CNC 8040

# Operating Manual
# (MC option)

Ref. 0204-ing

The information described in this manual may be subject to variations due to technical modifications.

FAGOR AUTOMATION, S.Coop. Ltda. reserves the right to modify the contents of the manual without prior notice.

# INDEX

## 1. GENERAL CONCEPTS



## 2. OPERATING IN JOG MODE



## 3. WORKING WITH OPERATIONS OR CYCLES

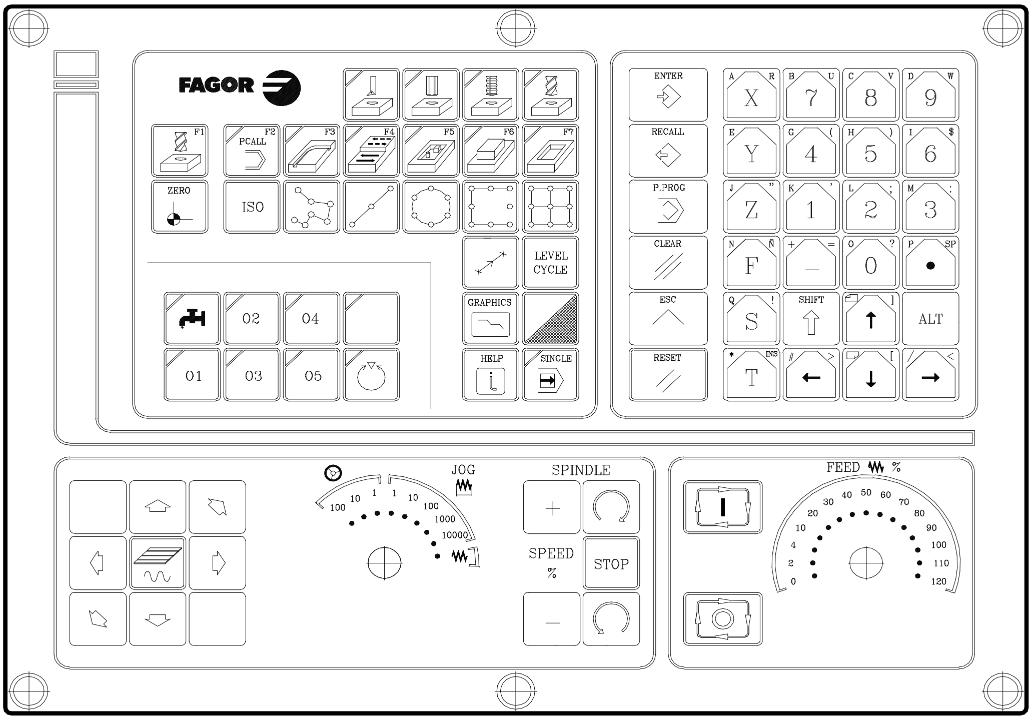
**4. STORAGE OF PROGRAMS**

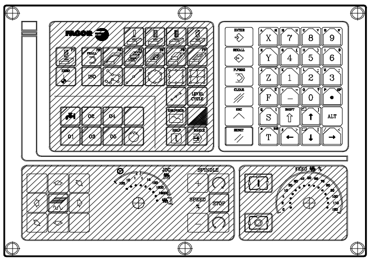


**5. EXECUTION AND SIMULATION**

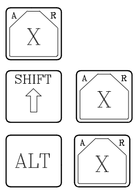
# *1. GENERAL CONCEPTS*

## *1.1 KEYBOARD*



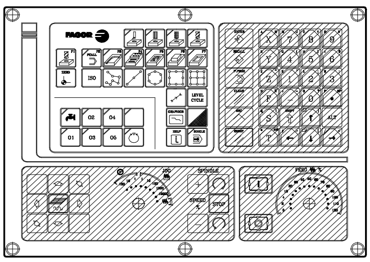
Alphanumeric keys and command keys.




Selects character X

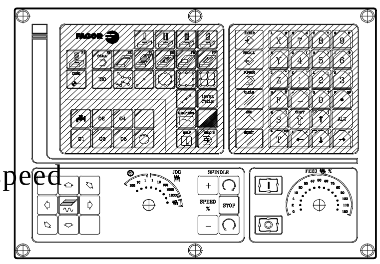
Selects character A

Selects character R



Specific keys for the MC 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 speed

Starting and stopping execution

## *1.2 GENERAL*

It has all the performance features of the M model plus the specific features of the MC mode.

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

In the MC 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 MC 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 configuration card (CARD A).

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

**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 contained in user RAM memory have been erased 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 MC format and executing these afterwards.

