖολυτιονσ περ μινυτε.

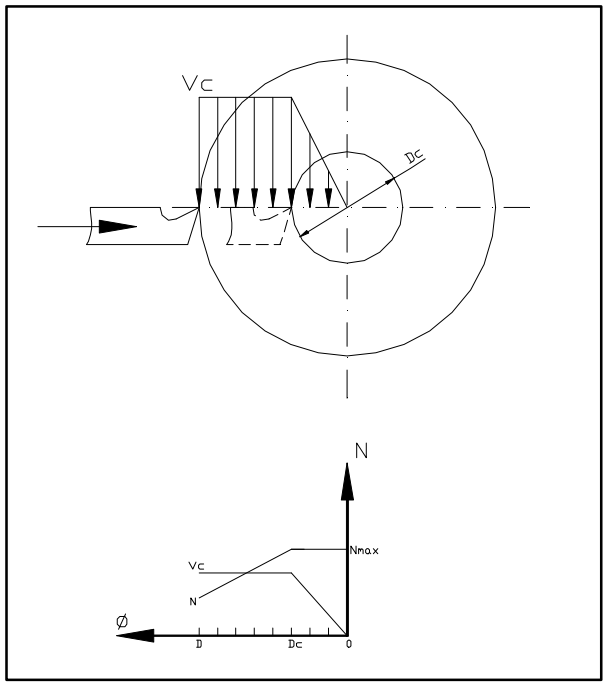


Τηε ΧΝΧ μαινταινσ τηε τυρνινγ σπεεδ (Ν) χονσταντ ωηιλε ϖαρψινγ τηε χυττινγ σπεεδ (ς).

To work at CSS, two things must be borne in mind:



The part zero must be at the part's turning axis so that the calculated turning speed is the same as the best cutting speed.



The maximum turning speed must be programmed because the turning speed increases as the diameter decreases and a particular speed should not be exceeded on parts with a large diameter.

The CNC works at Constant Surface Speed (Vc) and, starting at diameter Dc (when N=Nmax), it works at constant turning speed (N).

1.8 Axis feedrate.

The feedrate units are set by the machine manufacturer, being:

- mm/rev: The axis feedrate varies depending on the spindle speed.
- mm/min: The axis feedrate is independent from the spindle speed.

NOTE

It is recommended to work at Constant Surface Speed (CSS) and with the feedrate in mm/rev. This way, the tool lasts longer and the resulting part finish is better.

Chapter 2

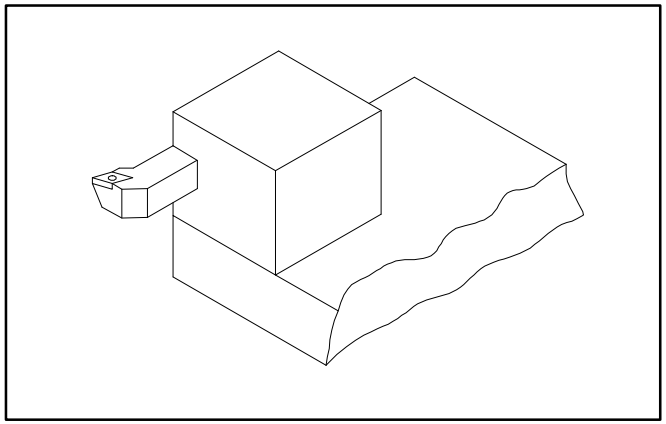
Theory on tools

This chapter describes:

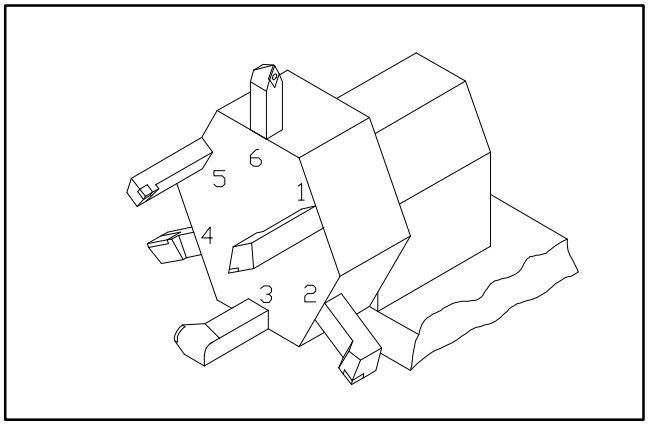
- What the tool turret is.
- What the tool table is and what information it contains.
- What tool presetting is.
- Defects due to errors in the tool table.
  - > Due to wrong tool calibration.
  - > Due to wrong tool location codes (tool shapes).
  - > Due to wrong tool radius values.

2.1 The tool turret.

The tools this CNC can use are placed on the tool turret. This turret may have either a manual or automatic tool changer. When manual, the tool change is carried out like on a conventional machine. When automatic, all the tools will be placed on the turret and the CNC will rotate the whole turret to put the tool at the work position.



Turret with manual tool change



Turret with automatic tool changer

2.2 Tool table.

The tool table contains tool information such as their position on the turret, dimensions, etc.

When changing the tool, the CNC takes this tool information.

The information kept in the tool table refers to: T, D, X, Z, I, K, A, B, C, R, F:

T: Tool number.

D: Table storing tool data.

The type of tool insert being selected must also be defined:

 Diamond shaped insert

 Threadcutting tool

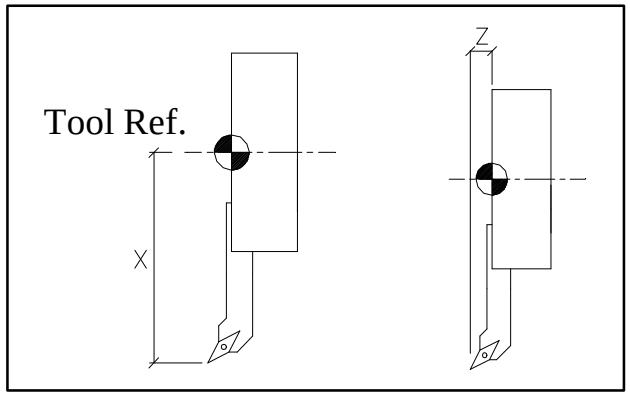
 Square insert

 Round insert

 Live tool

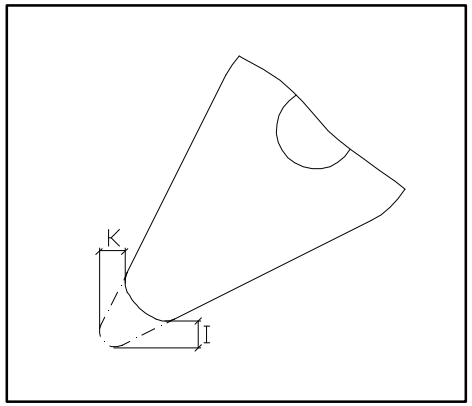
X: Tool length (in radius) along the X axis

Z: Tool length along the Z axis.



I: Tool wear along the X axis.

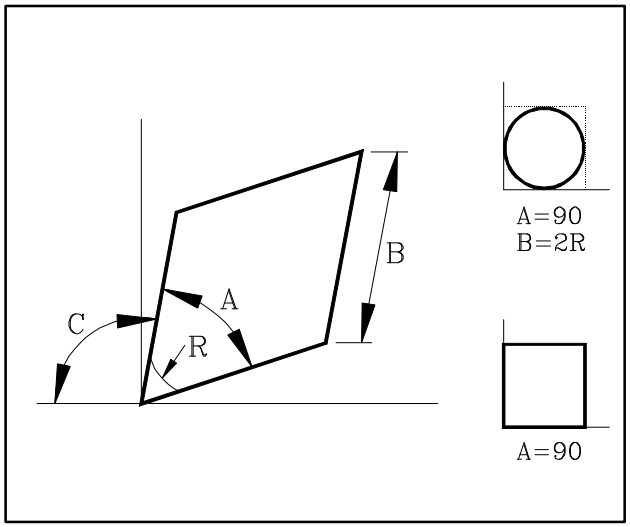
K: Tool wear along the Z axis.



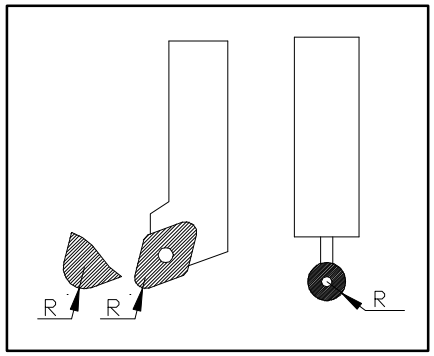
A: Cutter angle.

B: Cutter width.

C: Cutting angle.

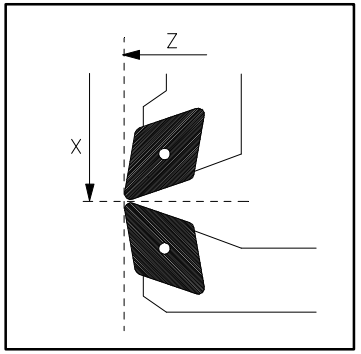


R: Tool radius.

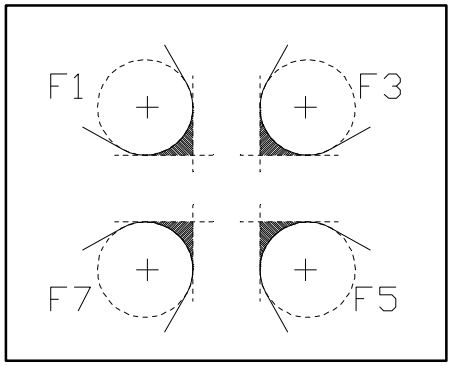


F: Location code or tool shape as it has been calibrated.

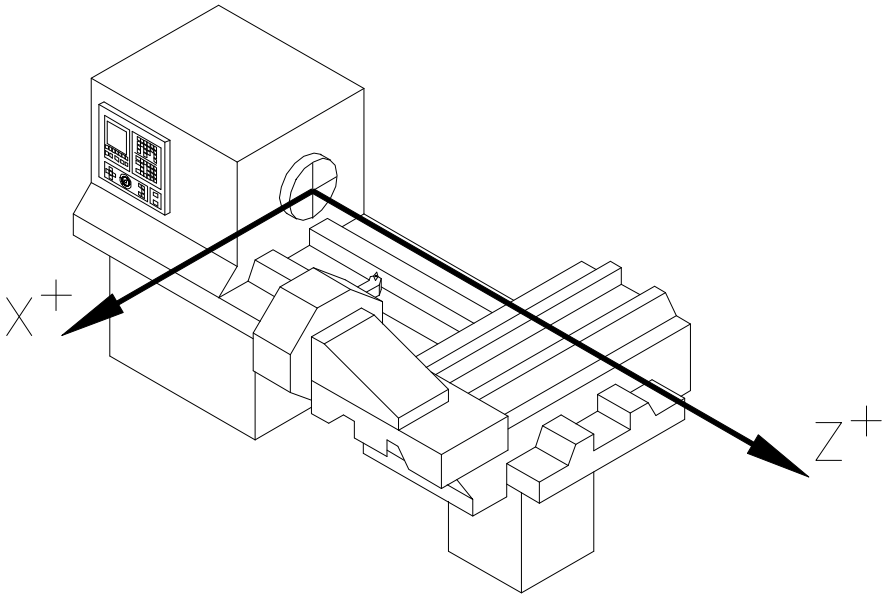
Once the tool dimensions are known;



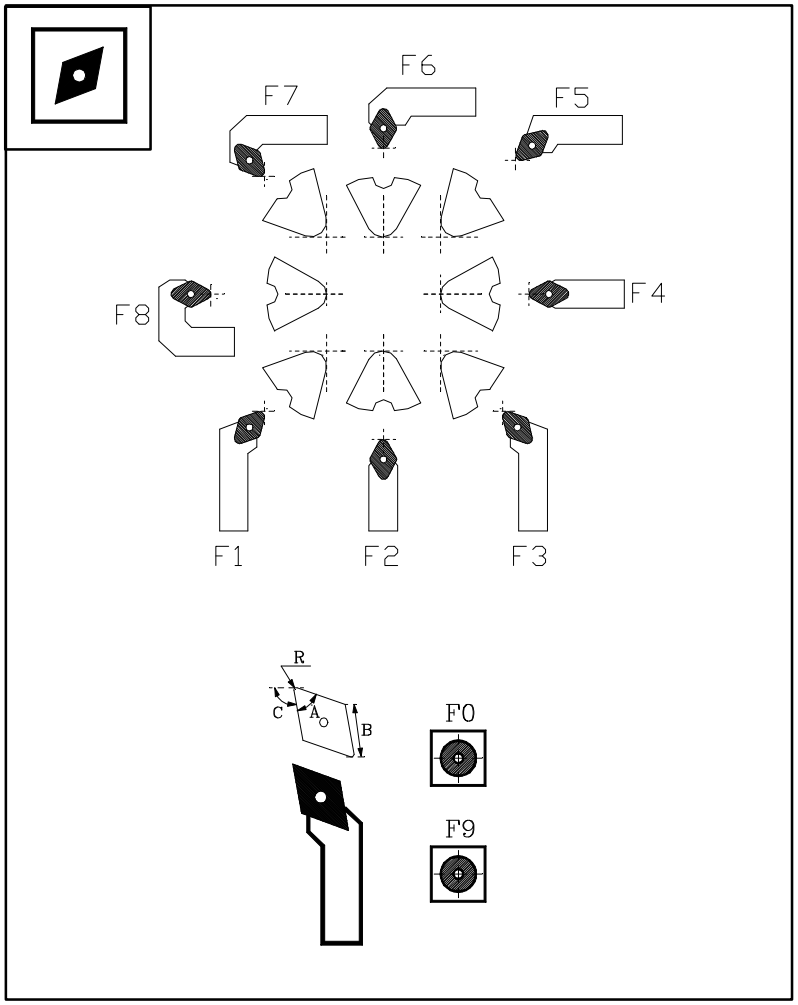
The CNC must know which is the calibration point for that tool (location code) to compensate for the shaded area (radius compensation).

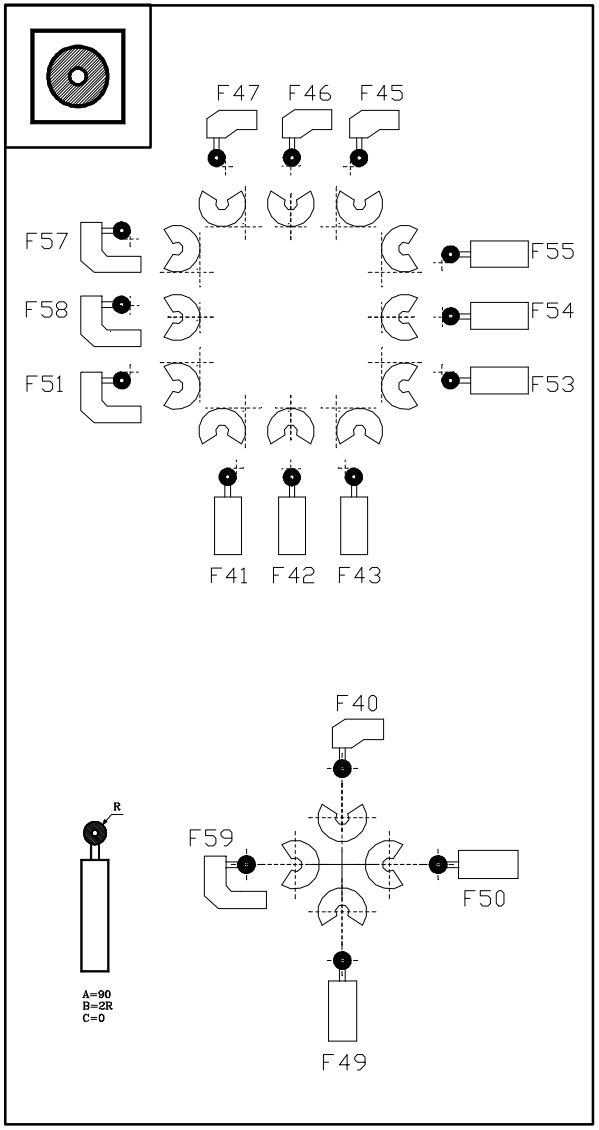


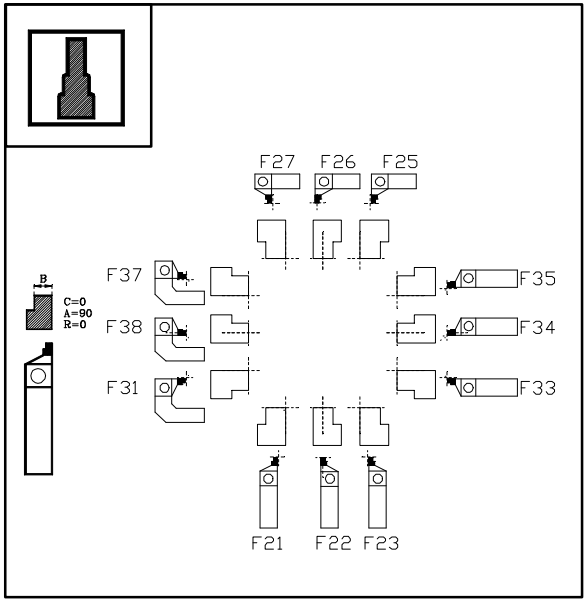
The location code depends on the orientation of the machine axes.

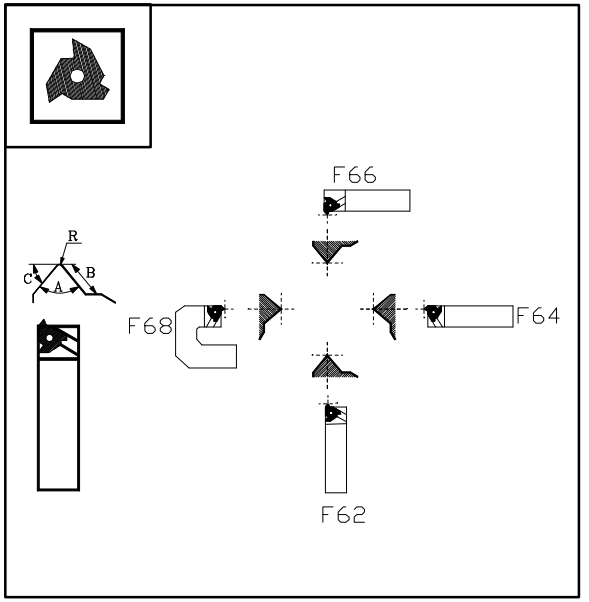


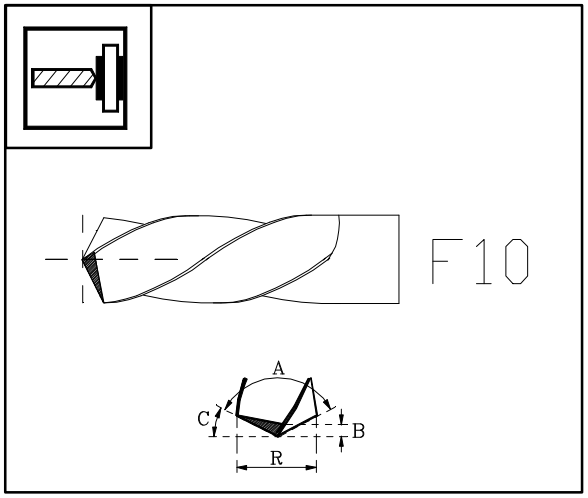
Most common location codes.