> **Warning**
>
> No modifications of this program are allowed. If this program is modified or erased, 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 MC 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 MC 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: DRILLING 1")
Translated: N9500(MSG"SAVEMSG: 1. ZULAKETA ZIKLOA")

Notes regarding errors:
The format must be respected. Only the text between quotes( "xxxx") may be translated
Example:
Original: N9000(ERROR"DRILLING CYCLE 1: F=0")
Translated: N9000(ERROR"1. ZULAKETA ZIKLOA: f=0")

> **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** Are the profiles defined by the user by means of the profile editor and corresponding to the pocket cycle with profiles. In the MC mode, the user defines them with three digits (0 through 999) and the CNC stores them internally as P 998xxx.

**P997000 ... P997999** Are the profiles defined by the user by means of the profile editor and corresponding to the profile milling operation. In the P 997xxx.

## *1.2.1 MANAGEMENT OF TEXT PROGRAM P999997*

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

It checks if program P999997 is in the user memory. If it is not, it looks in "CARD A", if it is not there either, it assumes the ones provided by default and it copies them into the P999997 program of the user memory.

If the mainland Chinese language is selected, program P999997 is ignored. It always assumes the ones provided by default.

If when switching from M mode to MC or MCO mode, program P999997 cannot be found, because it has been erased, it is re-initialized like after power-up.

After modifying the texts of program P999997, turn the CNC off and back on to assume the new texts.

When changing a language, software version or adding conversational modes MC, MCO (new software features) the CNC carries out the following operations:

The texts that were being used are saved into "CARD A" as program P999993.

Program P999997 is erased from "CARD A"

The new default texts are assumed and are copied into program P999997 of the user memory.

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

## *1.3 POWER-UP*

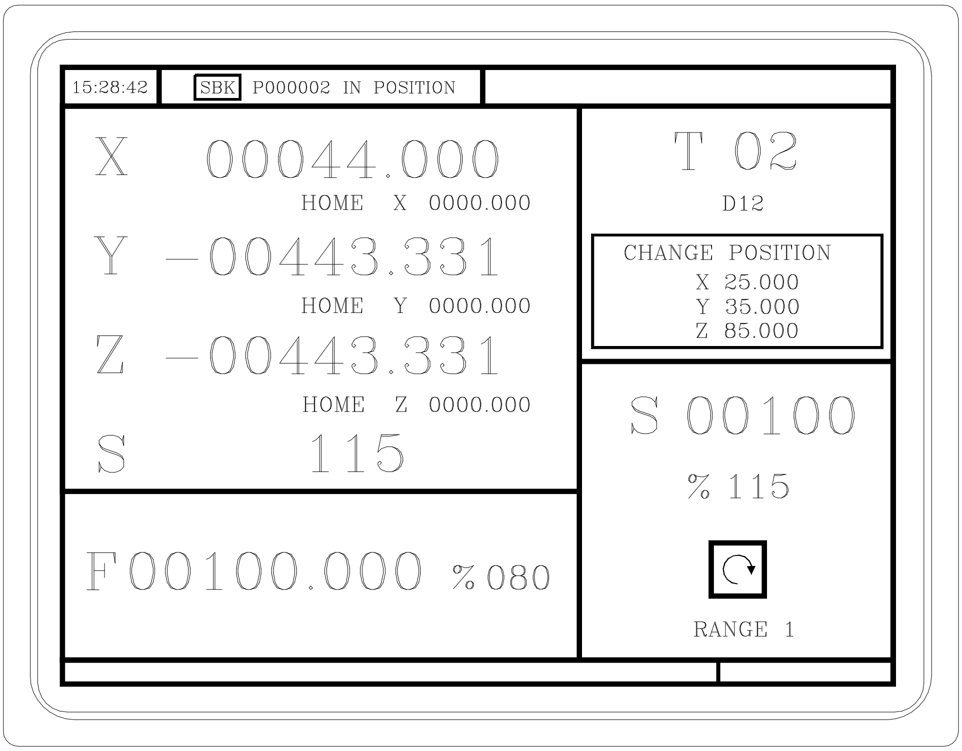
Both on CNC power-up and after the keystroke sequence: SHIFT RESET the CNC acts as follows:

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: MC mode and M mode. To switch from one mode to the other, press SHIFT ESC

The standard MC mode screen is:




**Warning**

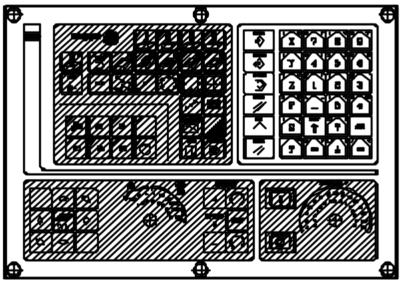
CNC setting should be done in M mode.

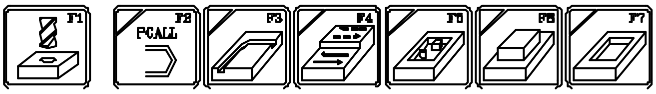
Some errors must be eliminated in the M mode.

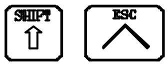
## 1.4 OPERATING IN M MODE WITH AN MC KEYBOARD

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

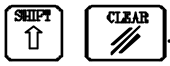
Alphanumeric keyboard:



The keys which replace softkeys F1 to F7 are: 


To switch from one operating mode to another, press key sequence 


## 1.5 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.

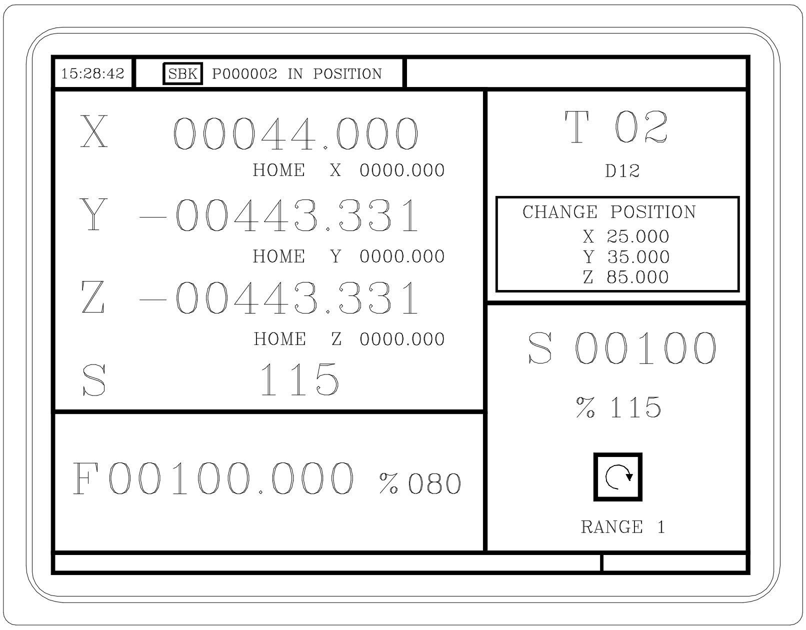
## 1.6 HANDLING THE CYCLE-START KEY

In order to avoid unwanted executions when keying sequences not supported in the MC mode, the CNC changes the color of the "CYCLE START" icon located at the top of the window from green to grey and it shows a message indicating that it is an invalid action.

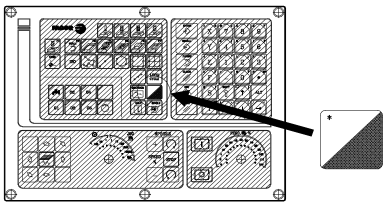
For example, if while a part-program is selected, "M3 Start" is pressed, (sequence not supported by the MC model), the CNC displays a warning message and prevents the part-program from running when detecting the "cycle start" key.

# *2. OPERATING IN JOG MODE*

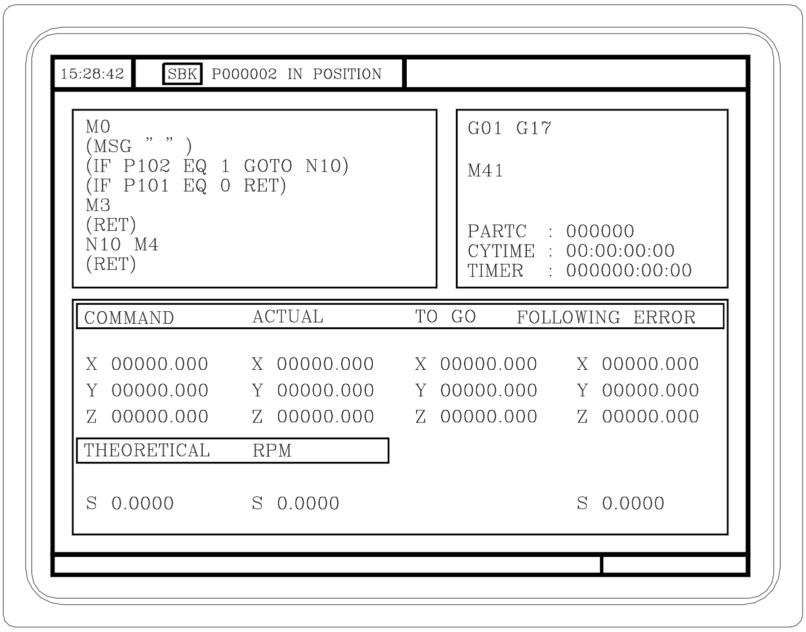
The standard MC operating mode screen is:




If one presses key

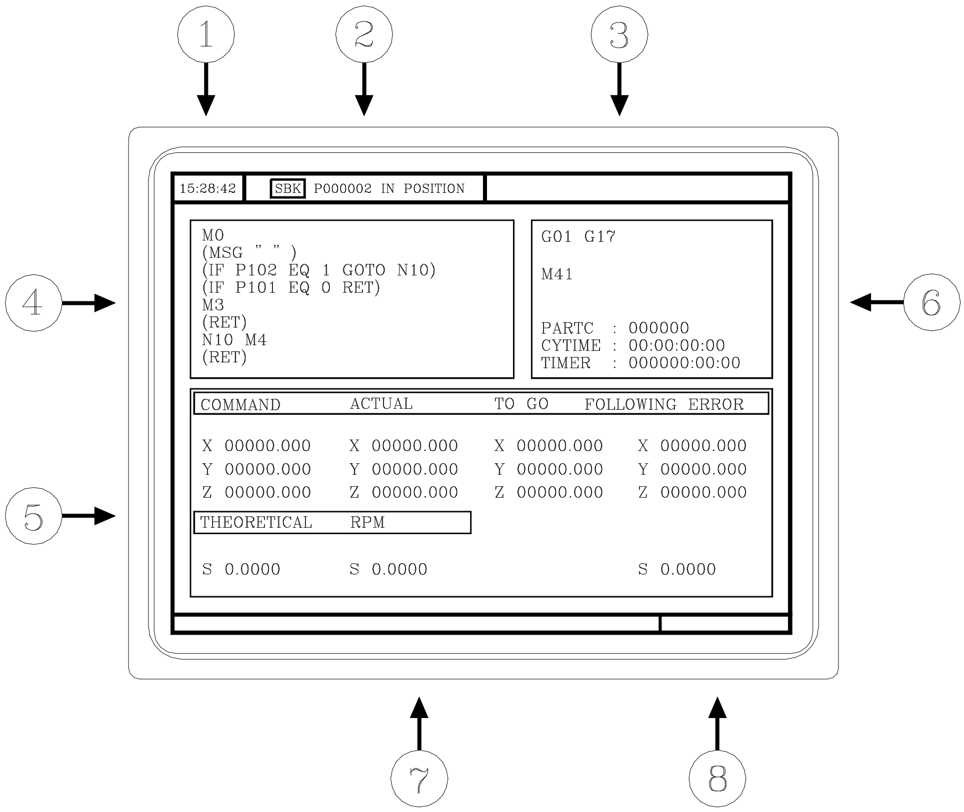


The CNC displays the special MC operating mode screen.



## 2.1 INTRODUCTION

The standard MC 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:

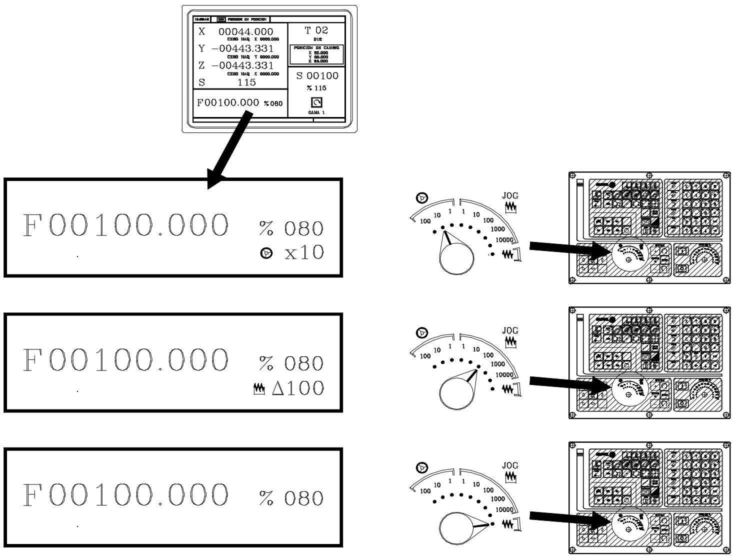
* The X, Y, Z coordinates of the axes.
* In small characters, the axis coordinates referred to machine zero reference (home). This values are very useful when allowing the operator to set a tool change position (see zone 6). The CNC does not show this data when text 33 has not defined in program 999997.
* The coordinates of the auxiliary axes which are defined.
* The real spindle rpm "S".

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

In all cases, it shows the feedrate of the «F» axes that has been selected and the % of F being applied.