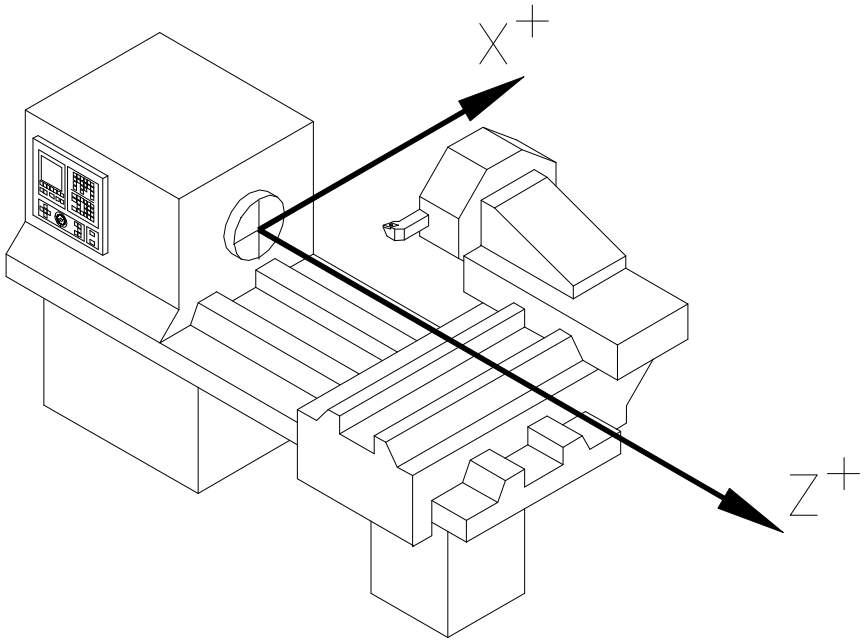



F47
F46
F45
F57
F55
F58
F54
F51
F53
F41
F42
F43
F40
R
F59
F50
A=90
B=2R
C=0
F49

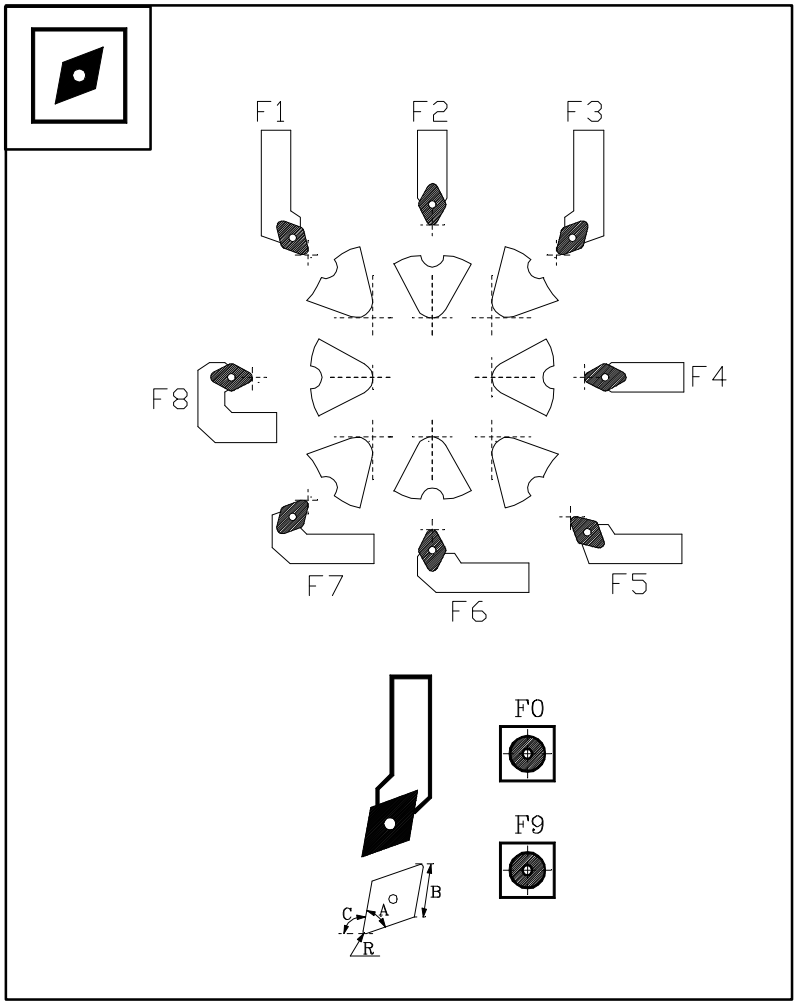

F27
F26
F25
B
C=0
A=90
R=0
F37
F35
F38
F34
F31
F33
F21
F22
F23

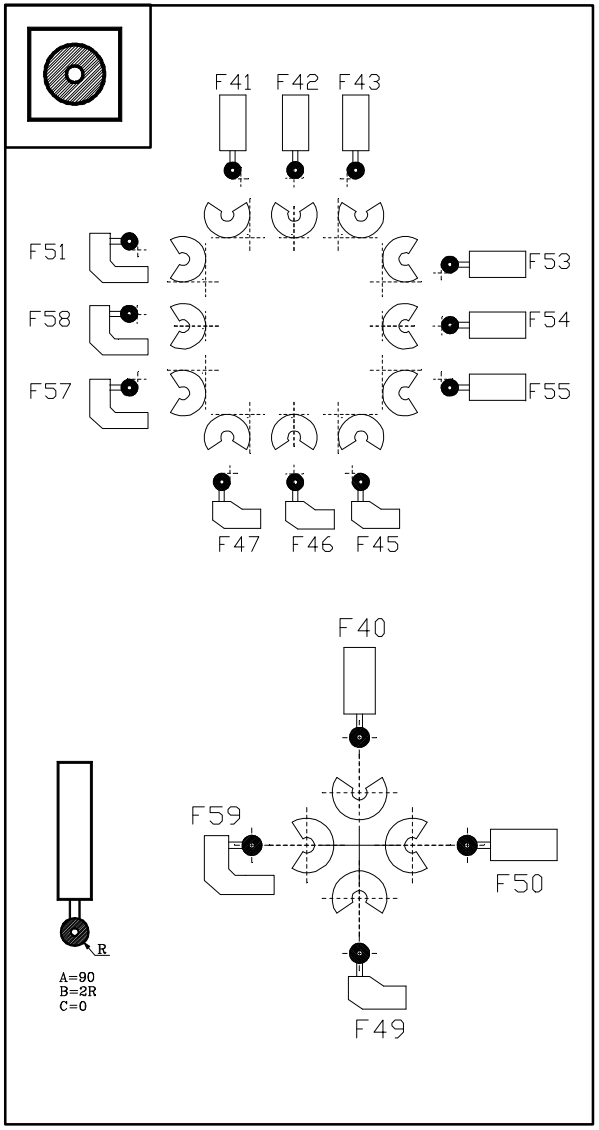

F66
R
C
A
B
F68
F64
F62

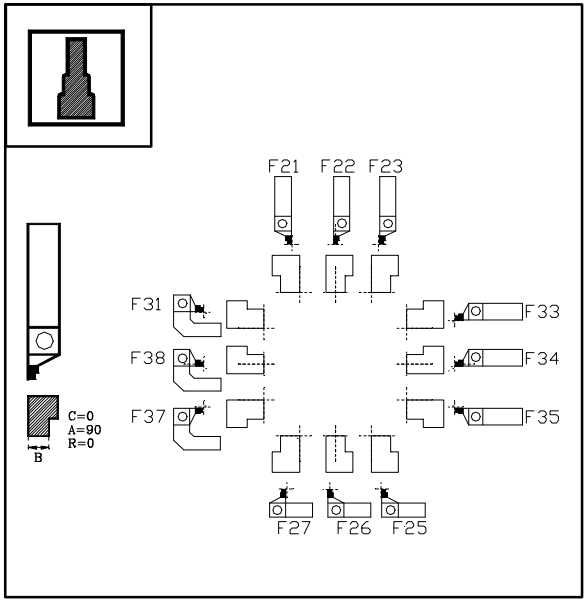

F10
A
C
B
R

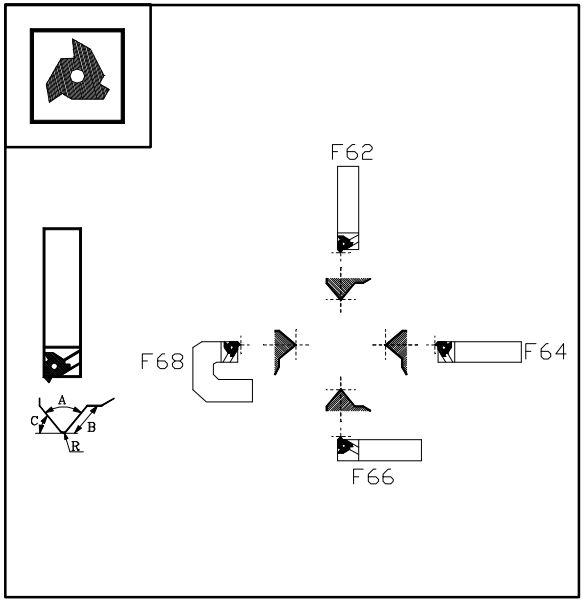


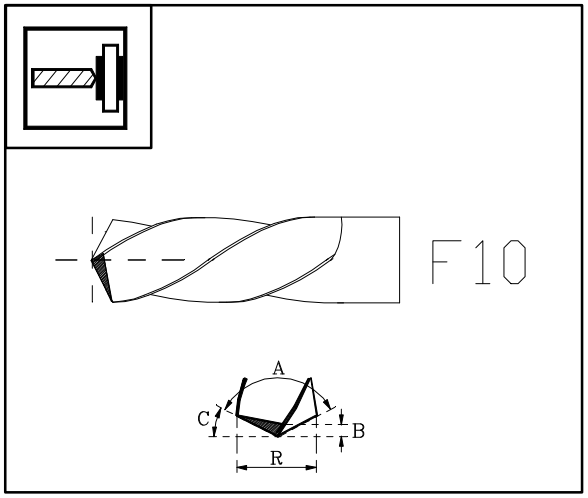
Most common location codes.




F41
F42
F43
F51
F53
F58
F54
F57
F55
F47
F46
F45
F40
F59
F50
R
A=90
B=2R
C=0
F49

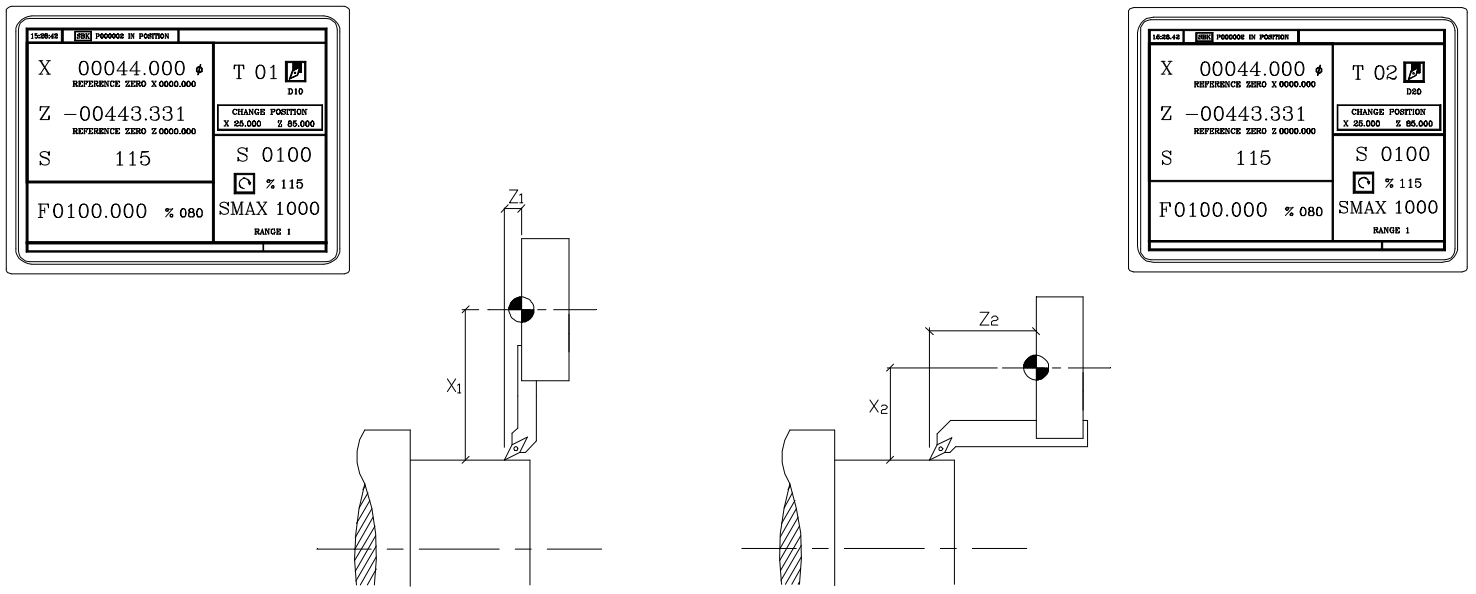

F21
F22
F23
F31
F33
F38
F34
F37
F35
C=0
A=90
R=0
B
F27
F26
F25


F62
F68
F64
A
C
B
R
F66


F10
A
C
B
R

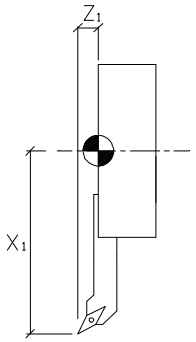
2.3 Tool calibration.

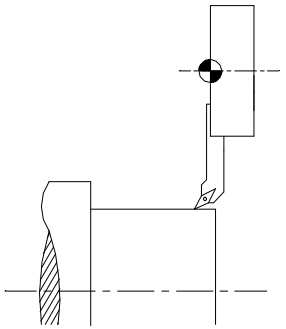
By calibrating a tool, we indicate to the CNC the tool dimensions. It is essential to carry this operation out properly for obtaining the parts with the right dimensions and for controlling the same point after changing a tool.



Different tool dimensions, same point.

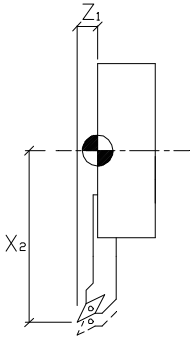
DEFECTS DUE TO WRONG LENGTH CALIBRATION

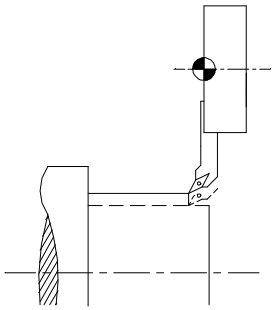




X1: Real dim.
Z1: Real dim.

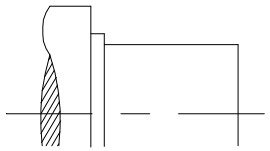
Right part dimensions



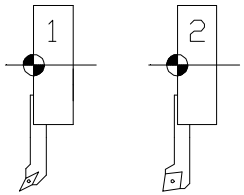


X2: Wrong dim.
X2<X1(Real dim.)

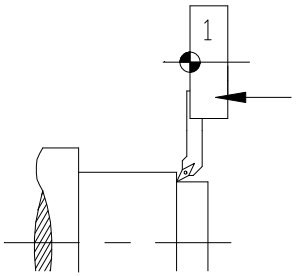
Wrong part dimensions

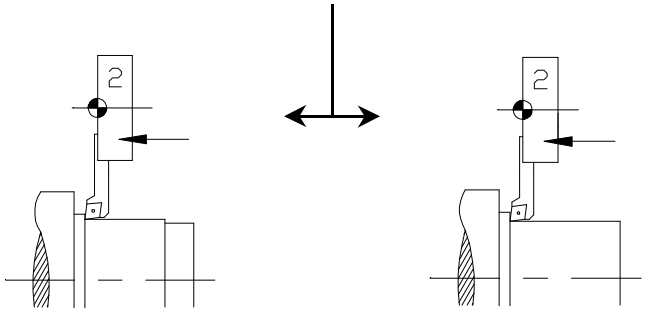


Part to be machined



Tools



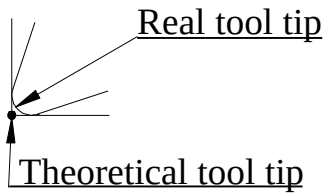


Wrong machining
Tools calibrated wrong

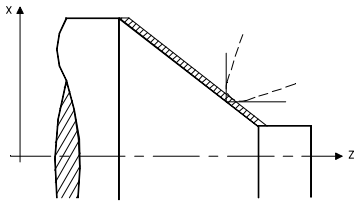
Right machining
Tools calibrated right

DEFECTS DUE TO WRONG LOCATION CODES

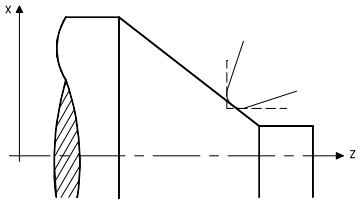
The tool has a real tip and another theoretical tip.



When working with the theoretical tool tip, there are machining ridges left. To avoid this, the CNC works with the real tool tip.

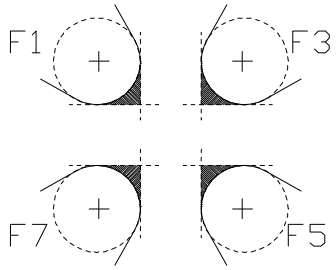


Working with theoretical tool tip

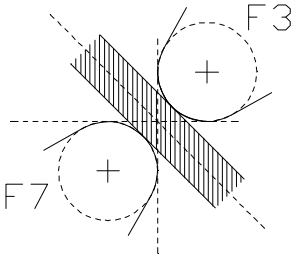


Working with real tool tip

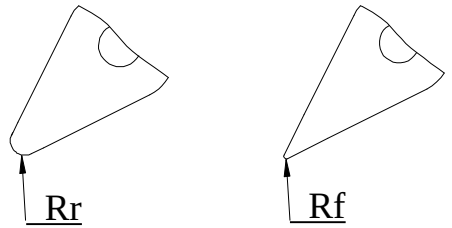
To work with the theoretical tool tip, the CNC needs to know which is the tool calibration point (location code) in order to compensate for the shaded area (radius compensation).



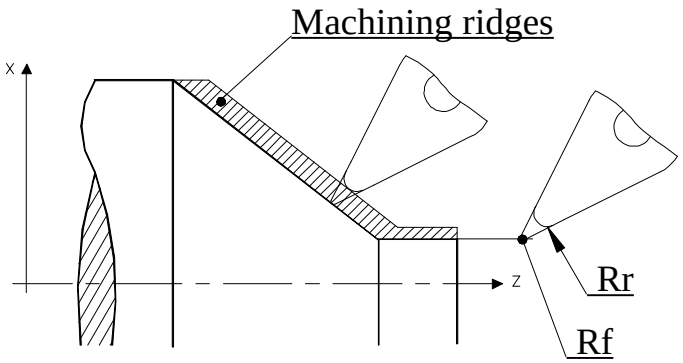
If the wrong location code is entered, the error increases instead of being compensated.



DEFECTS DUE TO WRONG RADIUS VALUES



Rr: Real radius.
Rf: Wrong radius (false).



There is residual stock due to the radius difference.

Chapter 3

Hands-on training

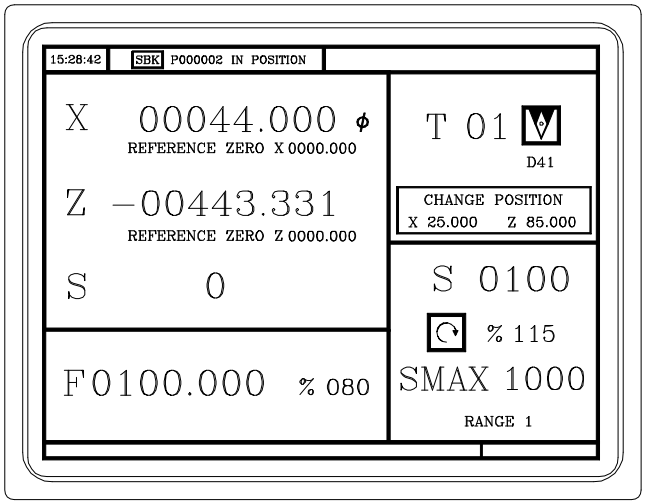
This chapter describes:

- The keyboard and the screen
- How to carry out a “Home Search”.
  - > Maintaining the part zero.
  - > Without maintaining the part zero.
- How to operate with the spindle.
  - > What the speed ranges (gears) are.
  - > How to operate at CSS or in RPM.
- How to jog the axes. (Handwheels, incremental and continuous JOG, etc.)
- How to handle tools.
  - > Types of tool changer. (Manual or automatic).
  - > Tool calibration.
  - > Tool table.
  - > Tool change position.
- How to check the tool calibration.

3.1 Screen and keyboard description.

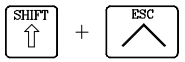
3.1.1 Power-up.

On power-up, the CNC will display the following screen.



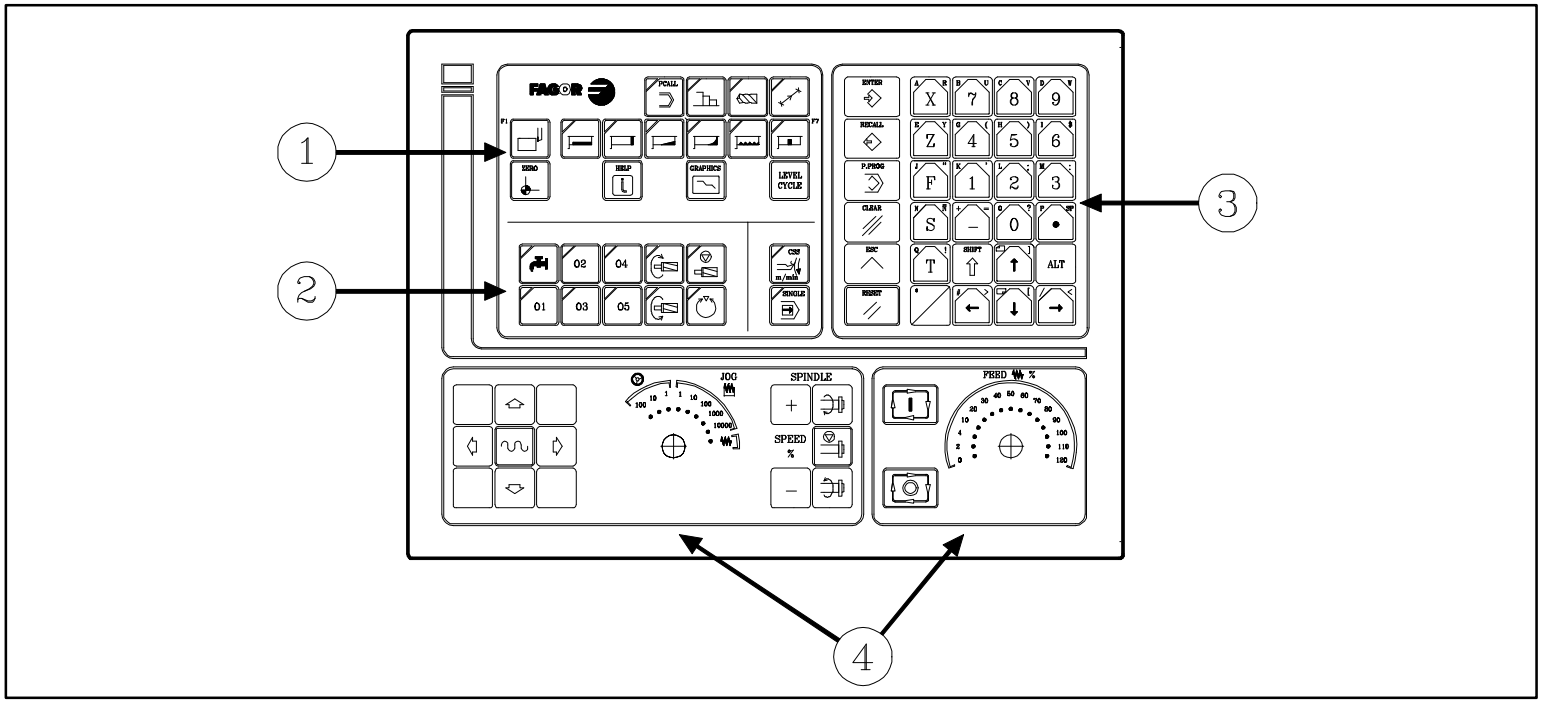
Screen for the 8055TC mode.

If this screen is not displayed, it is because the CNC is in 8055T mode. To enter in 8055TC mode, press:



NOTE: Refer to the Operation Manual Chapter 2 Section 2.3

3.1.2 Keyboard description.



1.- Keys to define the machining operations.

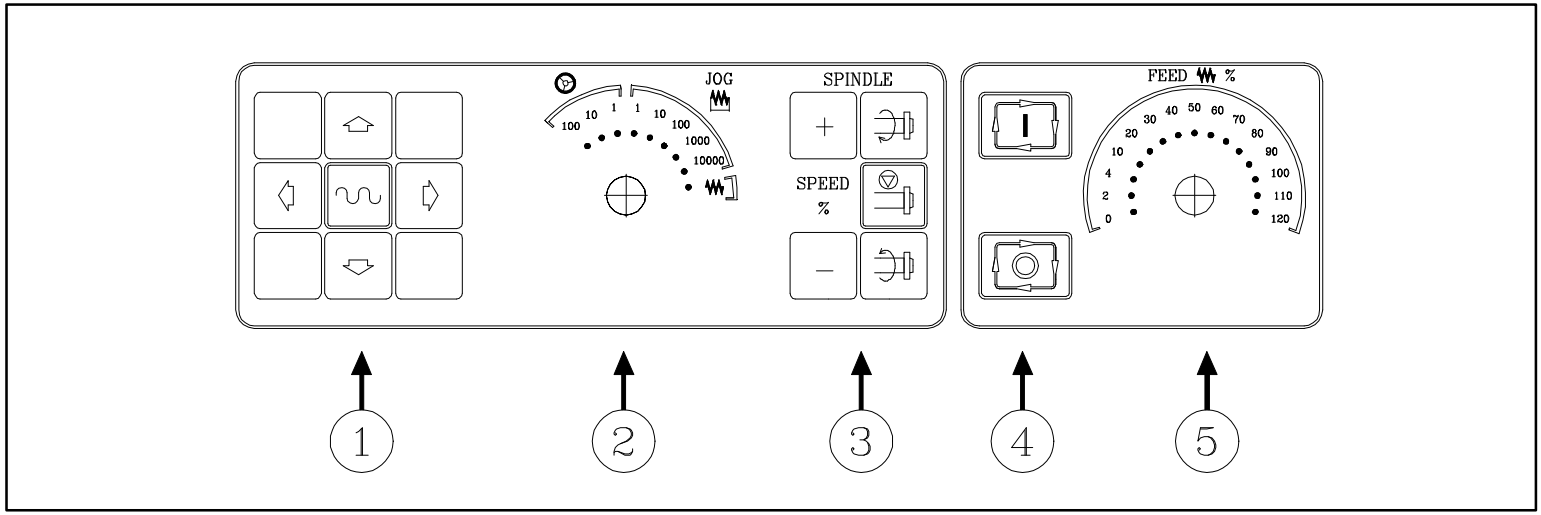
2.- Keys for external devices: Live tool and work modes for the spindle (RPM/CSS).

3.- Alphanumeric keyboard and command keys.

4.-Operator panel.

NOTE: Refer to the Operation Manual Chapter 2 Section 2.1

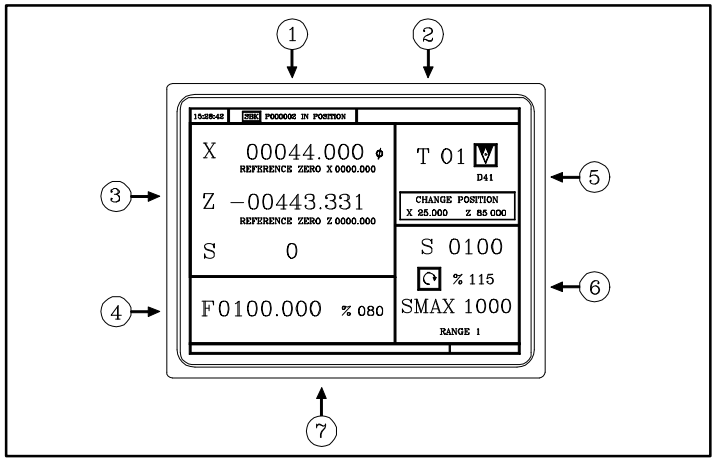
Description of the operator panel.



1. Axes jogging keys.
2. Work mode selector. (Continuous JOG ( ), incremental JOG ( ) or with handwheel ( )).
3. Selection of spindle turning direction ( ) and start-up. Spindle speed override percentage ( ).
4. Keys for CYCLE START ( ) and CYCLE STOP ( ).
5. Axis feedrate override percentage.

NOTE: Refer to the Operation Manual Chapter 2 Section 2.1

3.1.3 Description of the standard screen.



1.- Time, single-block/continuous execution, program number, execution status (In position, Execution, Interrupted or Reset) and PLC.

2.- CNC messages.

3.- Tool position referred to part zero and to home. Actual (real) spindle rpm.

4.- Selected axis feedrate and applied override %.

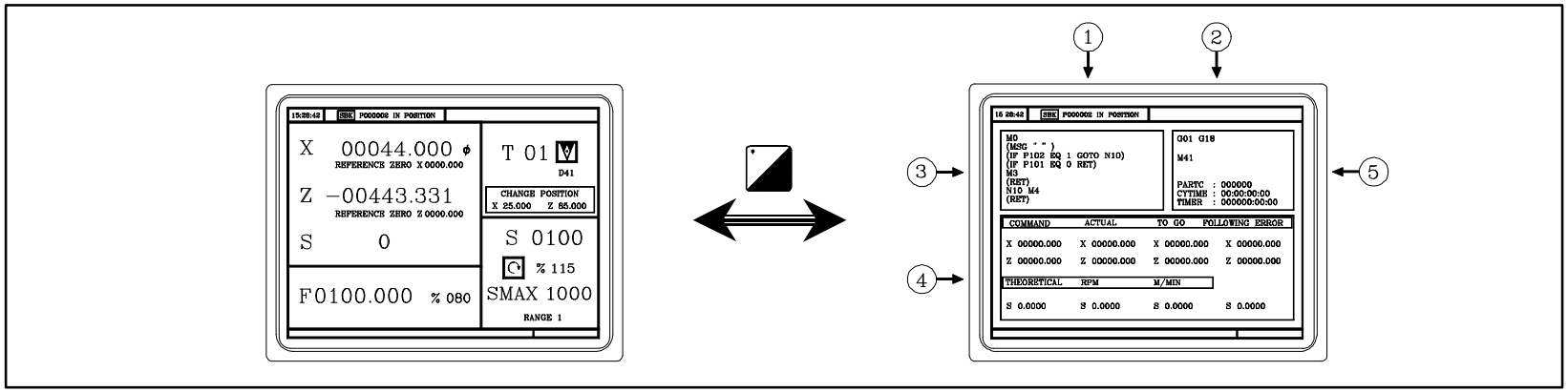
5.- Tool information. Active tool and tool change position.

6.- Spindle information. Selected speed and override percentage applied, maximum rpm and spindle status (turning clockwise, counter-clockwise or stopped) and active range.

7.- Help messages.

NOTE: Refer to the Operation Manual Chapter 3 Section 3.1

3.1.4 Description of the auxiliary screen.



1.- Time, single block/continuous execution, program number, execution status (In position, Execution, Interrupted or Reset) and PLC.

2.- CNC messages.

3.- Lines of the selected program.

4.- Axes movement information: Movement target point (COMMAND), Current tool position (ACTUAL), remaining distance (TO GO) and difference between the theoretical axis position and its actual position (FOLLOWING ERROR or axis lag).

Spindle information: programmed theoretical speed, speed in rpm, speed in m/min.

5.- Status of the active G and M functions. Number of consecutive parts executed with the program (PARTC), execution time for a part or cycle time (CYTIME), and PLC clock (TIMER).

NOTE: Refer to the Operation Manual Chapter 3 Section 3.1

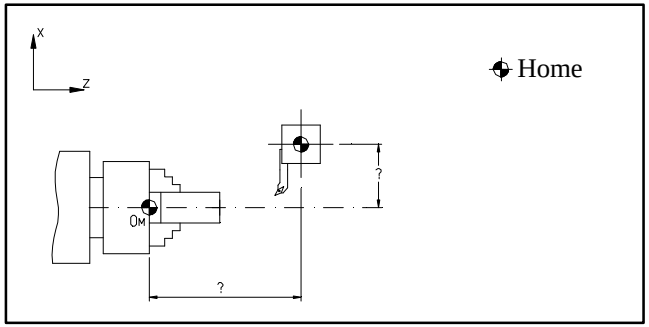
3.2 Home search.

After powering the machine up, carry out the “Home Search” just in case the axes of the machine have moved while the CNC was off. A “Home Search” can be carried out in two ways.

3.2.1 Maintaining the part zero.

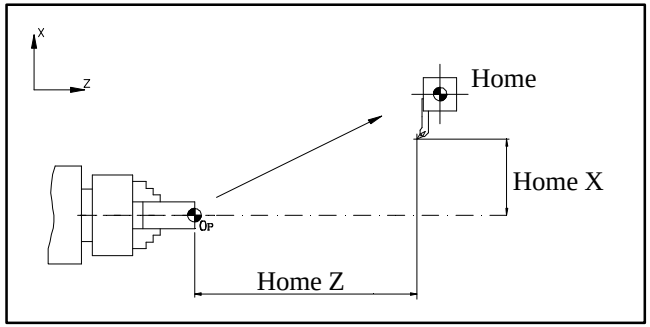
The “Home Search” is carried out on both axes at the same time.

The CNC does not know the possition of the carriages.
X?, Z? different from the displayed X, Z.



Press ZERO + I

The CNC shows the coordinates referred to part zero OP considering the X, Z dimensions of the tool.

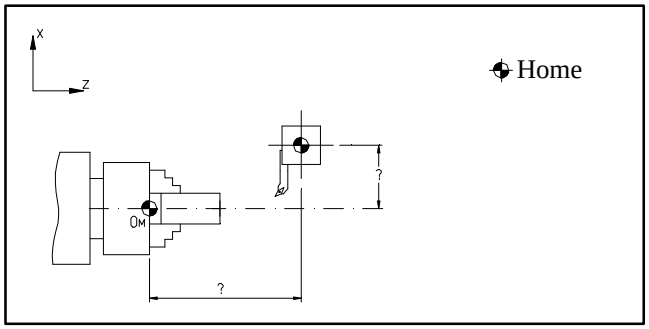


NOTE: Refer to the Operation Manual Chapter 3 Section 3.3

3.2.2 Without maintaining the part-zero.

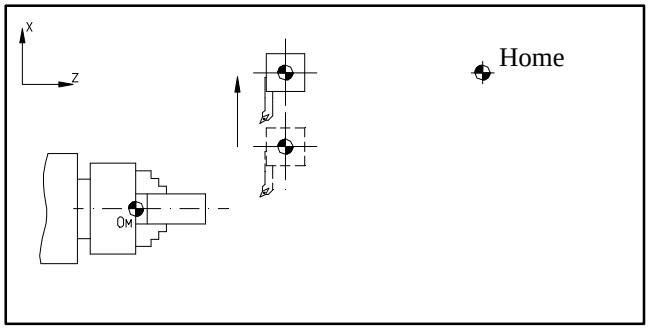
The “Home Search” is carried out on one axis at a time.

1st.-The CNC does not know the possition of the carriages. X?, Z? different from the displayed X, Z.



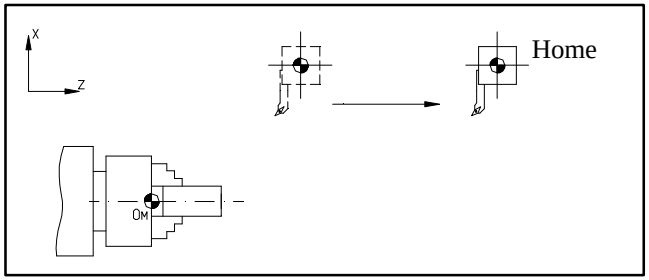
2nd.-Home search on the X axis.

Press [X] + [ZERO] + [I]

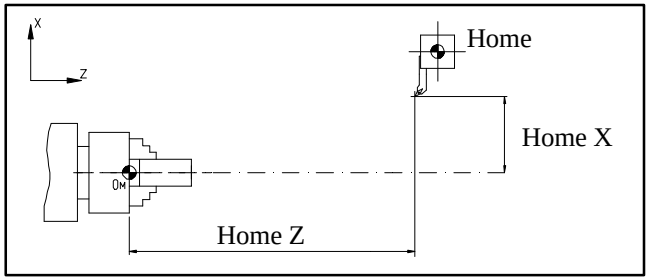


3rd.-Home search on the Z axis.

Press [Z] + [ZERO] + [I]



4th.- The CNC shows the coordinates referred to OM, considering the X, Z dimensions of the tool.



NOTE: Refer to the Operation Manual Chapter 3 Section 3.3

3.3 Spindle.

The spindle of a machine can work in two modes:

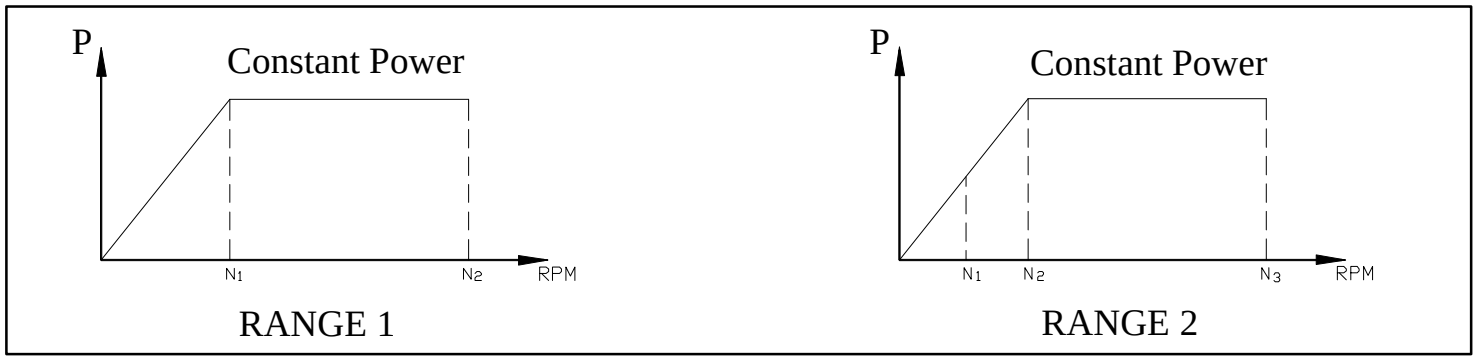
- RPM: At constant turning speed. (Section 1.7)
- CSS: At constant surface speed.(Section 1.7)

Press 

to select the work mode.

3.3.1 Speed ranges (gears).

With this CNC the machine can have a gear box. By means of RANGES, we can choose the best gear ratio for the programmed spindle speed.



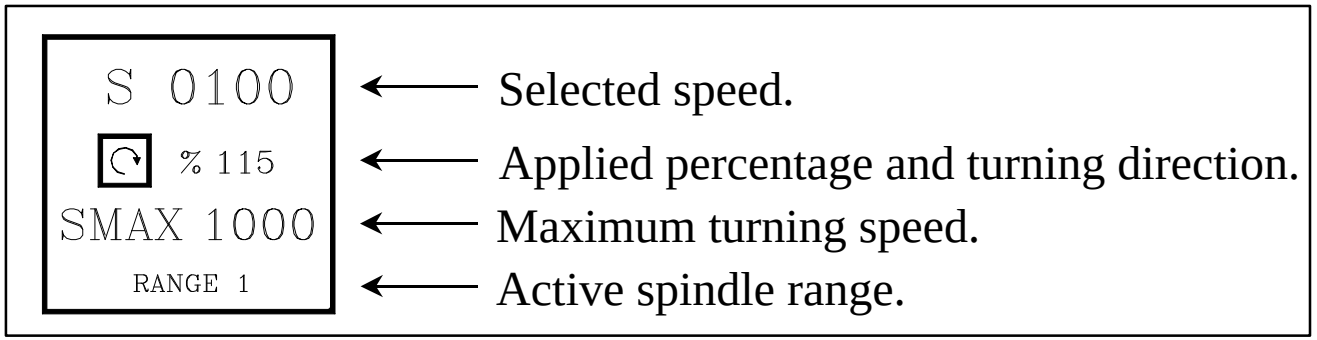
If the work speed is between N_1 and N_2, RANGE 1 should be used and if between N_2 and N_3, RANGE 2. Always try to work at constant power.

3.3.2 Work in RPM mode. (Revolutions per minute)

To select the work speed (in rpm), press:

S + (turning speed) + I

The CNC shows the following information:



Use the following keys of the operator panel to start the spindle.

Start the spindle clockwise.

Stop the spindle.

Start the spindle counter-clockwise.

\+ – Increase or decrease the override percentage applied to the spindle turning speed.

NOTE: Refer to the Operation Manual Chapter 3 Section 3.6.1

3.3.3 Work at Constant Surface Speed. (CSS)

Before programming the cutting speed, the working speed range must be selected. The CNC assumes the current range by default. Once the change is completed, enter the CSS mode and press [CSS m/min].

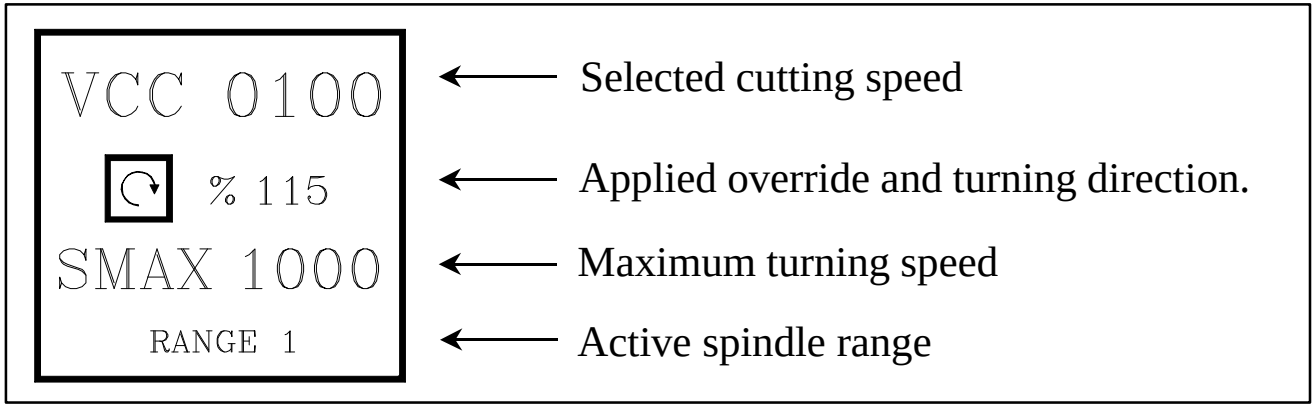
To select the cutting speed (m/min) , press:

[S] + (cutting speed) + [I]

To select the maximum turning speed (in rpm), press:

[S] + [S] + (maximum speed) + [ENTER]

The CNC shows the following information :

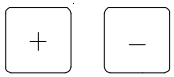


Start the spindle using the JOG keys of the operator panel.

 Spindle clockwise.

 Stop the spindle.

 Spindle counter-clockwise.

 Increases or decreases the applied override % to the turning speed.

Depending on the position of the axes, the turning speed will be different:

If X decreases, the RPM increase.

If X increases, the RPM decrease.

NOTE

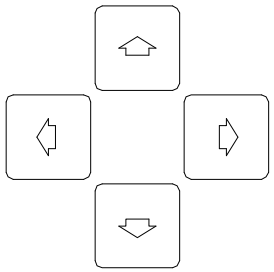
While machining an operation, NO range change will take place.

To work at constant surface speed, the tools MUST BE calibrated.

NOTE: Refer to the Operation Manual Chapter 3 Section 3.6.2/3.6.2.1

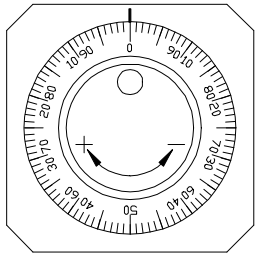
3.4 Axis jog.

To jog the axes, we will use:



JOG keys

Each key is used for moving the axis in one direction according to the axes of the machine. (Section 1.1)

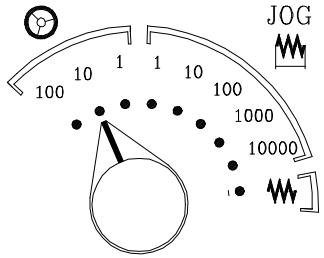


Handwheel

It can have one or two handwheels. The axes move in the turning direction of the handwheels.

To select the jog mode, use the selector switch:

Handwheel jog

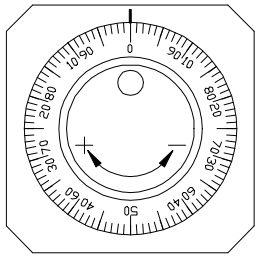


Incremental movement

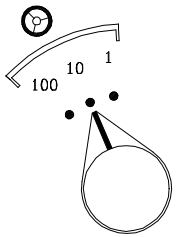
Continuous movement

3.4.1 Handwheels.

– Select the jog mode with the selector switch. (position)



Handwheel



Selector switch

SWITCH POSITION	Distance per line of the handwheel dial
1	1 micron.
10	10 micron
100	100 micron

Jog table

– Jog the axes with the handwheels.

- If the machine has 1 handwheel:

  Select an axis with the JOG keys.

  The machine moves the axis as the handwheel is being turned.
- If the machine has 2 handwheels:

  The machine moves an axis with each handwheel.

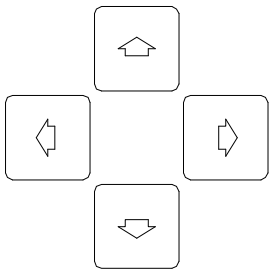
NOTE: Refer to the Operation Manual Chapter 3 Section 3.4.3

3.4.2 Incremental JOG.

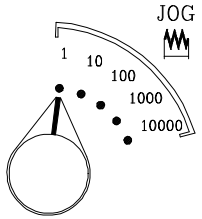
Every time a JOG key is pressed, the axis will move the selected increment at the programmed feedrate. (in rapid, if F=0).

- Select the distance to move at the selector. (position)
- Move the axes with the JOG keys.

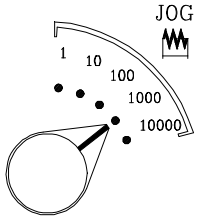
When working in mm/rev, a spindle speed will have to be already selected.



JOG keys



Actual displacement: 0.001 mm



Actual displacement: 1 mm

NOTE: Refer to the Operation Manual Chapter 3 Section 3.4.2

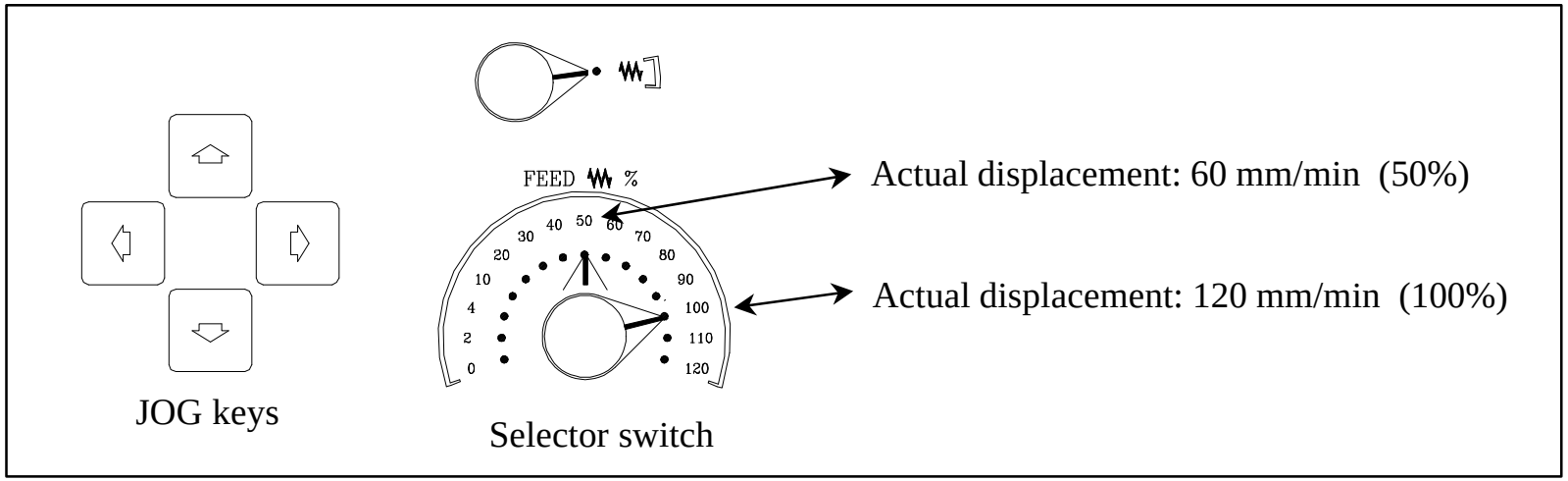
3.4.3 Continuous JOG. mm/min.

– Enter the feedrate value:

[F] + 120 <Feedrate value> + [ENTER]

– Change the % override of the axes with the selector switch in position.

– Jog the axes with the JOG keys.



NOTE: Refer to the Operation Manual Chapter 3 Section 3.4.1

3.4.4 Continuous JOG. mm/rev.

In this mode, the feedrate is a function of the spindle (either stopped or turning).

– Enter the feedrate value:

[F] + 0.1 <Feedrate value> + [ENTER]

– Change the % override for the axes feedrate with the selector switch. (position).

– Jog the axes with the JOG keys.

Depending on the spindle status:

1.- If the spindle is turning:

The CNC moves the axes at the programmed feedrate.

2.- If the spindle is stopped, but a specific S speed has already been selected:

The CNC calculates the theoretical feedrate in mm/min and moves the axes at that speed.

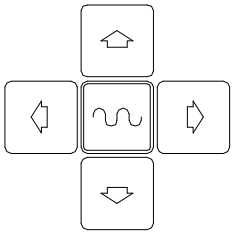
3.- If the spindle is stopped and no S speed has been selected, the axes will not move.

NOTE: Refer to the Operation Manual Chapter 3 Section 3.4.1

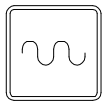
3.4.5 Rapid jog key.

– Jog the axes with the JOG keys and press the rapid jog key ( ).

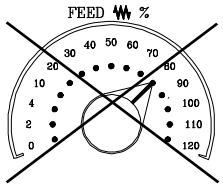
The axes move as fast as possible (set by the machine manufacturer).



JOG keys



Rapid jog key



Any position

Actual displacement: In rapid

NOTE: Refer to the Operation Manual Chapter 3 Section 3.4.1

3.5 Tools.

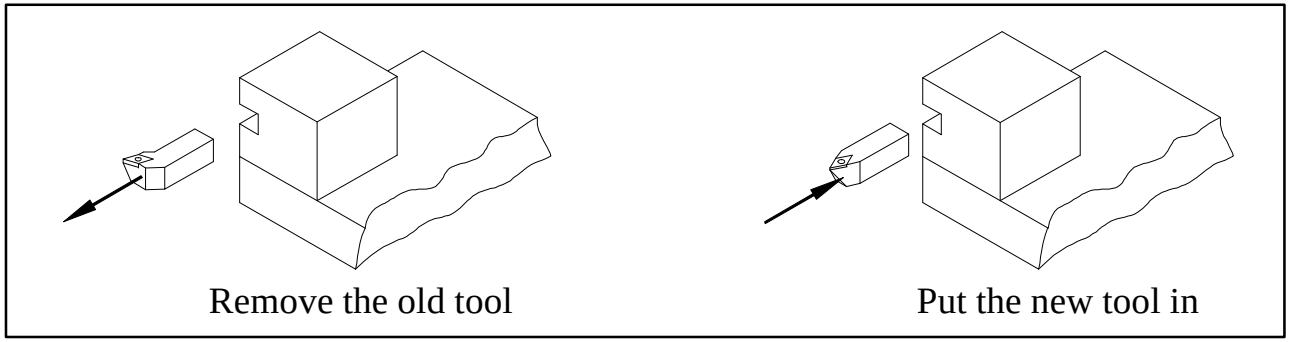
3.5.1 Tool selection.

Depending on the machine, there are two possibilities:

- Machine with manual tool changer.

  The tool change is carried out like on a conventional machine:

  – Change the tool on the machine.