When Feed-hold is active, the feedrate value changes colors.

All the possible cases are shown below.



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

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 actual spindle speed "S".

* 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 active spindle range.

* The range of the active spindle. The CNC does not display this information when text 28 of program 999997 is not defined.

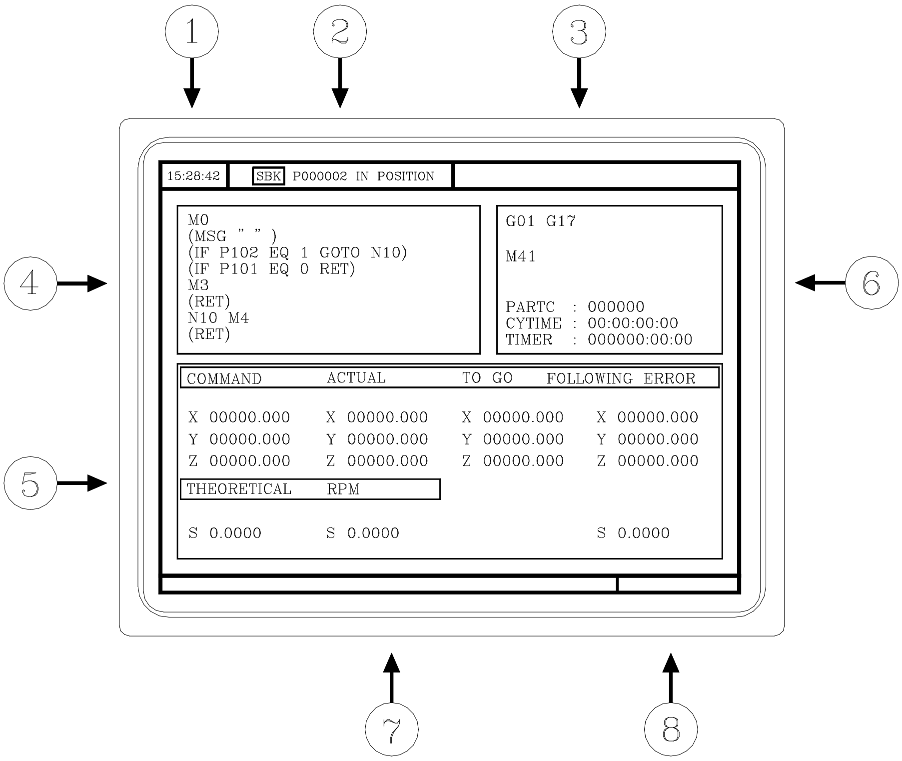
8.- Whenever a work cycle is accessed, the CNC shows the help text associated with the icon selected in this window.

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

The format and the points to be considered in the P999997 program are detailed in Chapter on "General concepts".

9.- Reserved.

The special screen for MC 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, Each axis has the following fields available:

| | |
|---|---|
| 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. |
| 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 (real) axis position.

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.

PARMC 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.

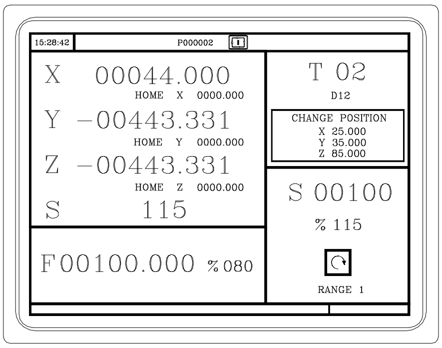
TIMER States the reading of the clock enabled by the PLC. It is expressed in format “hours : minutes : seconds”.

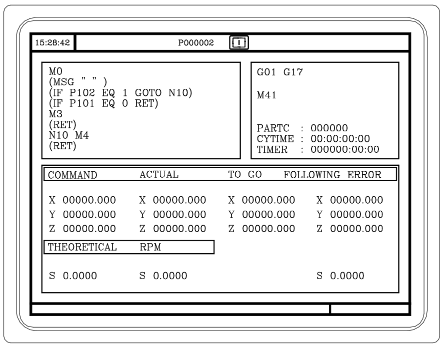
7.- Reserved.

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 deselected, the CNC deletes it from the top center window.

## 2.2 AXIS CONTROL

### 2.2.1 WORK UNITS

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

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

### 2.2.2 COORDINATE PRESET

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

1st Press the key for the axis required [X], [Y] or [Z]

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 [ESC]

3rd Press [ENTER] so that the CNC assumes said value as the new value for the point.

The CNC requests confirmation of the command. Press [ENTER] to confirm or [ESC] to exit preset.

### 2.2.3 HANDLING THE FEEDRATE OF THE AXES (F)

To set any particular value for the axis feedrate, proceed as follows:

1st Press [F]

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

2nd Enter the new feedrate required.

To exit coordinate preset press [ESC]

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

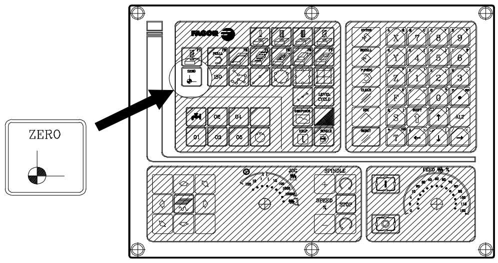
## 2.3 HOME SEARCH (MACHINE REFERENCE ZERO)

Home search can be done in 2 ways:

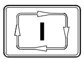
- Home search on all the axes.
- Home search on a single axis.

### *Home search on 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)

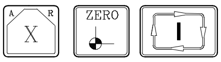
Press  ,The CNC will execute the machine reference zero routine defined by the manufacture in the general machine parameter P34 (REFPSUB).

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

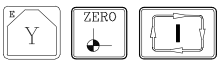
### *Home search on a single axis*

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.

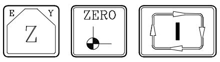
In either case, the CNC will request confirmation of the command (text 48 of program 999997)



Carries out the home search on the X axis



Carries out the home search on the Y 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). |

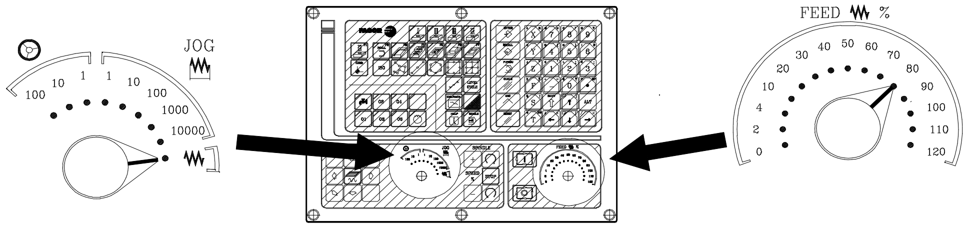
## 2.4 JOGGING THE AXES

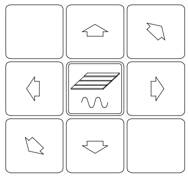
The axes of the machine can be jogged in the following ways:

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

### 2.4.1 CONTINUOUS JOG

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 "LAMCHM" general logic input the movement will be made in the following way:

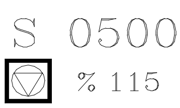
* If the PLC sets this mark at a low logic level (0V), the axis will only move while the relevant JOG key is pressed.

* 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

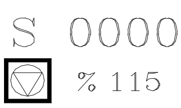
The CNC moves the axes to the F programmed.




***b) The spindle is stopped but there is a spindle speed S selected.***

The CNC calculates the corresponding feedrate in millimeters/minute and moves the axis. For example, if «F 2.000» and «S 500»:

F (mm/min) = F (rev/min.) x S = 2 x 500 = 1000 mm/min
The axis moves at a feedrate of 1000 in millimeters/minute.



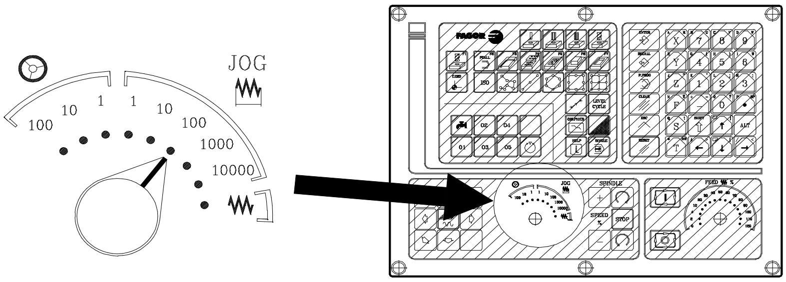

***c) The spindle is stationary and there is no spindle speed S selected.***

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

## 2.4.2 INCREMENTAL JOG

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




Incremental jog must be done one axis at a time. To do this press the JOG key for the direction of the axis to be moved.