Remove the old tool Put the new tool in

  – Press T

  – Enter the tool number so the CNC assumes the values of the corresponding tool table.

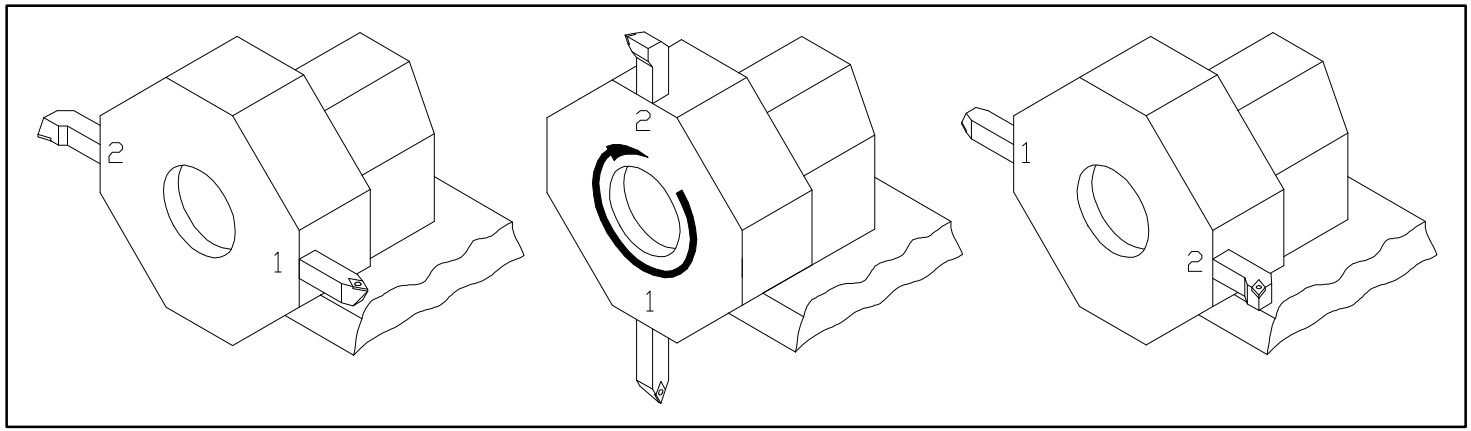
  – Press I

NOTE: Refer to the Operation Manual Chapter 3 Section 3.5.1

- Machine with automatic tool changer.

  No tool has to be removed.

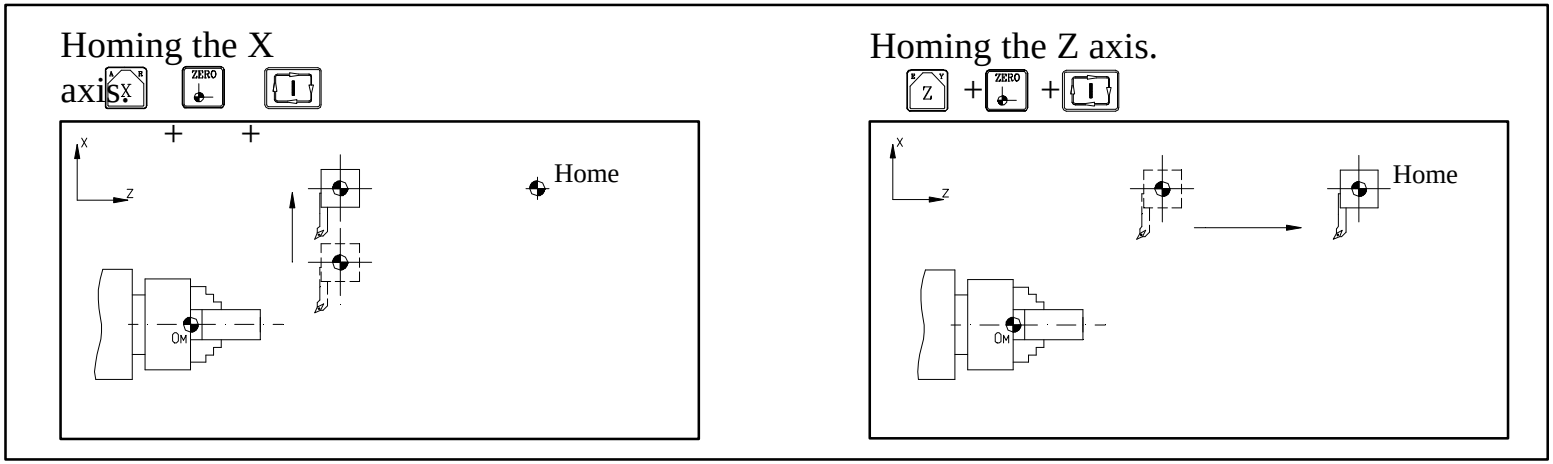
  – Press [T]
  – Enter the tool number.
  – Press [I]
  – The CNC rotates the turret until the new tool is in work position.



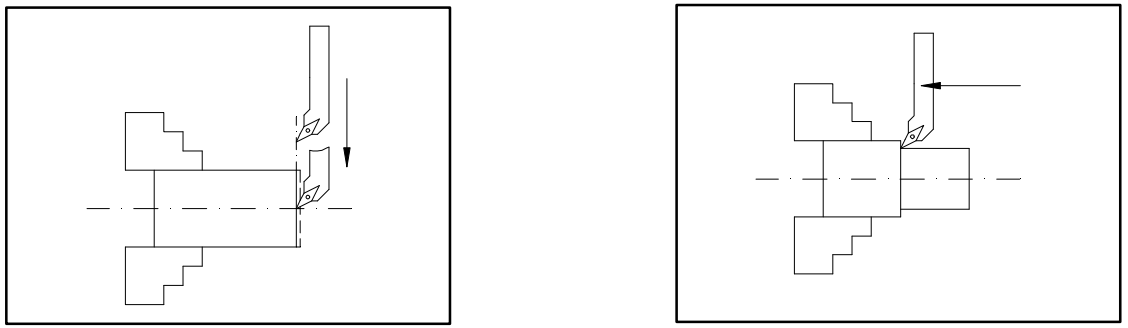
NOTE: Refer to the Operation Manual Chapter 3 Section 3.5.1

3.5.2 Tool calibration.

– Just before calibrating the tools, a “Home Search” must be carried out on all axes.

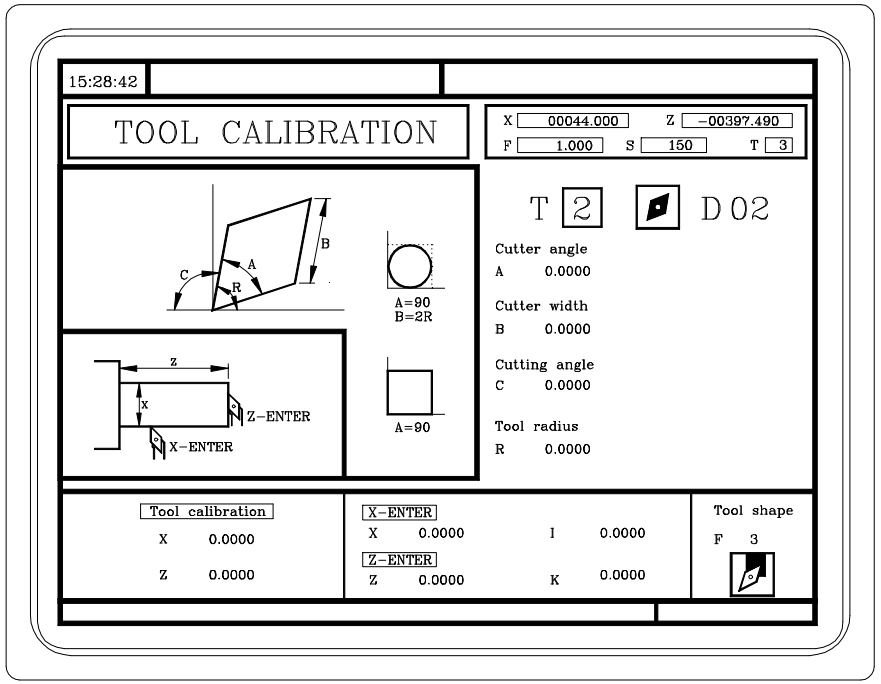


– To calibrate a tool, a part previously turned and faced is needed.



Use continuous JOG or handwheels

– Access the calibration mode. Press . The CNC shows the tool calibration screen.



Work mode.

Help graphics.

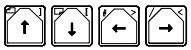
Dimensions of the part used for tool calibration.

Actual axes position Cutting conditions.

Number and type of tool as well as tool geometry.

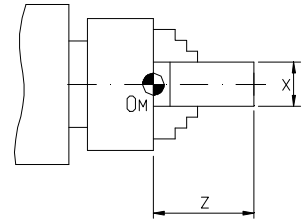
Calibration tool tip (location code).

Tool dimensions.

NOTE
To move the cursor, use:

To change icons, press .

NOTE: Refer to the Operation Manual Chapter 3 Section 3.5.2

1.- Measure the part.



Part dimensions

– Go to the tool calibration window.
– Enter the X a value.
– Enter the Z value.

2.- Start the spindle.

3.- Select the tool to be calibrated.

[T] + (tool number) + [I]

4.- Jog the axes until touching the part along the X axis. Press: [Z] + [ENTER]

The CNC shows the tool length along the X axis.

5.- Jog the axes until touching the part along the Z axis. Press: [X] + [ENTER]

The CNC shows the tool length along the Z axis.

6.- Enter the rest of the data (angle, width, radius and location code).

The I, K values are set to zero when calibrating.

To calibrate another tool, repeat steps 3, 4, 5 and 6.

NOTE: Refer to the Operation Manual Chapter 3 Section 3.5.2.2

3.5.3 How to change any data on the tool table.

To change the values (T, D, A, B, C, R, Location code, I, K), enter the calibration mode and press:

[T] + (Tool number) + [RECALL]

The CNC shows the data for that tool.

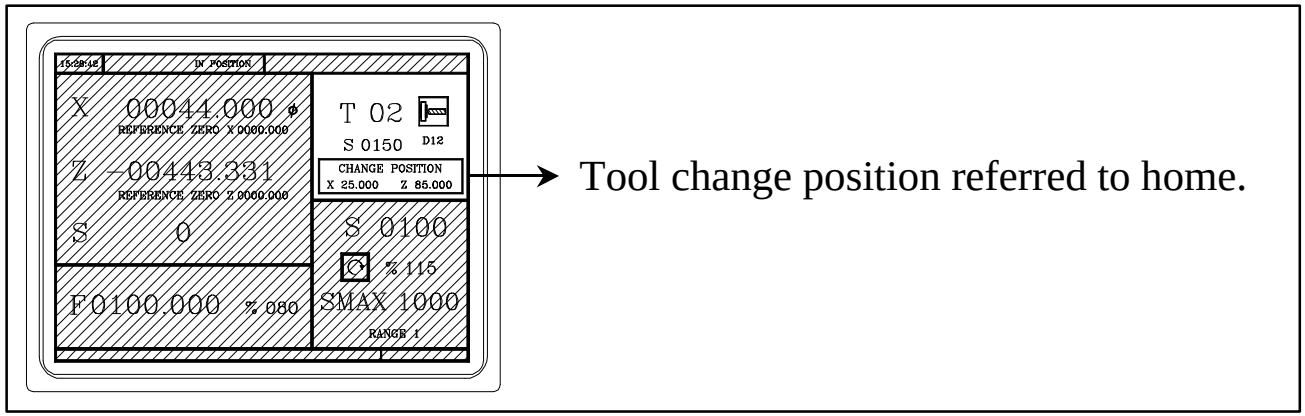
- To change the data, place the cursor over the value to be modified and key in the new value.
- To change the icons of tool type and location code, place the cursor over the icon and press [ENTER]. Once selected, press [◢]

To quit the calibration option, press [ESC].

NOTE: Refer to the Operation Manual Chapter 3 Section 3.5.2.1

3.5.4 Tool change position.

The machine manufacturer may allow selecting the tool change position.



Enter the X and Z coordinates of the tool change position.

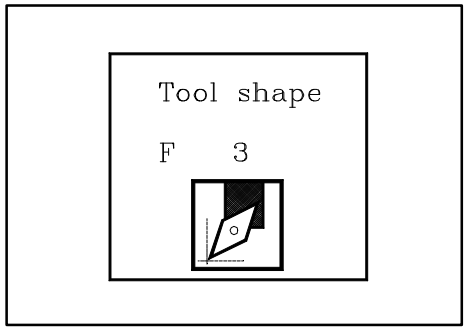
- [T] + [X] + (X Value) + [ENTER]
- [T] + [Z] + (Z Value) + [ENTER]

When a tool change is required and if the machine manufacturer has set it this way, the CNC will move the axes to this position for a tool change.

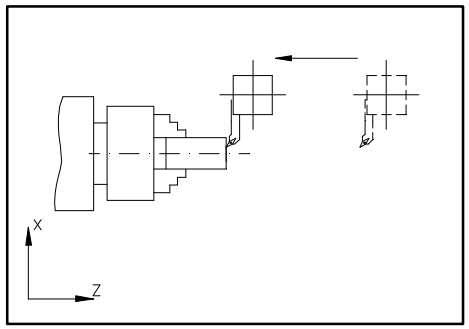
NOTE: Refer to the Operation Manual Chapter 3 Section 3.5.1.1

3.6 Checking for proper calibration.

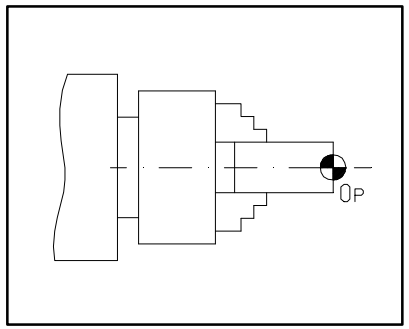
– Preset the part zero.



Select a tool
e.g. Location code 3



Approach the tool along Z.
Press Z + 0 + ENTER



Withdraw the tool
Part zero position.

– Start the spindle and touch the part diameter with several tools while checking the value on the screen.
– The tools are different but the value on the screen must be the same.

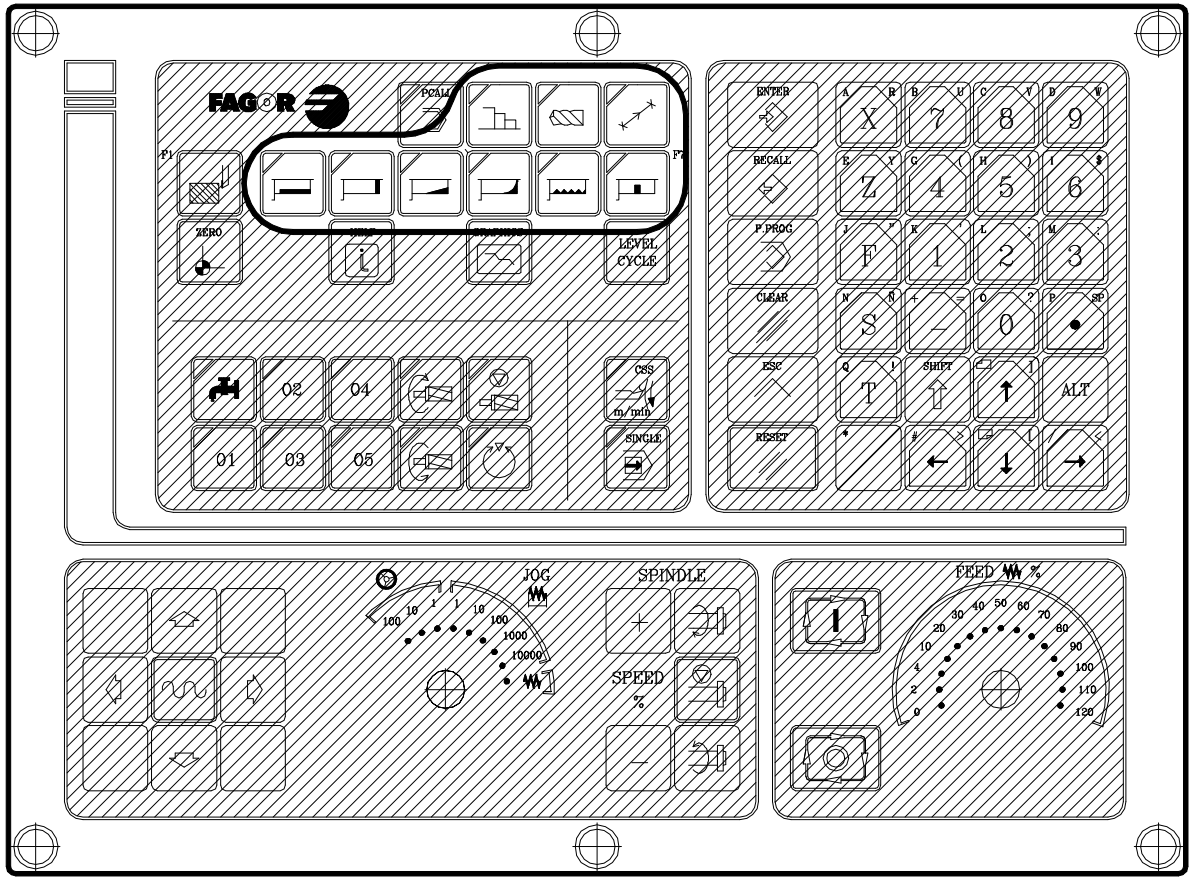
Chapter 4

Automatic Operations

This chapter describes:

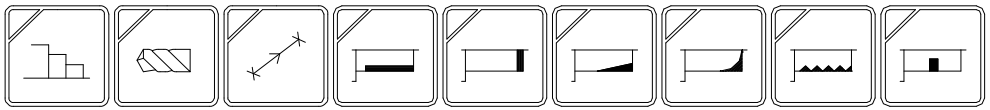
- Which are the keys associated with the automatic operations.
- Which are the various work modes.
- Taper turning example.
  - > How to edit the parameters of the operation and what they mean.
  - > How to simulate an operation and which are the graphic parameters.
  - > How to execute an operation.
    - – Tool inspection.
    - – Tool wear compensation.

4.1 Operation keys.



Layout of the automatic function keys

Operation keys:



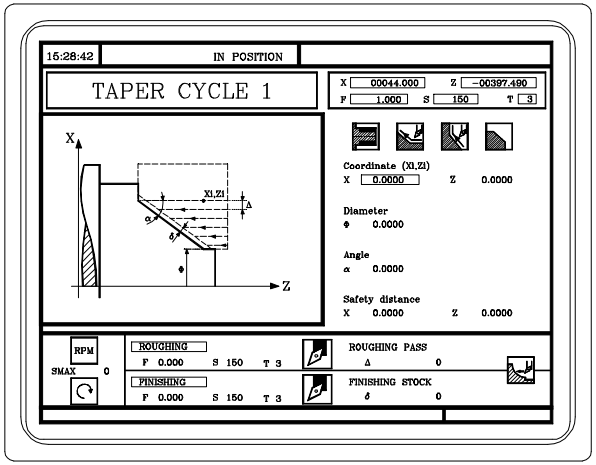
Grooving.
Threading.
Rounding.
Taper turning.
Facing.
Turning.
Positioning.
Drilling/Tapping.
Profile.

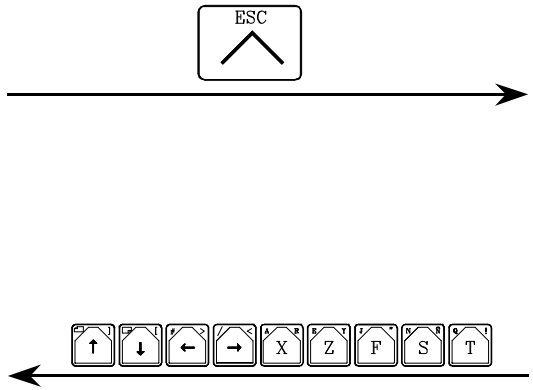
LEVEL CYCLE Cycle level selection within an operation.

4.2 Work modes.

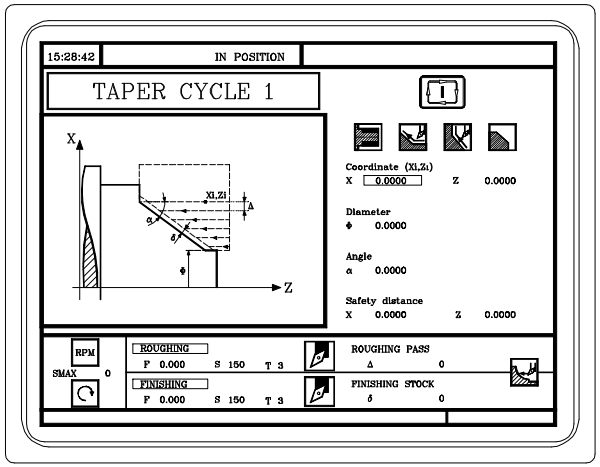
There are 2 work modes:

Edit Mode





Execution mode



Editing the parameters of the operation or cycle.

Simulation of an operation or cycle.(GRAPHICS)

Simulation of an operation or cycle. (GRAPHICS)

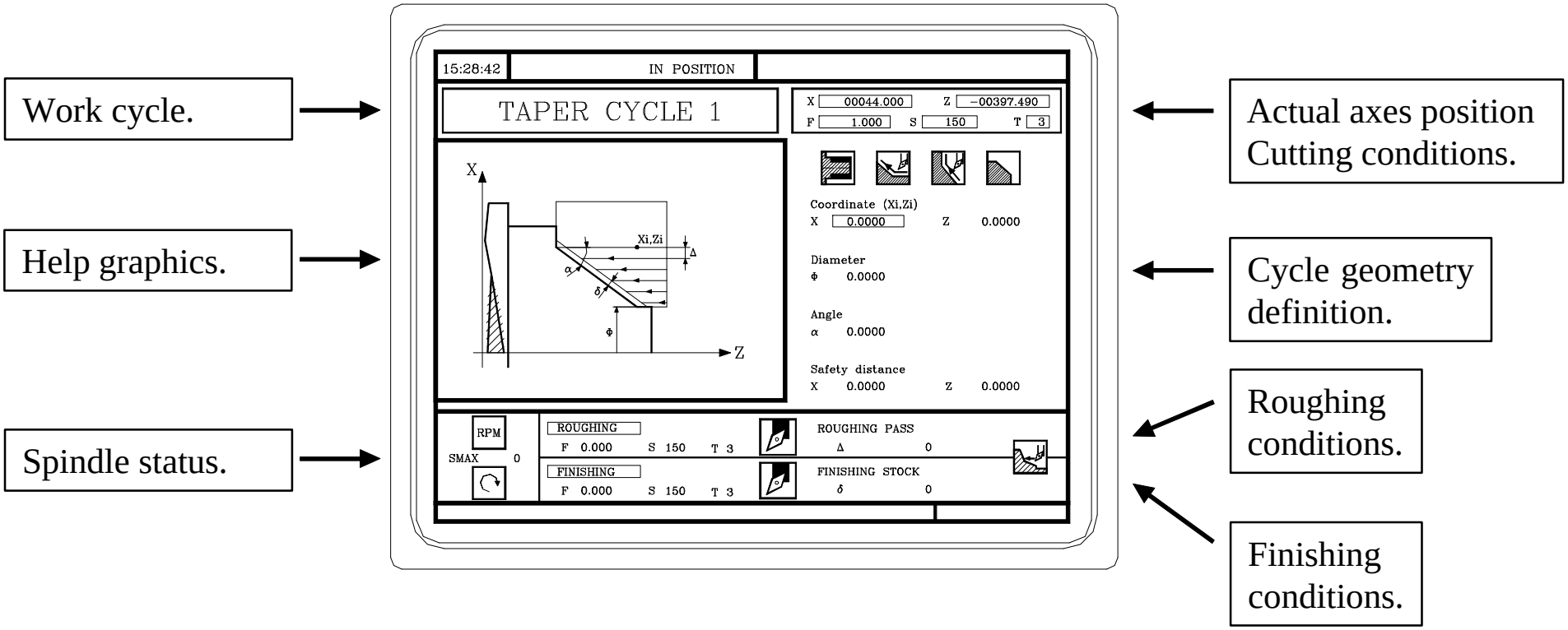
Execution of an operation or cycle. (I)

NOTE: Refer to the Operation Manual Chapter 4 Section 4.2

4.3 Example of an automatic operation. Taper turning.

4.3.1 Edit an operation.

– Select the turning operation. Press  .