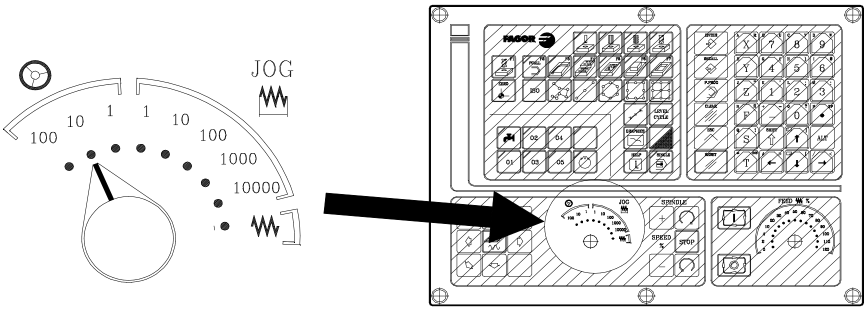
Each time a key is pressed, the corresponding axis moves the amount set by the switch. This movement effects the «F» feedrate selected.

| 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 |

## 2.4.3 *JOGGING WITH AN ELECTRONIC HANDWHEEL*

This option means the machine movements can be governed by means of an electronic handwheel.

To do this the left-hand switch has to be located in one of the positions of the handwheel




The positions available are 1, 10 and 100, all of these indicating the multiplication factor applied to the pulses provided by the electronic handwheel.

Example:

| Position of the switch | Movement per turn | | |
|---|---|---|---|
| 1 | 0.100 mm | or | 0.0100 inches |
| 10 | 1.000 mm | or | 0.1000 inches |
| 100 | 10.000 mm | or | 1.0000 inches |

***<u>The machine has an electronic handwheel</u>***

After selecting the position required on the switch, press one of the JOG keys for the axis which is to be moved. The axis selected will be displayed in small characters next to the handwheel symbol at the bottom of the screen.

If a FAGOR electronic handwheel with push button is available, the selection of the axis to be moved can also be done in the following way:

Press the push button located on the rear of the handwheel. The CNC will select the first of the axis and display this in highlighted text.

If the push button is pressed again the CNC will select the following axis, making this selection on a rotative basis.

If the push button is held down for longer than 2 seconds, the CNC will stop selecting said axis.

After selecting the axis the machine will move this as the handwheel is turned, also respecting the turning direction applied to the same.

***<u>The machine has two or three electronic handwheels</u>***

The machine will move each of the axis according to how the corresponding handwheel is turned, taking into account the position selected on the switch and also respecting the turning direction applied.

**<u>Warning:</u>**

It may occur that depending on the turning speed of the handwheel and the position of the switch, the CNC may be requested to make a movement with a feedrate higher than the maximum allowed ("G00FEED" axis machine parameter). The CNC will move the axis the amount required, but limit the feedrate to said value.