– Use the  key to select the cycle level to be executed.
(Only in certain operations).

– Set the operation data.

To select an icon (symbol), data or coordinate:

- Use the ↑ ↓ ← → keys to move the cursor.
- Press X or Z. The CNC selects the first coordinate of the axis. Press it again to select the second coordinate. Press it again to select the data for safety distance.
- Press F . The CNC selects the roughing feedrate. Press it again to select the finishing feedrate.
- Press T. The CNC selects the roughing tool. Press it again to select the finishing tool.
- Press S . The CNC selects the roughing “S” data. Press it again to select the finishing “S” data. Press it again to select the maximum spindle speed.

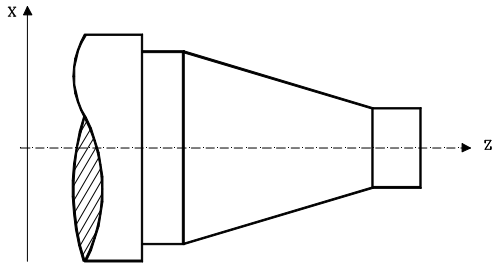
After making this selection:

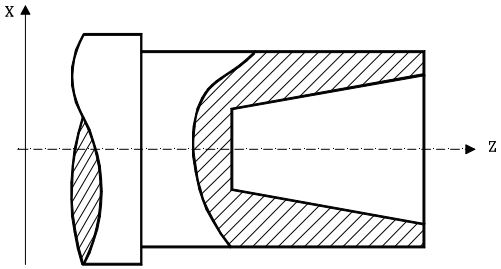
- If it is a data or a coordinate, key in the new value and press ENTER.
- If it is an icon, press ◢ until the desired one is selected and then ENTER.

NOTE: Refer to the Operation Manual Chapter 4 Section 4.1

The information shown by the icons is:

•Type of taper.



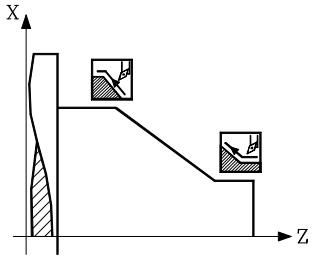


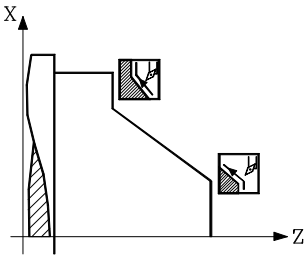
 :Outside taper

 :Inside taper

When changing the type of taper, the graphic assistance also changes.

•Type of section before and after the taper turning.

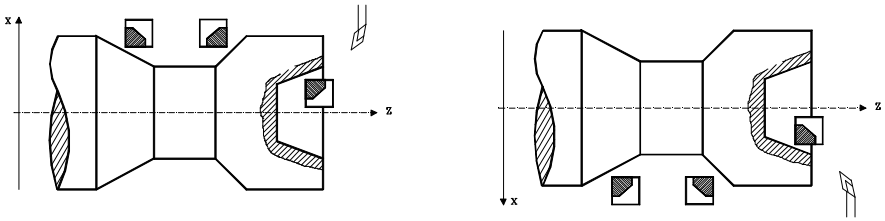




When changing the type of section, the graphic assistance also changes.

•Work quadrant

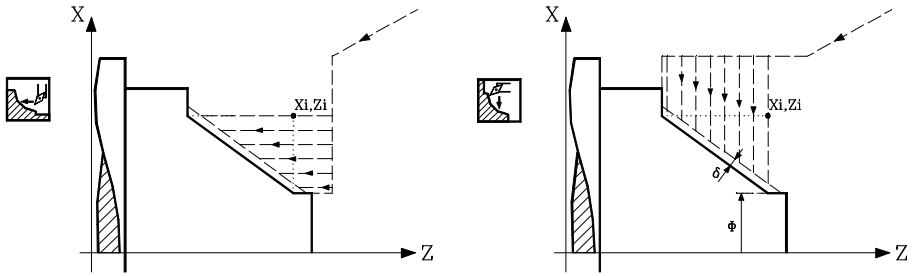
Indicates the type of corner to be machined.



•Machining direction.

: Turning.

: Facing.



When changing the machining direction, the graphic help also changes.

NOTE: When selecting an icon, its meaning appears at the bottom of the screen.

NOTE: Refer to the Operation Manual Chapter 4 Section 4.6.1

4.3.2 Simulate an operation.

It is used for checking the tool path on the screen.

– Press GRAPHICS. The CNC will display the graphics menu. To access the various options, press their corresponding keys:

Funtion:	TYPE OF GRAPHIC	DISPLAY AREA	ZOOM		GRAPHIC PARAMETERS	CLEAR SCREEN	+
Key:	F1	F2	F3	F4	F5	F6	F7

To begin simulating, press .

The simulating speed is selected with the FEED selector.

Other useful keys are:

: Interrupts the simulation. While interrupted:

: Resumes the simulation.

RESET : Stops the simulation.

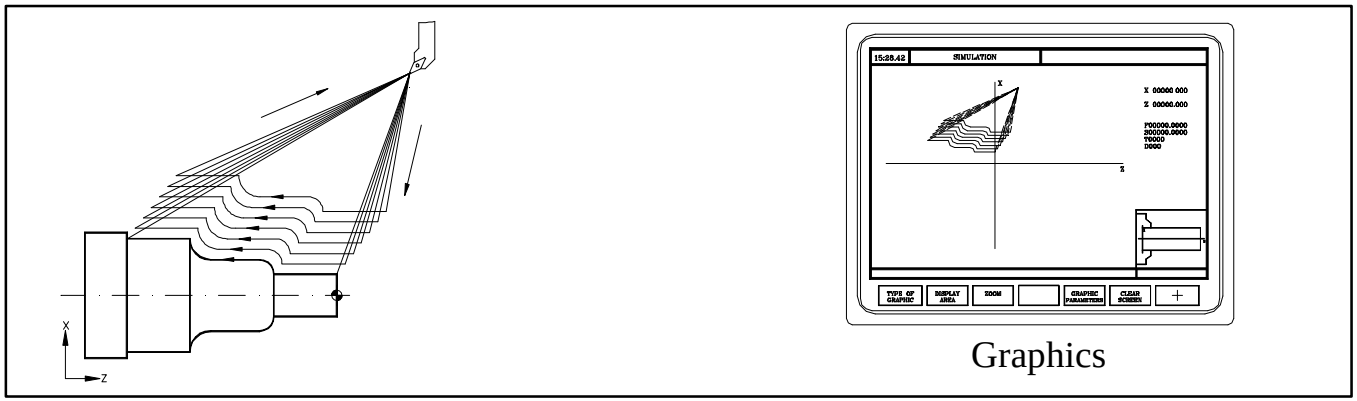
ESC or GRAPHICS : Quits the simulation mode.

NOTE: Refer to the Operation Manual Chapter 6 Section 6.5

- Type of graphics.
  - "X-Z" graphics.

    The tool path is represented by color lines.

NOTE: When carrying out this simulation, the screen shows only half of the part. This is because only the tool tip path is represented (not the part).



Graphics

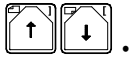
NOTE: To check the part dimensions on the simulation screen, the simulation must be carried out with a tool whose nose radius is R=0.

  - "Solid X-Z" graphics.

    Starting at the first block, the simulation shows how the tool removes material. When done, it shows the resulting part.

• Display area.

It is possible to define the display area by setting the maximum and minimum axis coordinates.

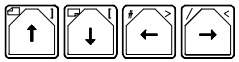
–To set the coordinates, use  .

–Once the data has been set, press  .

• ZOOM.

It is used for enlarging or reducing the drawing or part of it. The new display area is selected by means of a window superimposed on the shown tool path.

–To enlarge or reduce the drawing, use the keys for “ZOOM+” and “ZOOM-”.

–To move the window around, use: 

–For the CNC to assume the new values, press  .

–To draw the selected section, press  .

To return to the original display area, choose the INITIAL VALUE option.

• Graphic parameters.

Simulation speed: For selecting the % override of the simulation speed being applied.

Tool path colors: For changing the tool path colors on “X-Z” graphics.

Colors for solid graphics: For changing the colors of the tool and the part on “Solid X-Z” graphics.

• Clear screen.

It clears the screen and shows the axes or the part without being machined.

4.3.3 Execute an operation.

The operations can be executed from beginning to end or a pass at a time. This choice is made with 

.

To start the execution, enter into “Execution mode” and press  .

Once execution has started:

 : Interrupts the execution. While interrupted, if we press:

 : Resumes the execution.

: Cancels the execution.

: Switches to graphics mode.

The execution can be interrupted at any time, except while making a thread. In that case, the execution will be interrupted at the end of the pass.

NOTE: Refer to the Operation Manual Chapter 6 Section 6.3/6.4

Tool inspection.

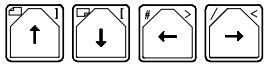
With this option, the operation may be interrupted for inspecting and replacing the tool or for modifying the tool wear value.

- Press  .
- Depending on the machine manufacturer, on some machines  will also have to be pressed to get into tool inspection.
- The top of the CNC screen displays the message: INSPECTION. Jog the tool with the jog keys or the handwheels.
- Once in "Tool Inspection", it is possible to move the axes, check or change the tool, stop or start the spindle and change the tool wear value.
- Press  to reposition the axes and resume execution.

NOTE: Refer to the Operation Manual Chapter 6 Section 6.4.1

Modifying the tool wear value.

With this option, the I, K values may be changed. The entered values are incremental and will be added to those stored previously.

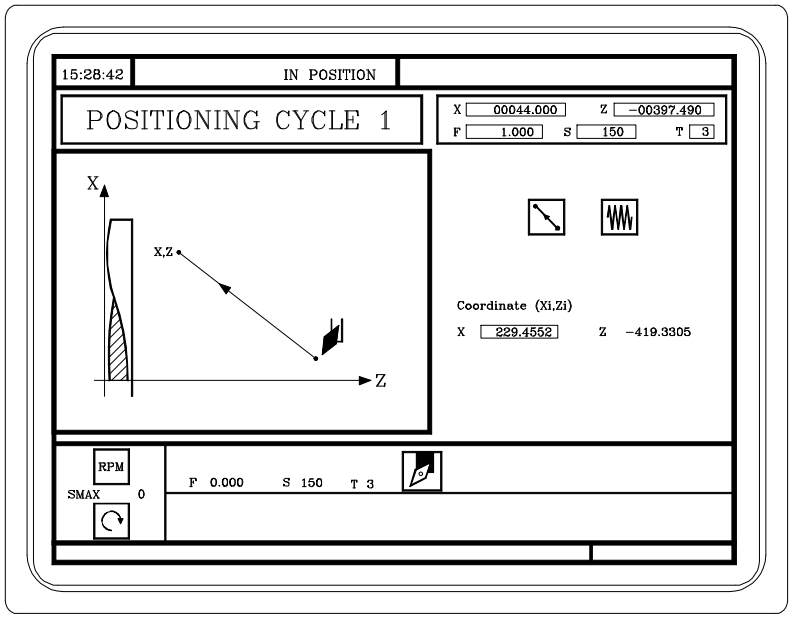
- Press  . The CNC shows the table for that tool.
- Use the  keys to position the cursor over the I value.
- Key in the I value and press  .
- Position the cursor over the K value.
- Key in the K value and press  .
- To change the offset of another tool, press:

   + (Tool Number) + 
- To finish, press  .

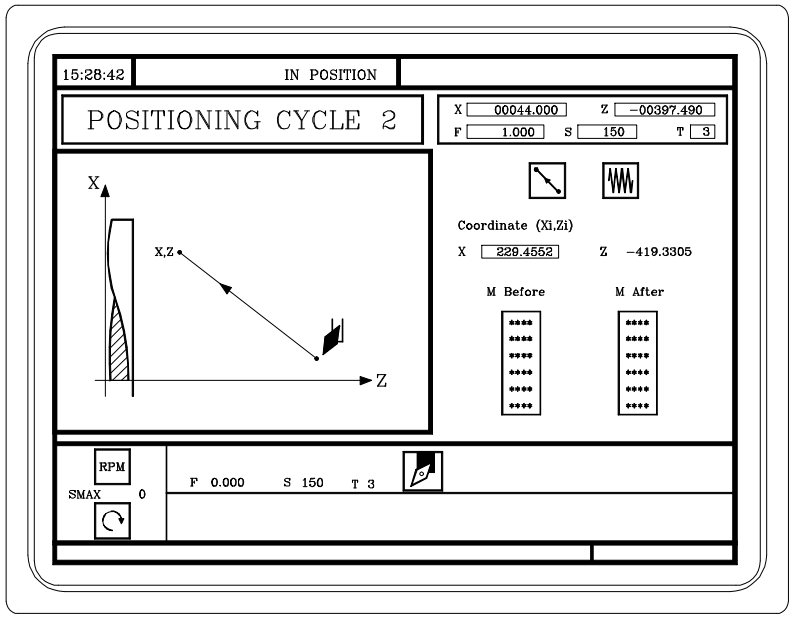
Chapter 5

Summary of work cycles

5.1 Positioning cycles.



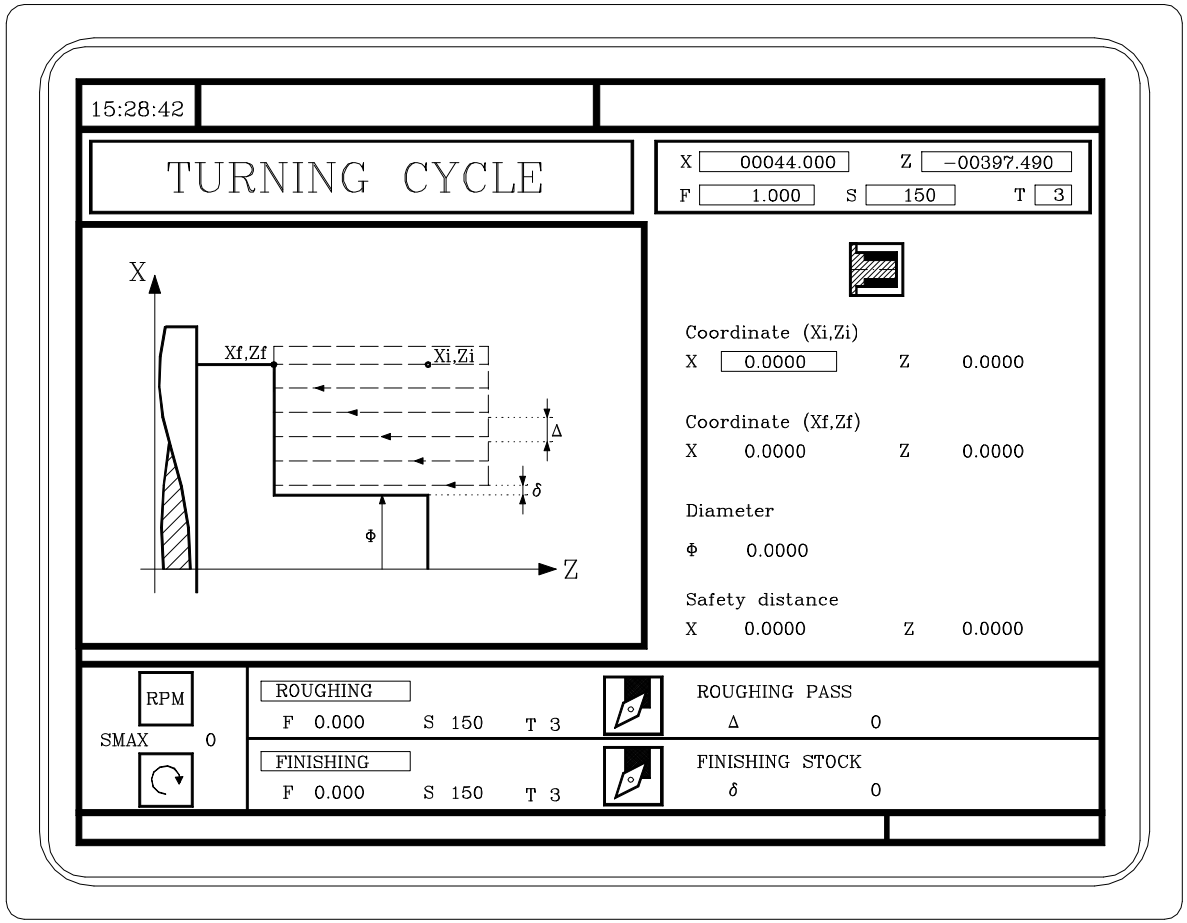
LEVEL CYCLE



In this cycle level, it is possible to define the auxiliary functions to be executed before and after the movement.

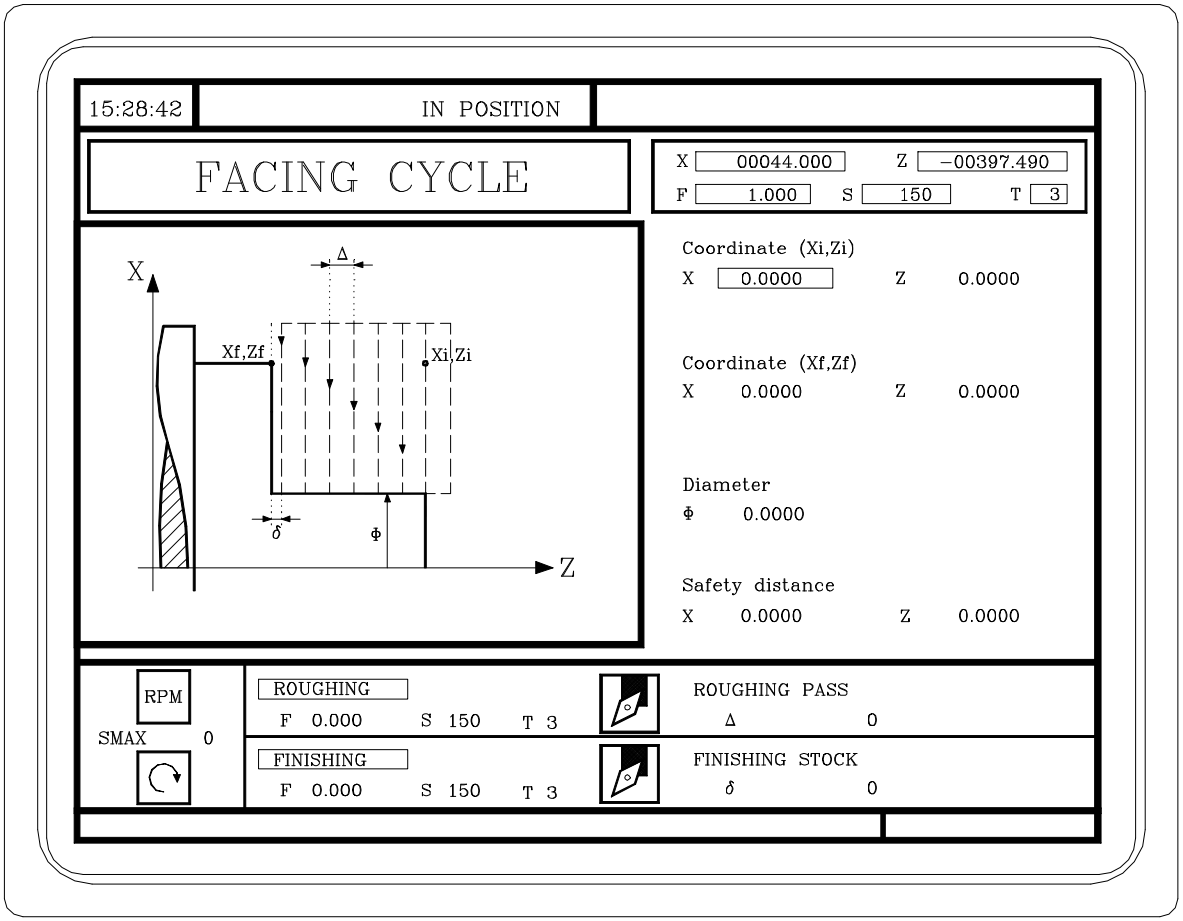
NOTE: Refer to the Operation Manual Chapter 4 Section 4.3

5.2 Turning cycle



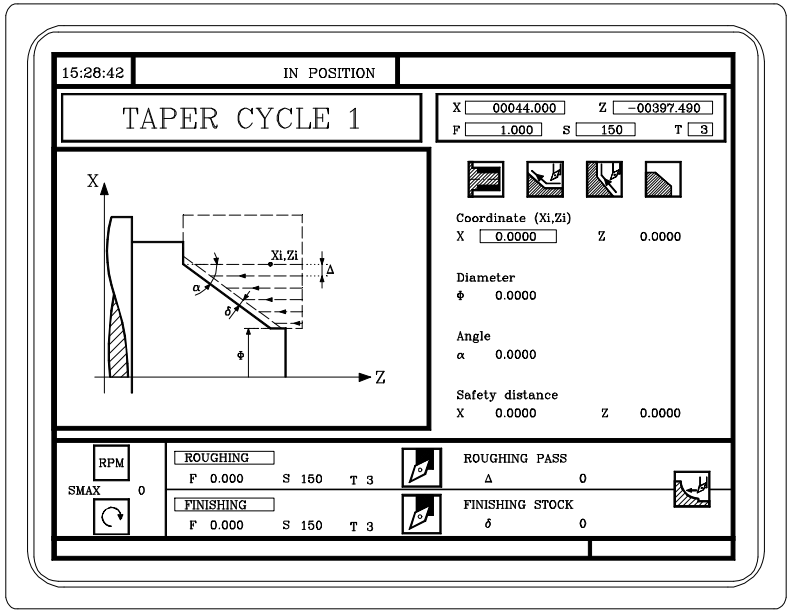
NOTE: Refer to the Operation Manual Chapter 4 Section 4.4

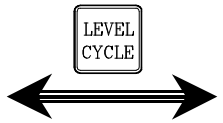
5.3 Facing cycle.

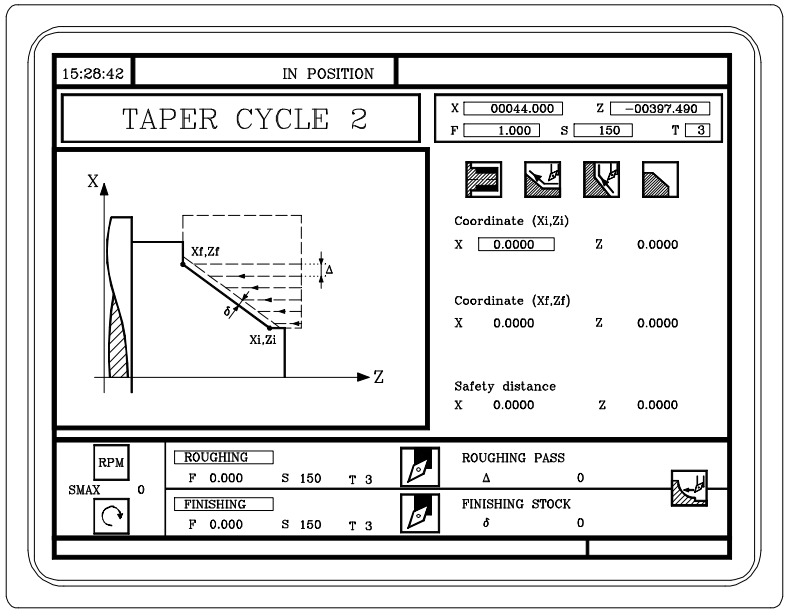


NOTE: Refer to the Operation Manual Chapter 4 Section 4.5

5.4 Taper cycles.





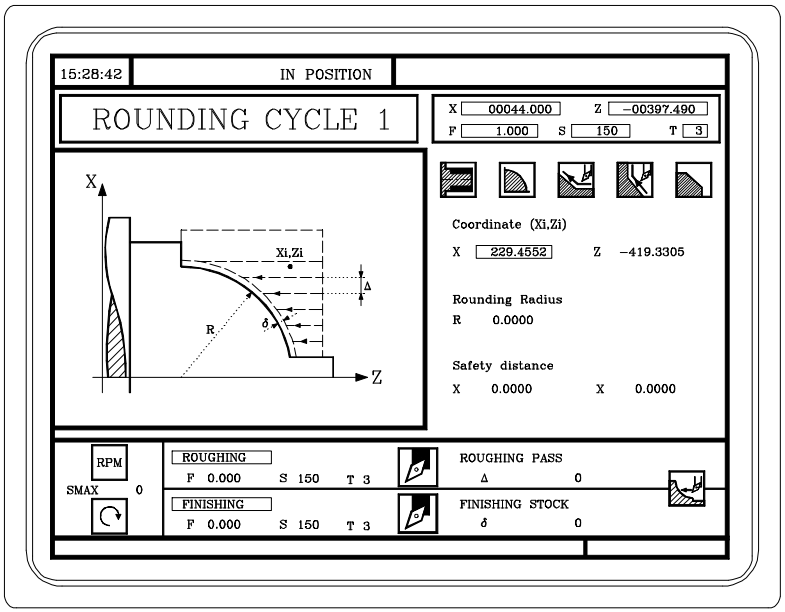


In this cycle level, one defines the coordinates of the theoretical corner, the taper angle and the final diameter.

In this cycle level, one defines the coordinates of the starting point and end point.

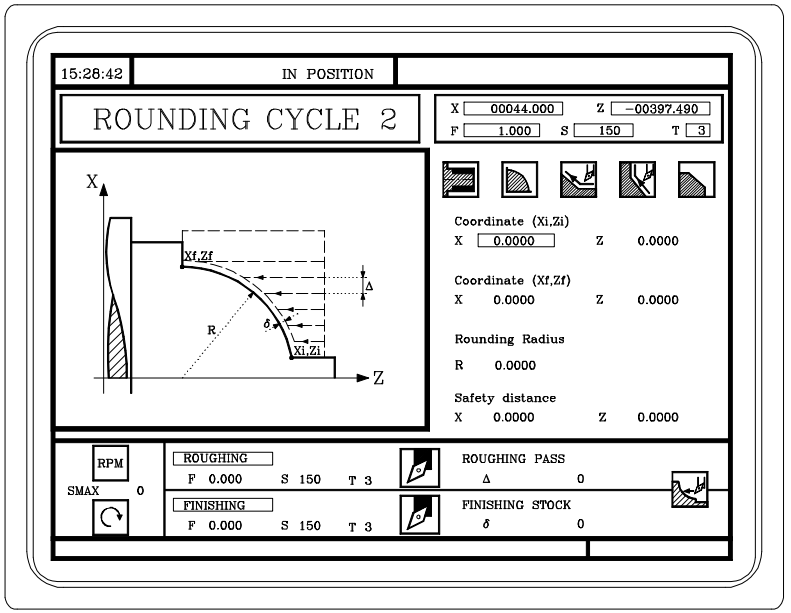
NOTE: Refer to the Operation Manual Chapter 4 Section 4.6

5.5 Rounding cycles.



In this cycle level, one defines the coordinates of the theoretical corner and the rounding radius.

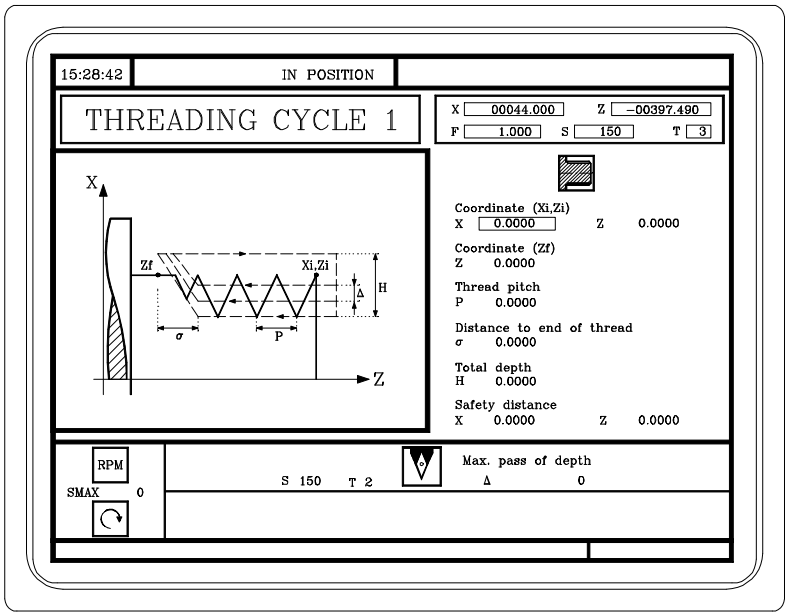
LEVEL CYCLE



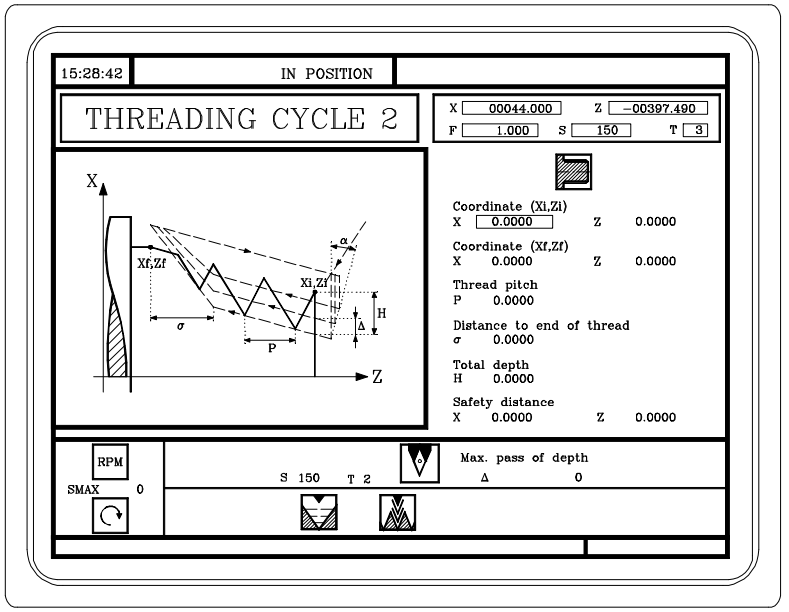
In this cycle level, one defines the coordinates of the starting point and end point as well as the rounding radius.

NOTE: Refer to the Operation Manual Chapter 4 Section 4.7

5.6 Threading cycles.

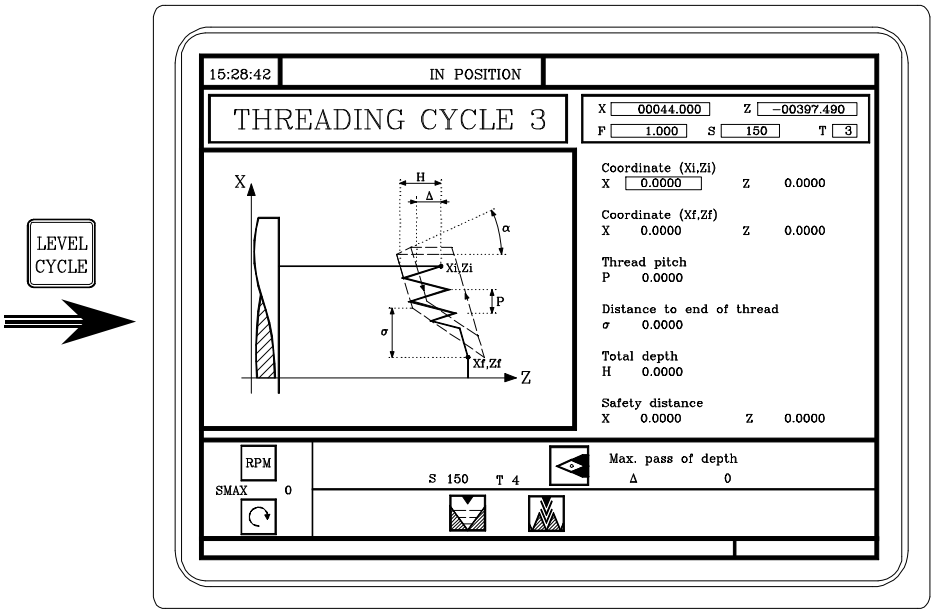


Longitudinal threading.

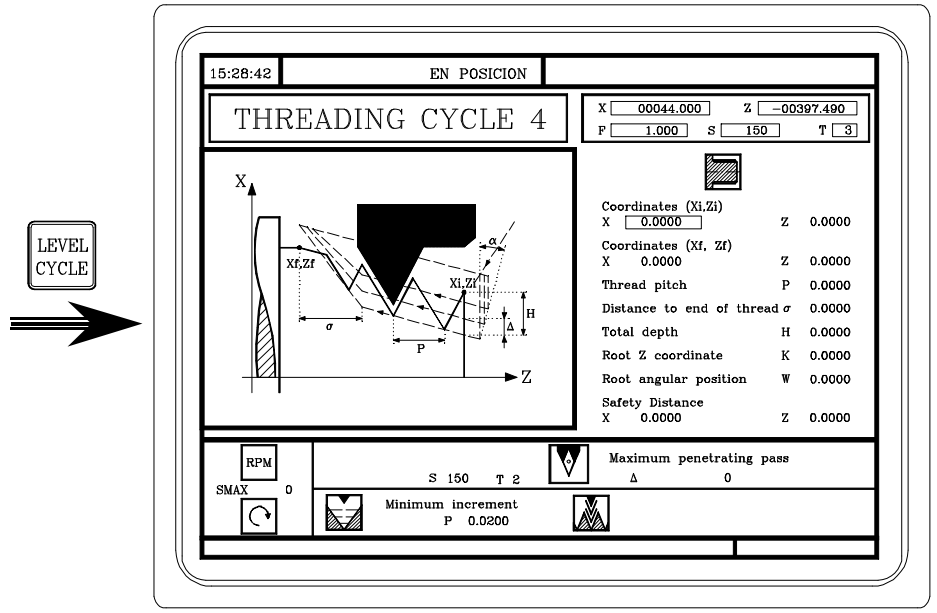


Face threading

NOTE: Refer to the Operation Manual Chapter 4 Section 4.8



Face threading

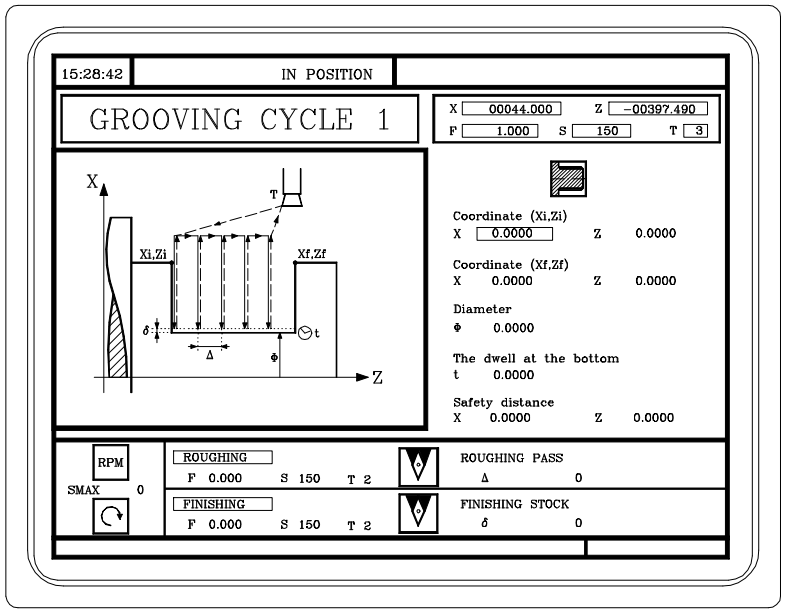


Thread repair

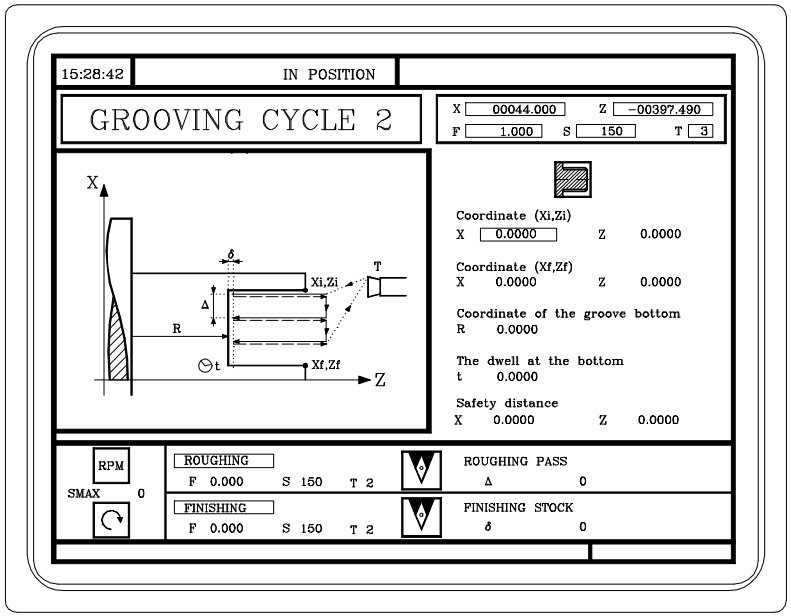
Only when having spindle orientation.

NOTE: Refer to the Operation Manual Chapter 4 Section 4.8

5.7 Grooving cycles.

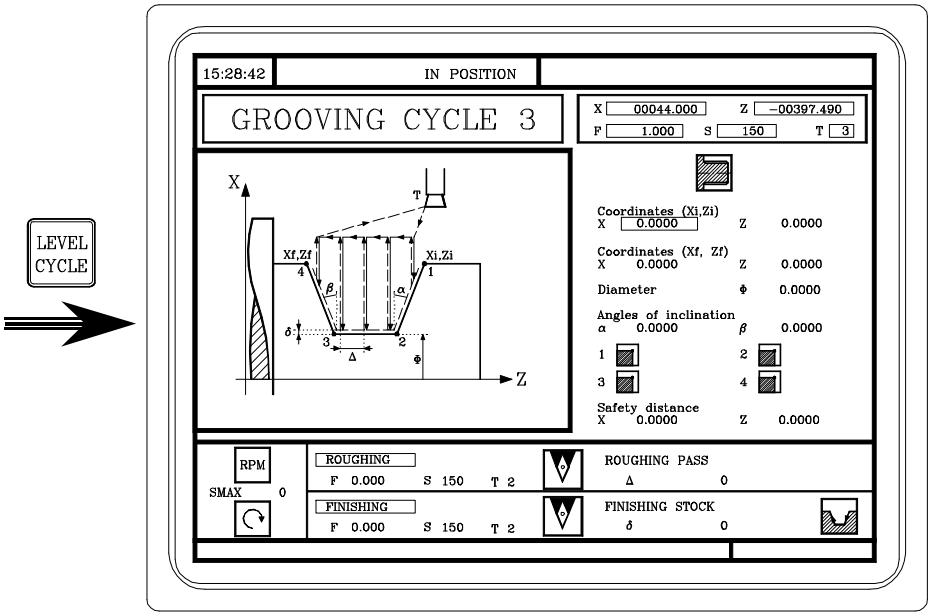


Radial grooving.

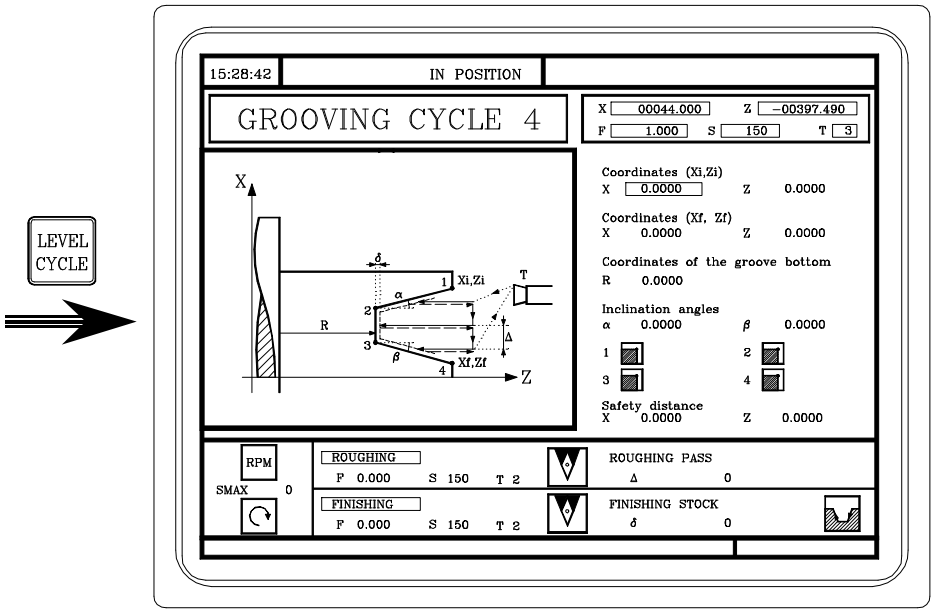


Face (axial) grooving

NOTE: Refer to the Operation Manual Chapter 4 Section 4.9



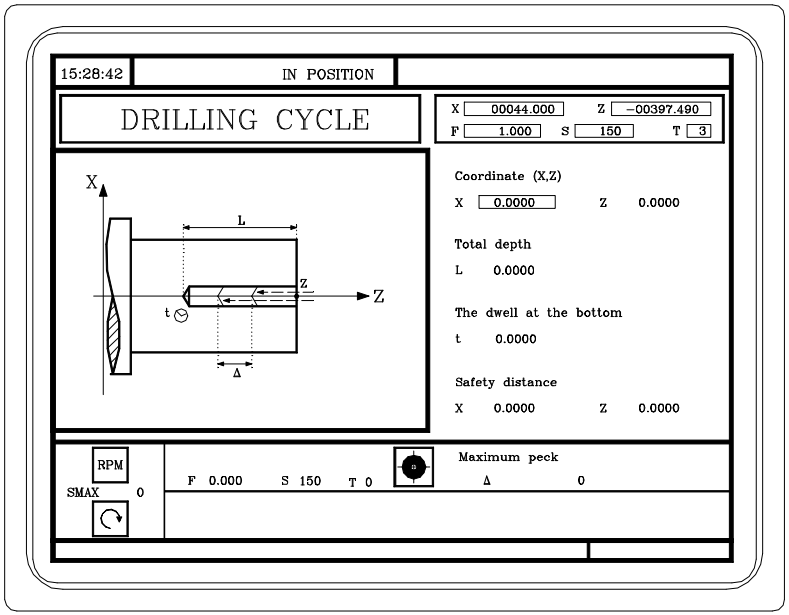
Radial grooving with incline walls.

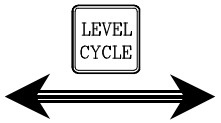


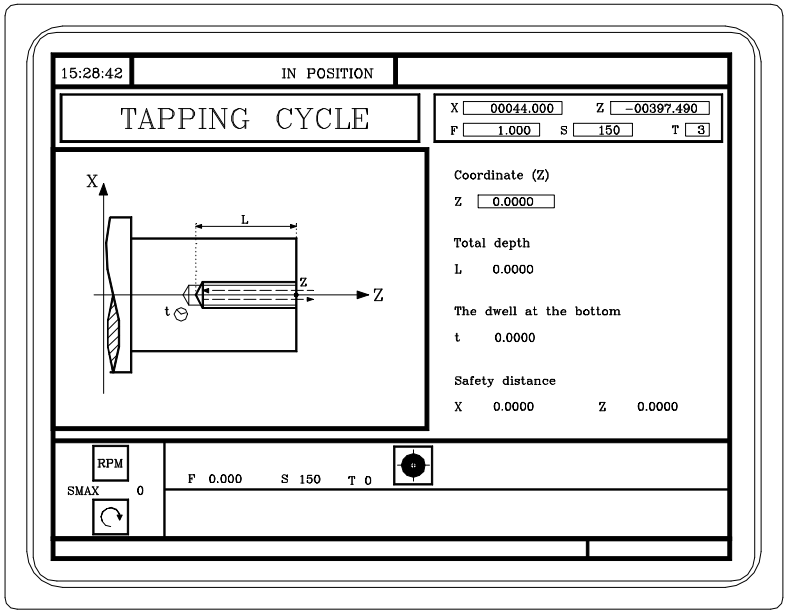
Face grooving with incline walls.

NOTE: Refer to the Operation Manual Chapter 4 Section 4.9

5.8 Drilling and Tapping cycles.







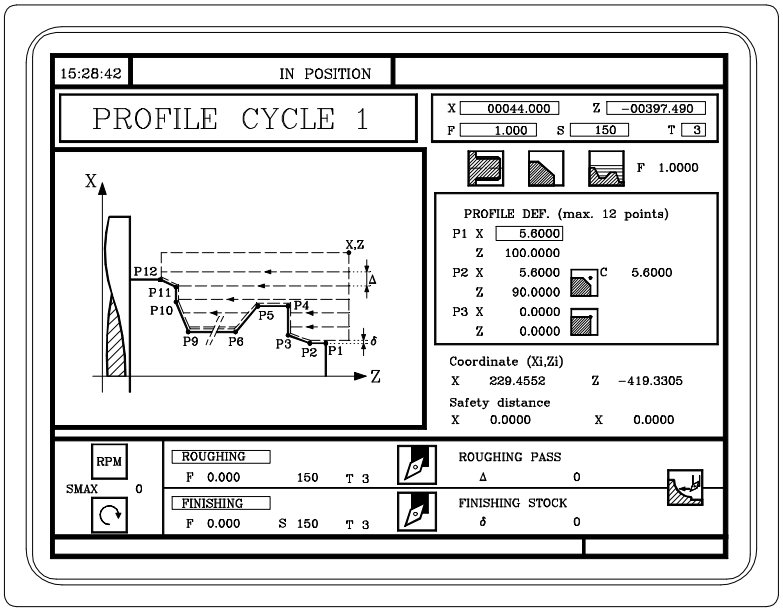
Drilling cycle.

Tapping cycle.

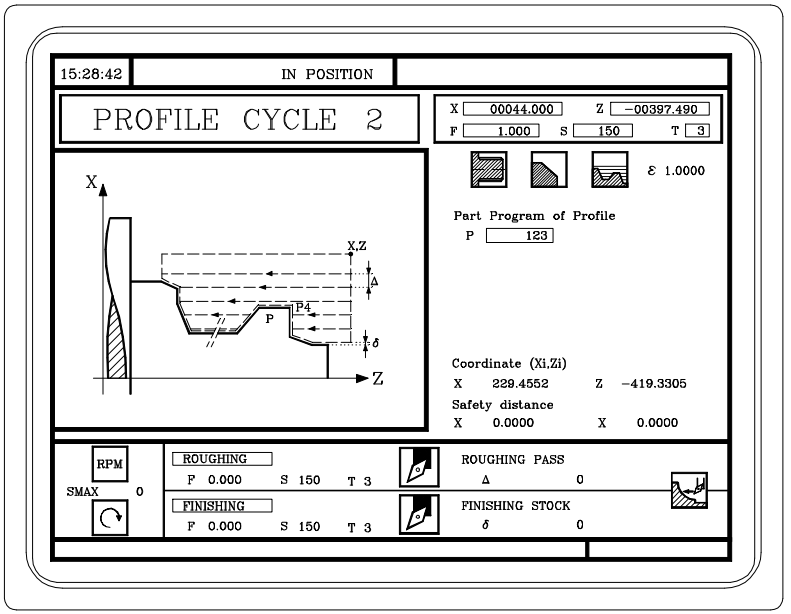
NOTE: When having spindle orientation, the cycles described in Appendix I will be displayed.

NOTE: Refer to the Operation Manual Chapter 4 Section 4.10

5.9 Profile cycles.



LEVEL CYCLE



In this cycle level, one defines all the points of the profile.

This cycle level uses a part-program containing all the profile data.

NOTE: Refer to the Operation Manual Chapter 4 Section 4.11

Chapter 6

Conversational part-programs

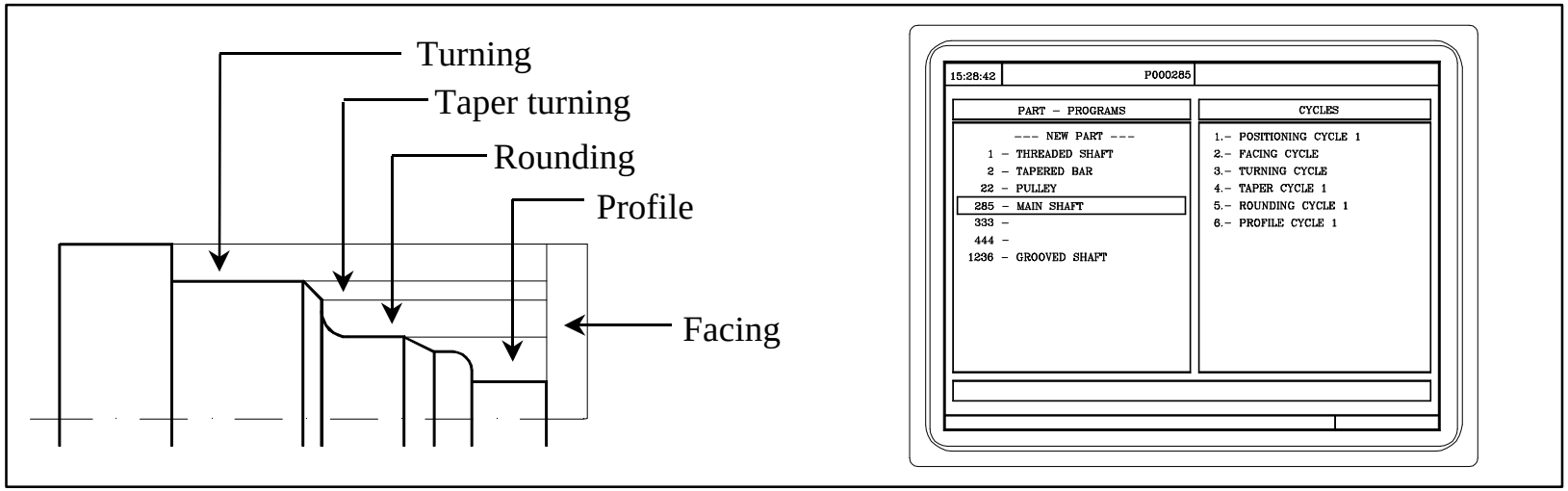
This chapter describes:

- What a conversational part-program is.
- How to edit it.
- How to change it. (Inserting or deleting operations).
- Simulate/execute an operation.
- Simulate/execute starting at a particular operation.
- Simulate/execute a part-program.
- Copy a part-program.
- Delete a part-program.

6.1 What is a conversational part-program?

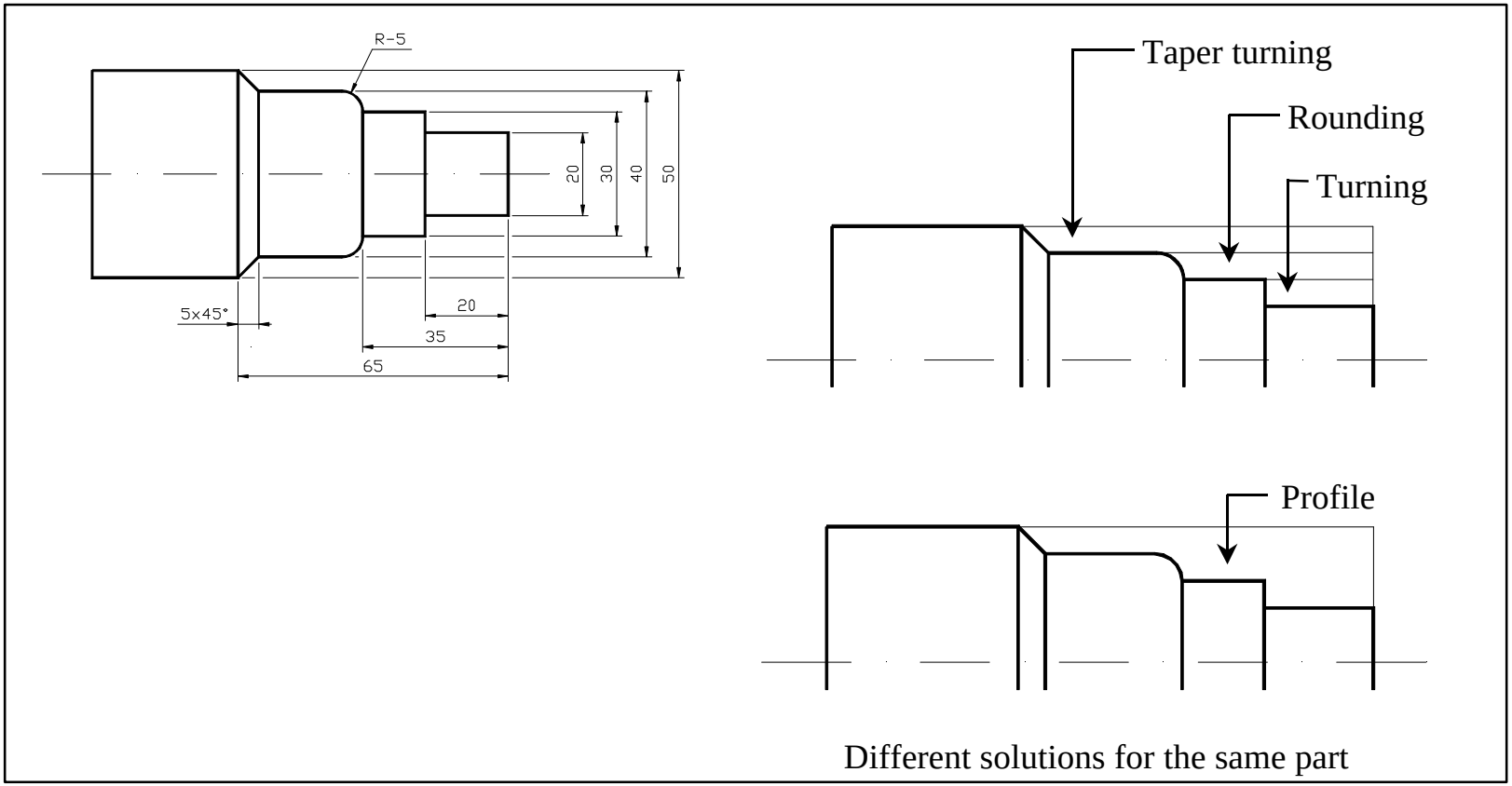
It is a set of operations ordered secuentially. Each operation is defined separately and they are then stored one after the other in a program

The name of the part-program can be any integer between 1 - 899999.



6.2 Edit a part-program.

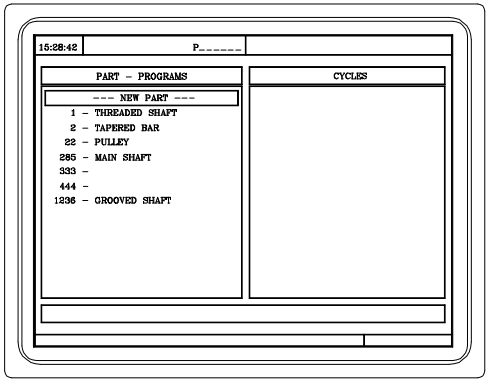
To edit a part-program, we first choose the operations needed to execute the part. A part may be executed in various ways.



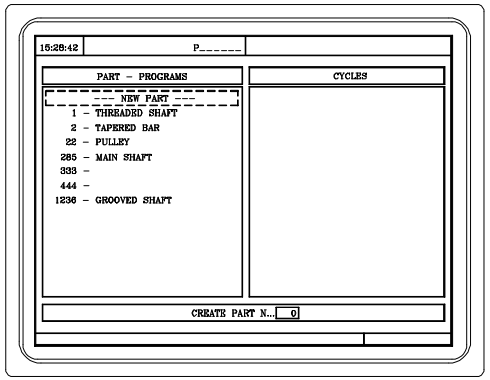
Different solutions for the same part

Once the sequence of operations has been chosen (in this case, we will make the previous example), the part-program is built by editing the operations one by one.

DRO MODE → P.PROG

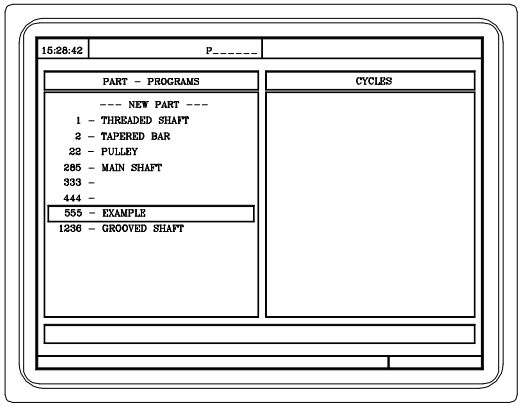


→ P.PROG



(Key in the number) + ENTER + (Comment) + ENTER →

v.g : <555> + ENTER + <EXAMPLE> + ENTER

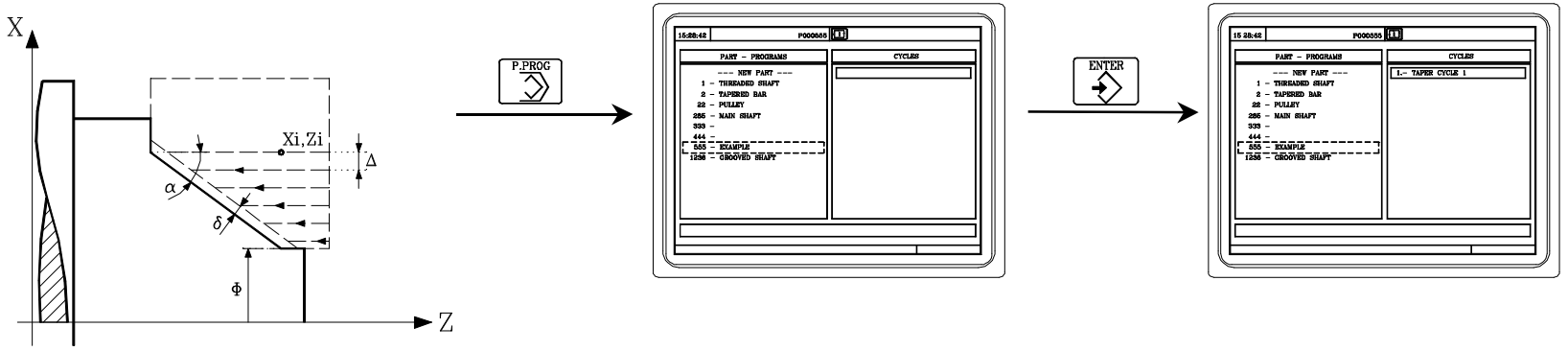


NOTE: The following keys are used:

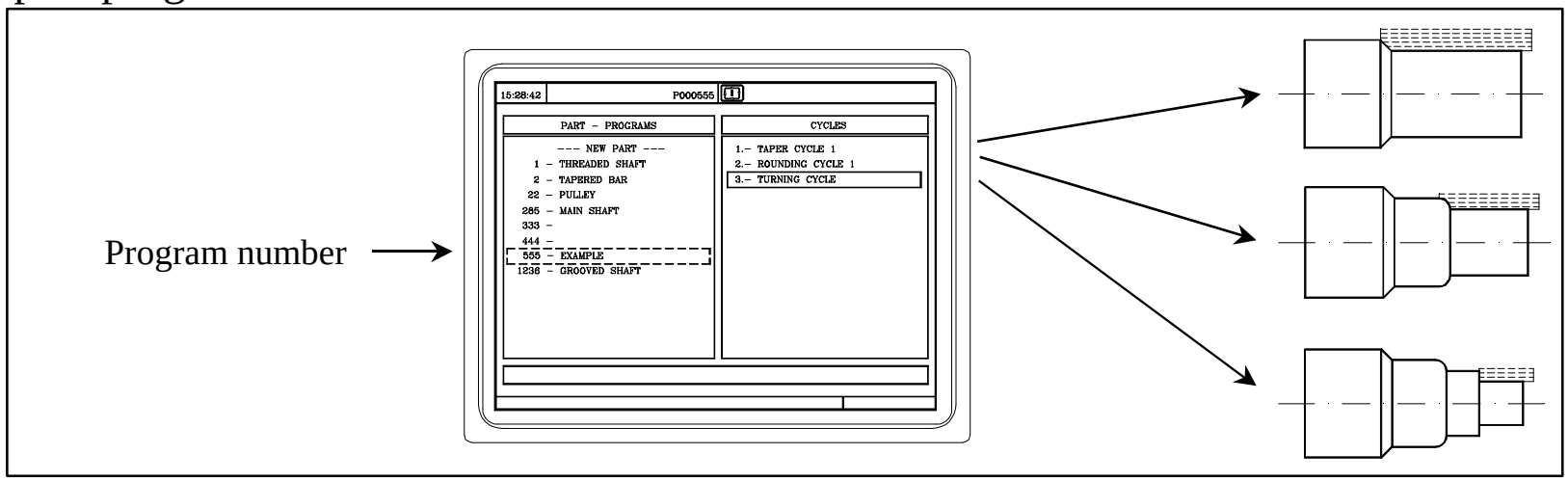
↑ ↓ : To move up and down on each column.

← → : To change columns.

Choose the operation and define the parameters



Repeat these steps with the other operations. In our case, the finished part-program will be:

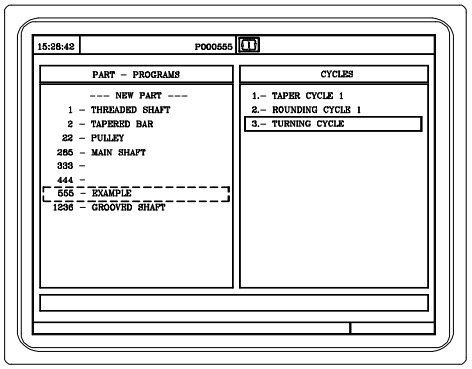


6.3 Modify a part-program.

The operations making up a part-program can be modified.

REPLACE AN OPERATION

Choose oeeration →

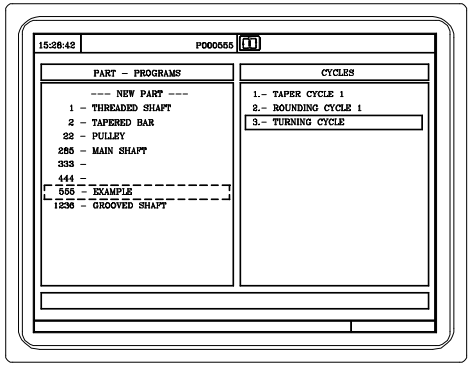


→ RECALL →

The CNC shows the cycle with all its data.

Modify the operation parameters like in the editing mode.

→ P.PROG →



→ ENTER →

The CNC requests an option. Choose REPLACE.

→ ENTER →

The new operation replaces the previous one.

NOTE: Refer to the Operation Manual Chapter 5 Section 5.6.4

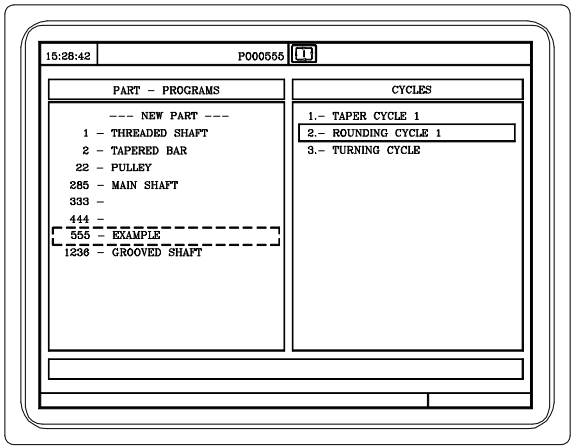
New operations can also be inserted into a part-program.

INSERT AN OPERATION

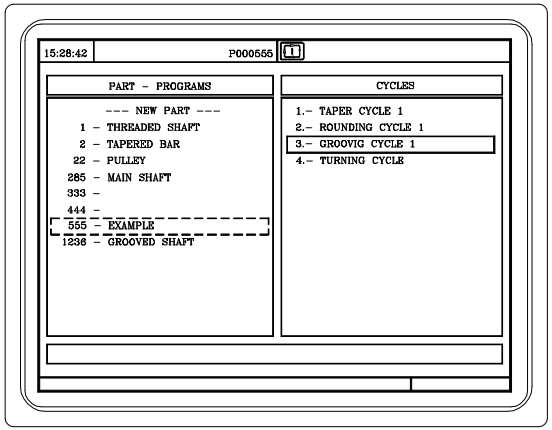
Choose operation →

Define the parameters and cutting conditions of the operation to be inserted. Press P.PROG

Choose position →



ENTER →



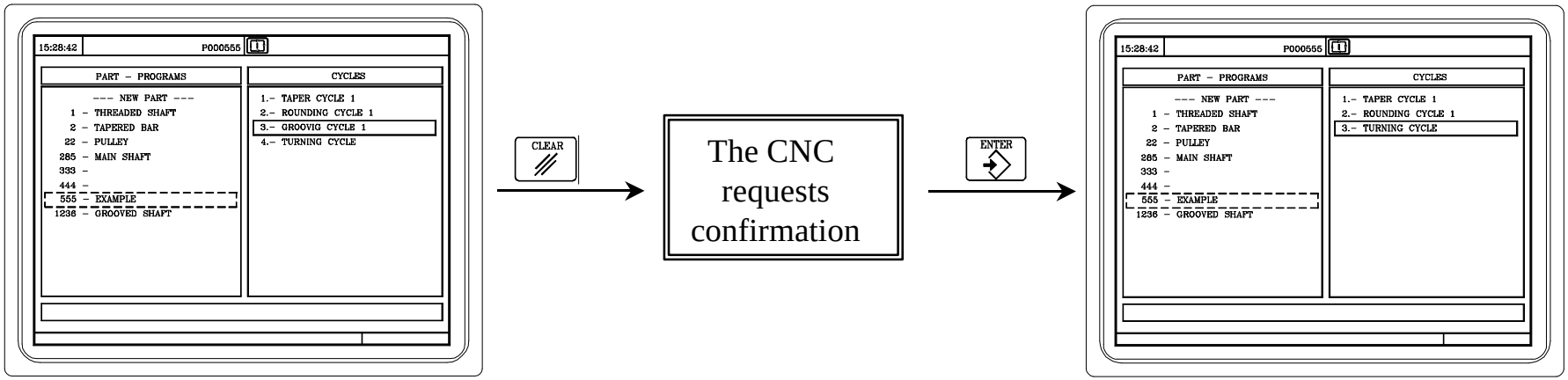
The new operation is inserted after the chosen position.

NOTE: Refer to the Operation Manual Chapter 5 Section 5.6.3

Operations can be deleted from a part-program.

DELETE AN OPERATION

Select, on the right column, the operation to be deleted.

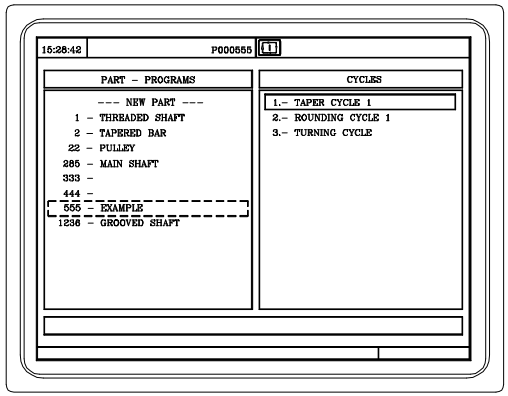


NOTE: Refer to the Operation Manual Chapter 5 Section 5.6.1

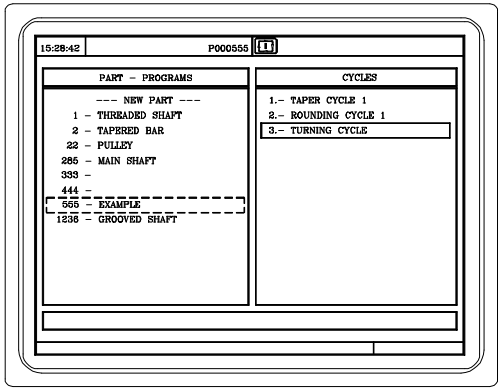
The position of an operation can also be changed.

CHANGE THE POSITION OF AN OPERATION

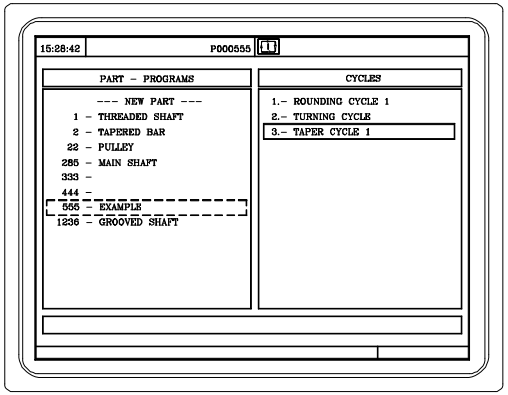
Select, on the right column, the operation to be moved.



Select the new position



ENTER

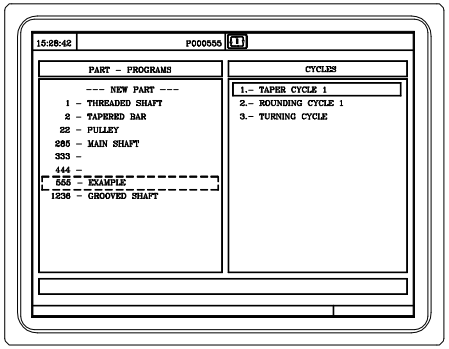


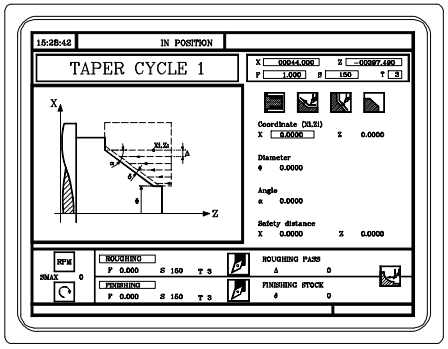
The operation is inserted behind the operation occupying that position.

NOTE: Refer to the Operation Manual Chapter 5 Section 5.6.2

6.4 Simulate/execute an operation.

Select, on the right column, the operation to be SIMULATED:

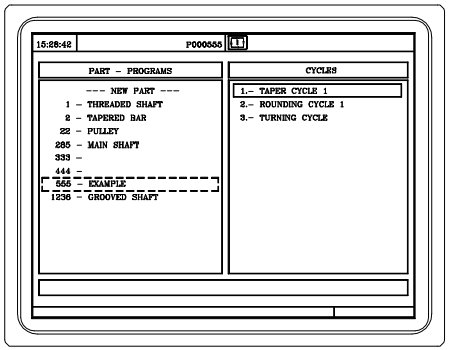


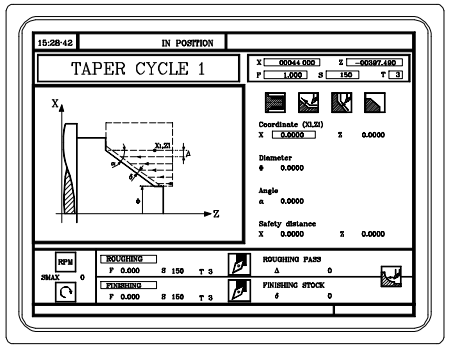


Graphics screen

More information about the graphics screen in chapter 4.3.2 of this manual.

Select, on the right column, the operation to be EXECUTED:

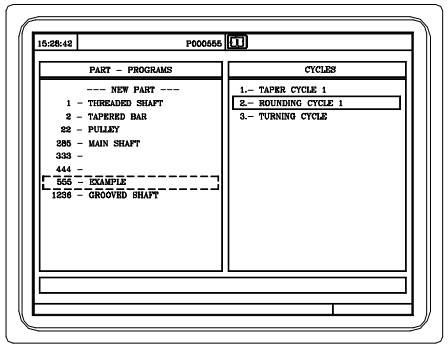




NOTE: Refer to the Operation Manual Chapter 6 Section 6.3

6.5 Simulate/execute starting at a particular operation.

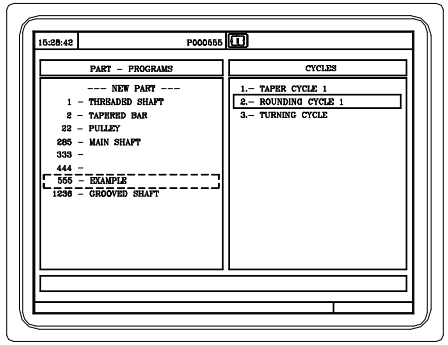
Select, on the right column, the operation where the SIMULATION is to be started:



Graphics screen

More information about the graphics screen in chapter 4.3.2 of this manual.

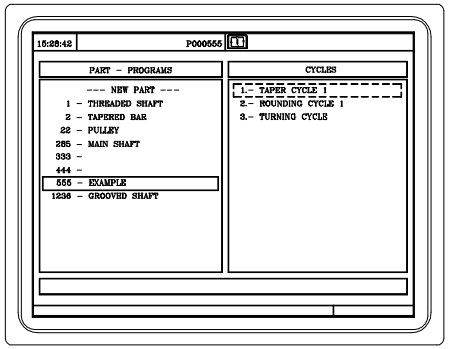
Select, on the right column, the operation where the EXECUTION is to be started:



NOTE: Refer to the Operation Manual Chapter 6 Section 6.2.1

6.6 Simulate/execute a part-program.

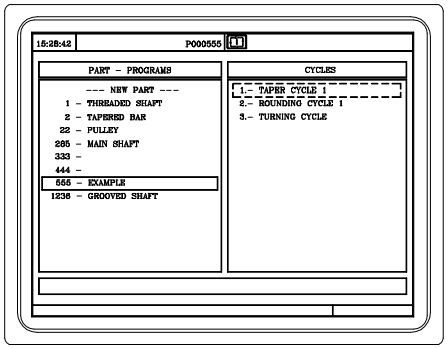
Select, on the left column, the part-program to be SIMULATED:



Graphics screen

More information about the graphics screen in chapter 4.3.2 of this manual.

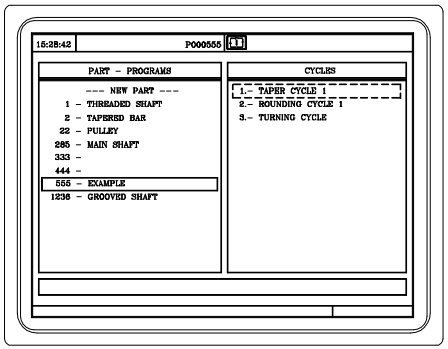
Select, on the left column, the part-program to be EXECUTED:

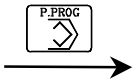


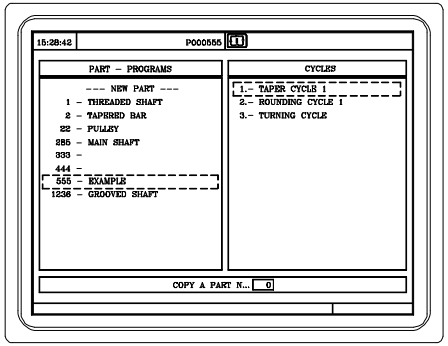
NOTE: Refer to the Operation Manual Chapter 6 Section 6.2

6.7 Copy a part-program into another one.

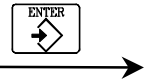
Select, on the left column, the part-program to be COPIED:







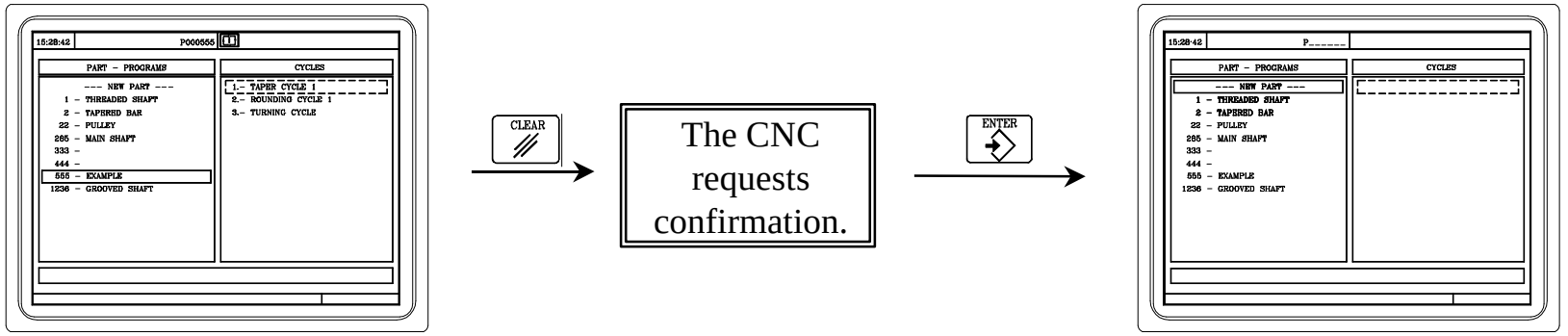
Key in the number and comment of the new program.



NOTE: Refer to the Operation Manual Chapter 5 Section 5.5

6.8 Delete a part-program.

Select, on the left column, the part-program to be deleted:



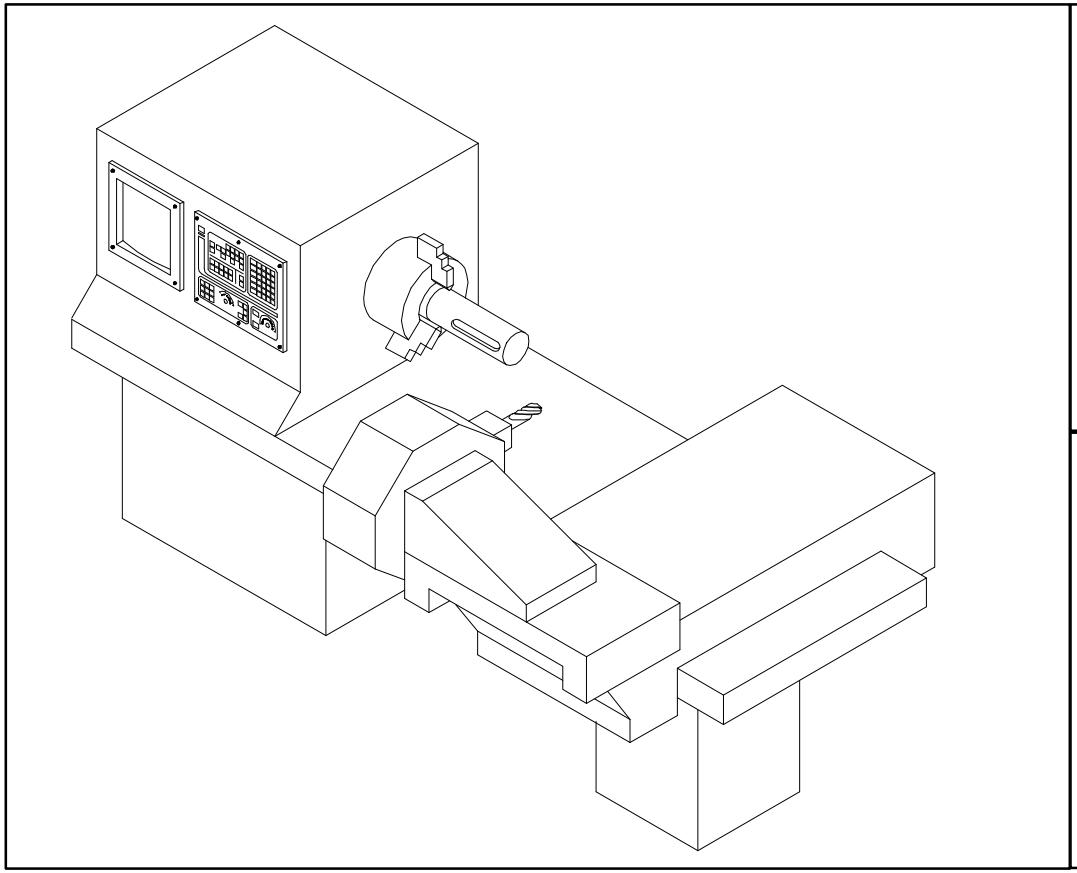
NOTE: Refer to the Operation Manual Chapter 5 Section 5.4

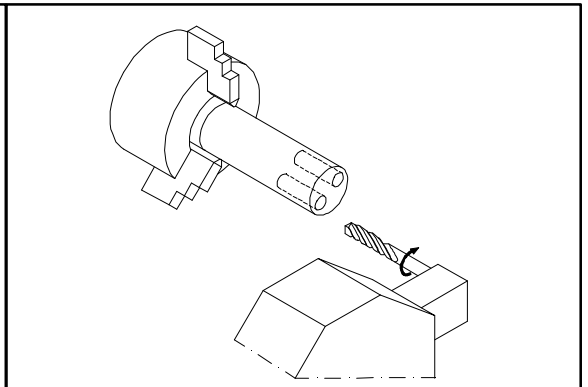
Appendix I

Other machining operations on a lathe

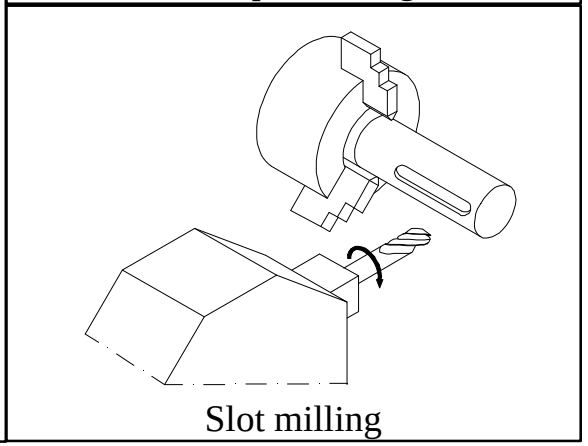
I.1 Introduction.

For this type of machining operations, the machine must have a spindle which can be oriented and a live tool. If the machine has these features, the CNC menu will offer the "Multiple drilling" and "Slot milling" choices when accessing the "Drilling cycle".





Multiple drilling

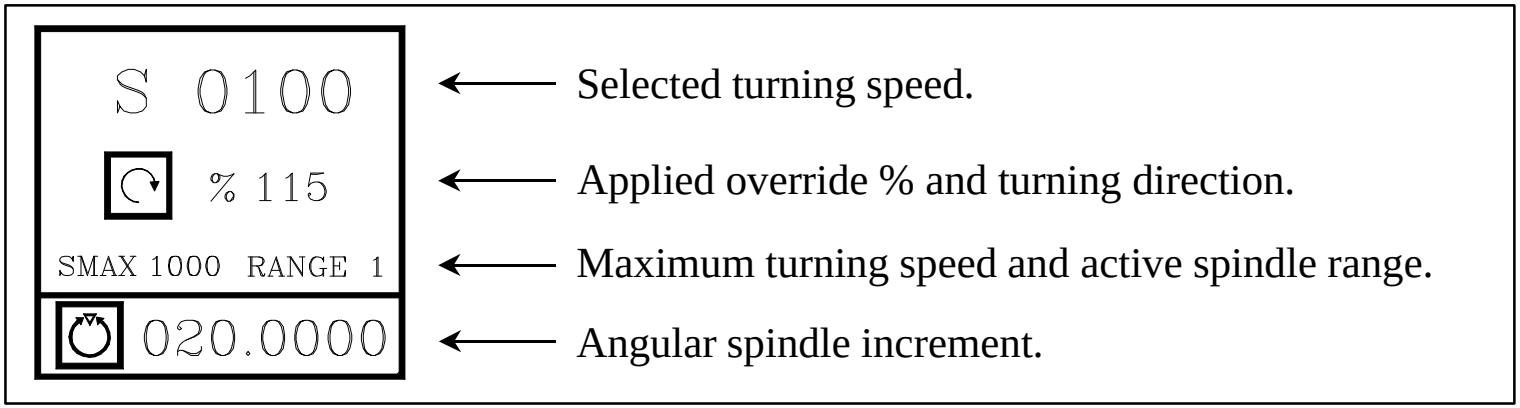


Slot milling

I.2 Spindle orientation.

With this feature, the spindle can be oriented to the desired angular position for drilling holes and milling slots both on the face of the part or on its turning surface.

The CNC shows the following information:



Press  three times to enter the value of the angular increment.

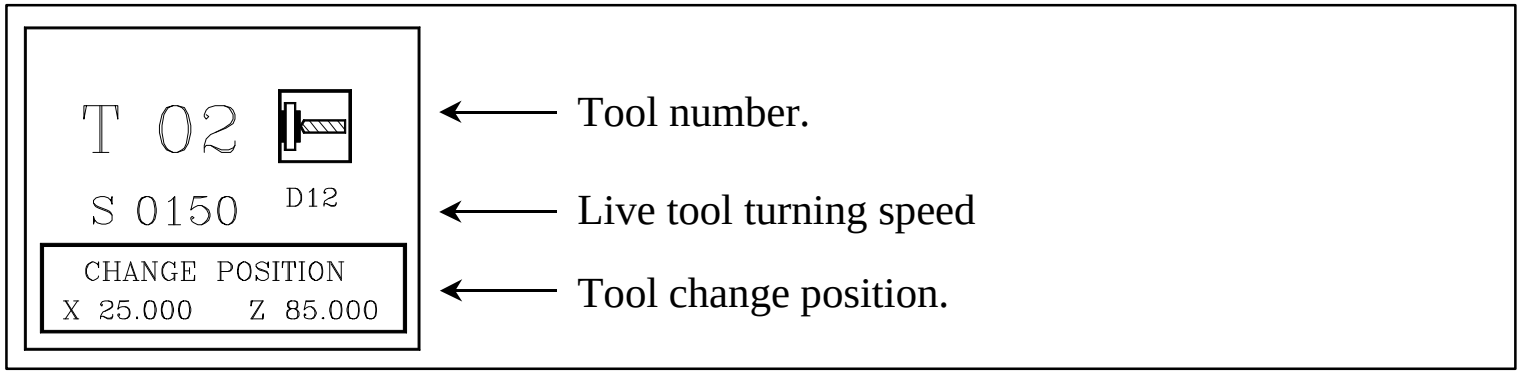
Press  to work in spindle orientation mode. The spindle stops (if it was turning) and it positions at the specified angle. Every time  is pressed, the spindle orientation increments by that angle.

To work in RPM mode, press one of these keys: 

NOTA: Consultar Manual de Operación Cap. 3 Aptd. 3.6.3

I.3 Live tool.

When selecting this type of tool, the CNC shows the following information:



To enter the live tool turning speed, press [T] to select the tool window. Then press:

[S] + (turning speed) + [ENTER]

To start the live tool, use:

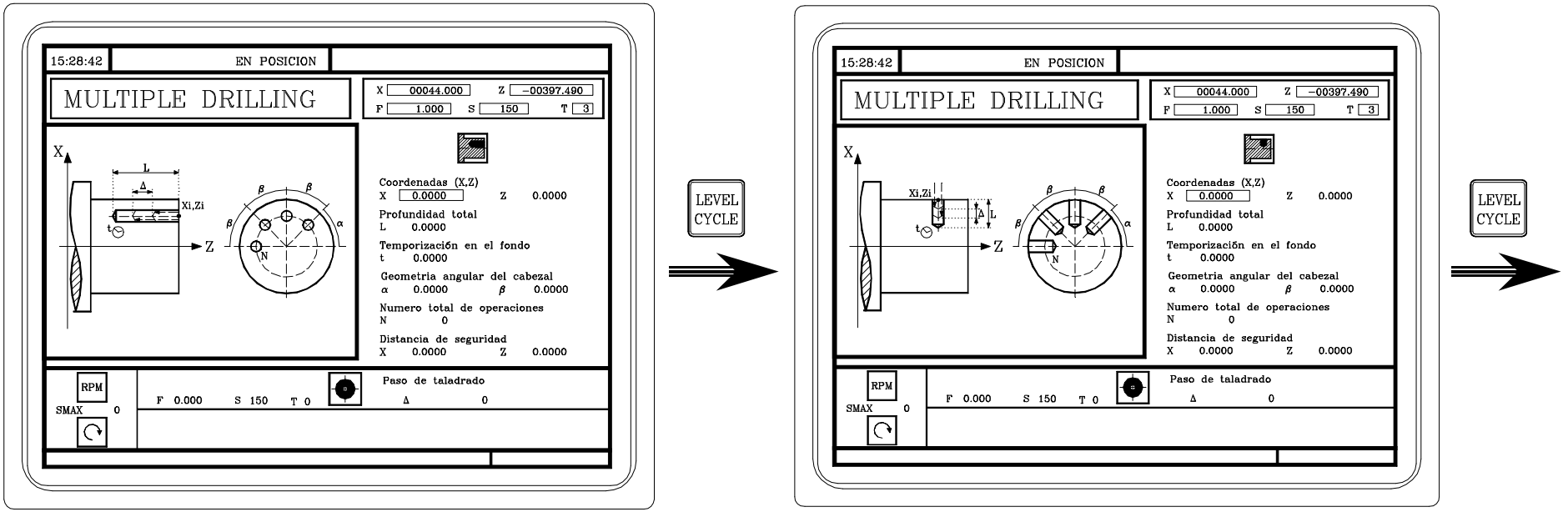
Live tool clockwise.

Live tool counter-clockwise.

Stop the live tool.

NOTE: Refer to the Operation Manual Chapter 3 Section 3.5.3

I.4 Multiple drilling. + LEVEL CYCLE

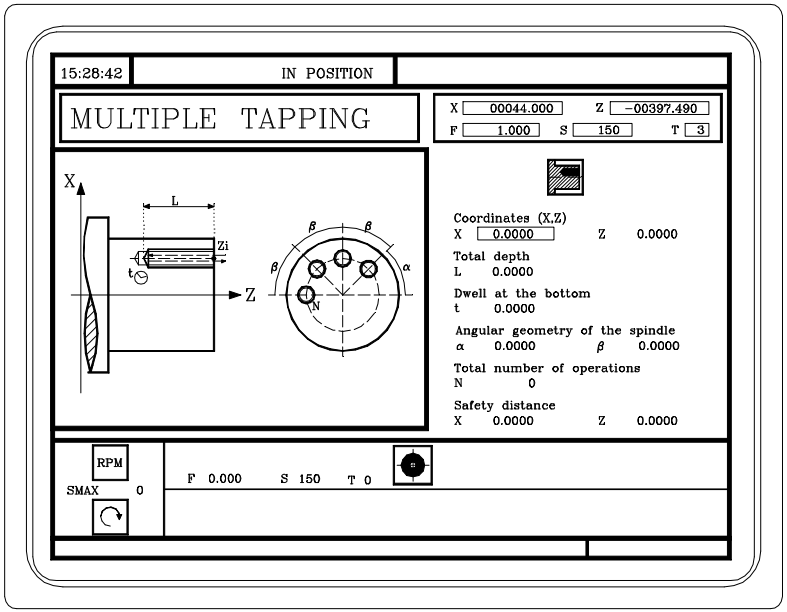


Multiple drilling on the turning surface.

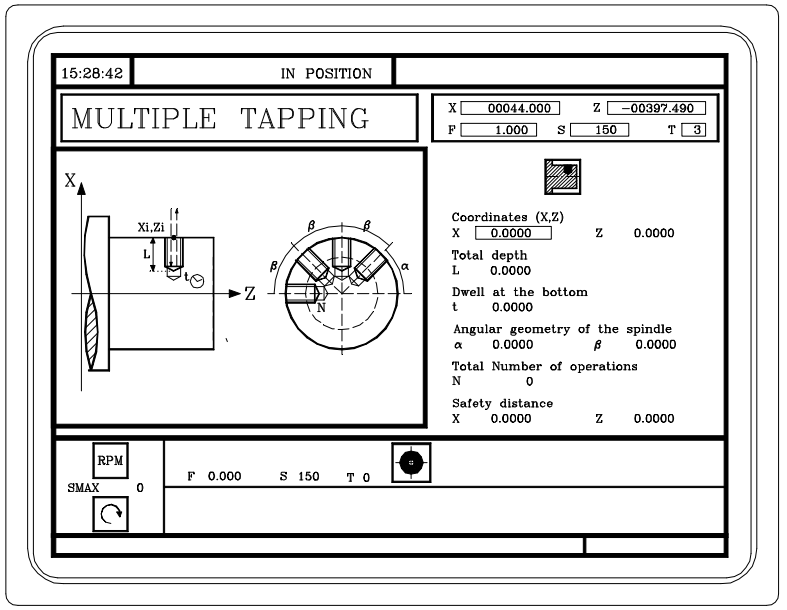
Multiple drilling on the face.

NOTE: Refer to the Operation Manual Chapter 4 Section 4.10/4.10.4

I.5 Multiple tapping.



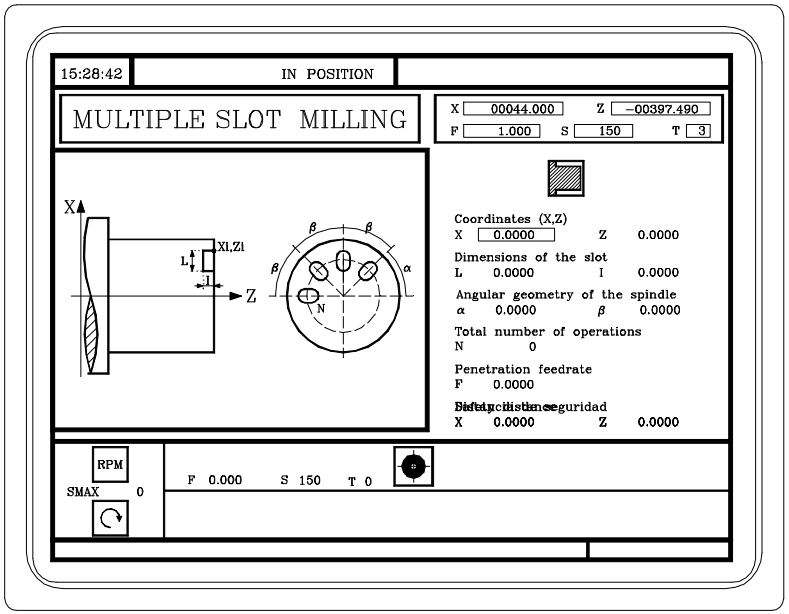
Multiple tapping on the turning surface.



Multiple trapping on the face.

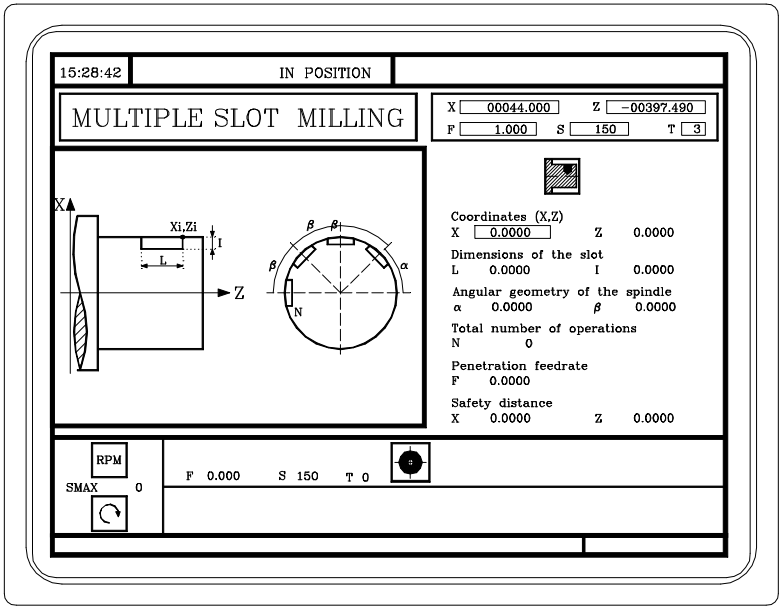
NOTE: Refer to the Operation Manual Chapter 4 Section 4.10/4.10.5

I.6 Slot milling.



Slot milling on the turning surface.

LEVEL CYCLE



Slot milling on the face.

NOTE: Refer to the Operation Manual Chapter 4 Section 4.10/4.10.6