## *2.4.4 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 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
```

## *2.4.5 MASTER HANDWHEEL*

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

More handwheels need not be installed on the machine. The one currently installed will be used for the usual work mode and for this feature (Master Handwheel).

If besides having a general handwheel (general machine parameter AXIS*=11 or 12) other handwheels are associated with the axes, the CNC assumes the one associated with the X axis (general machine parameter AXIS*=21) as the Master Handwheel.

This feature must be handled by the PLC.

The PLC activates or deactivates the "master handwheel" mode through logic CNC input "MASTRHND" M5054,

M5054 = 0 Standard handwheel mode ON. M5054 = 1 Master handwheel mode ON.

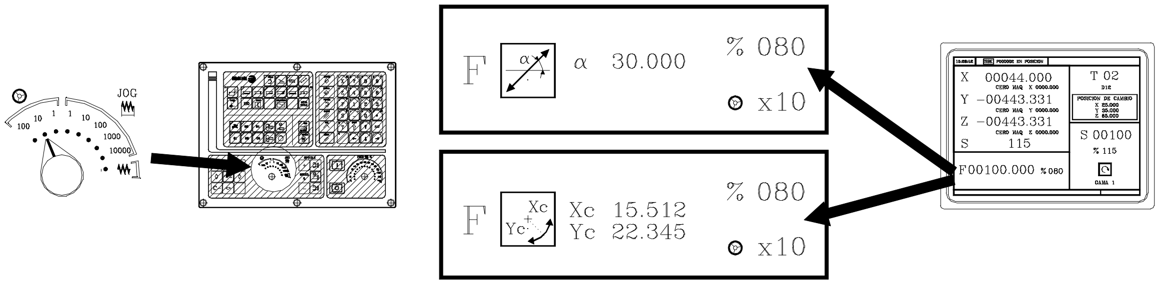
The PLC must indicate the type of jogging path to follow through logic CNC input "HNLINARC" M5053,

M5053 = 0 Linear jog M5053 = 1 Circular jog.

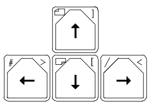
The following example uses the [O2] key to activate and deactivate the "master handwheel" mode and the [O3] to indicate the type of jog.

| | |
|---|---|
| DFU B29 R561 = CPL M5054 | Activate / deactivate the "master handwheel" mode. |
| DFU B31 R561 = CPL M5053 | Selects the type of jog, linear or circular. |

While in handwheel mode and selecting the "master handwheel", the CNC shows the following data:

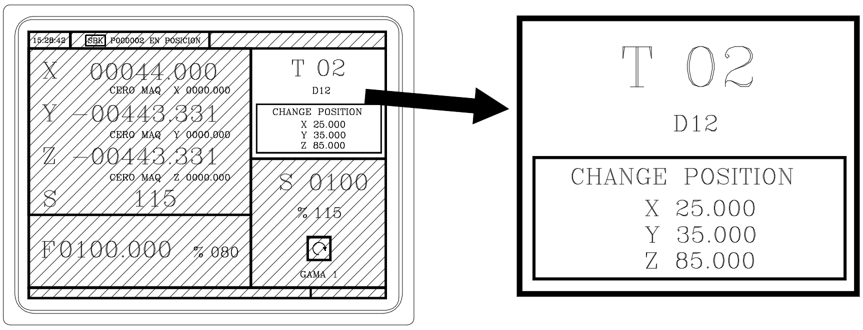



When choosing a linear jog (upper drawing) , the angle of the path must be indicated and when choosing a circular jog (lower drawing), the arc center coordinates must be indicated.

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

## *2.5 TOOL CONTROL*

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



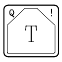

This window displays the following information:

> In large characters, the number "T" of the selected tool.

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

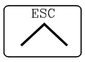
> The coordinates for the tool change point.
The CNC does not display this window when text 47 of program 999997 is not defined.

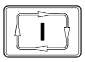
To select any other tool take the following steps:

1st Press 

The CNC will frame the tool number

2nd Enter the tool number to be selected

To exit the selection process press 


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

The CNC will handle the tool change

## *2.5.1 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 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 counterclockwise P102=1
G0 G53.... XP?? Y?? ZP??; Movement to change point
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.)

When having a Machining Center, general machine parameter "TOFFM06 (P28) = Yes", the CNC acts as follows:

If the execution of an operation or cycle involves a tool change, the CNC:
Selects the desired tool in the magazine
Executes the subroutine associated with the tool, general machine parameter "TOOLSUB (P60)"
Executes function M06 to carry out the tool change.

When selecting a new tool in JOG mode or when operating in M mode, the CNC only selects the too in the magazine and executes the associated subroutine.
The M06 function must be executed by the operator, either by programming an ISO block or by setting the PLC so it executes the M06 function when pressing a particular key. The following example uses the [O4] key: DFU B2 R562 = CNCEX1 (M06, M1)

***Note:*** On Machining Centers, the subroutine associated with the tool MUST NOT include the M06.

## *2.5.1.1 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, Y and Z of the change point.
For example: ;47 $CHANGE POSITION

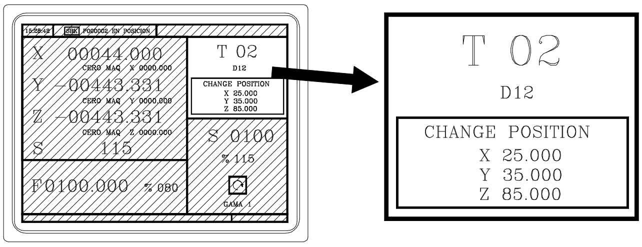
These coordinates should always refer to machine reference zero (home), for the zero offsets not to affect the tool change point.

For this reason, the CNC can display, along with coordinates X, Y, 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, P291 and P292 contain the values set by the operator as change position on X, Y, Z.




Arithmetic parameter P290
Change position on X
Arithmetic parameter P291
Change position on Y
Arithmetic parameter P292
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??? YP??? ZP??? ; Movement to the change point.
It should say: G0 G53 XP290 YP291 ZP292 ; Movement to the change point defined by the user.

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

Press key [T] for selecting field «T». Then press key for the relevant axis [X] [Y] [Z]

or keys: [↑] [↓] [←] [→]

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 the [ENTER] key

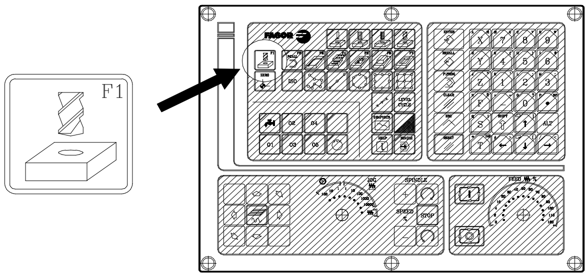
b) Assign the present position of the machine.
Move the axis, by means of the handwheel or the JOG keys, up to the point required.

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

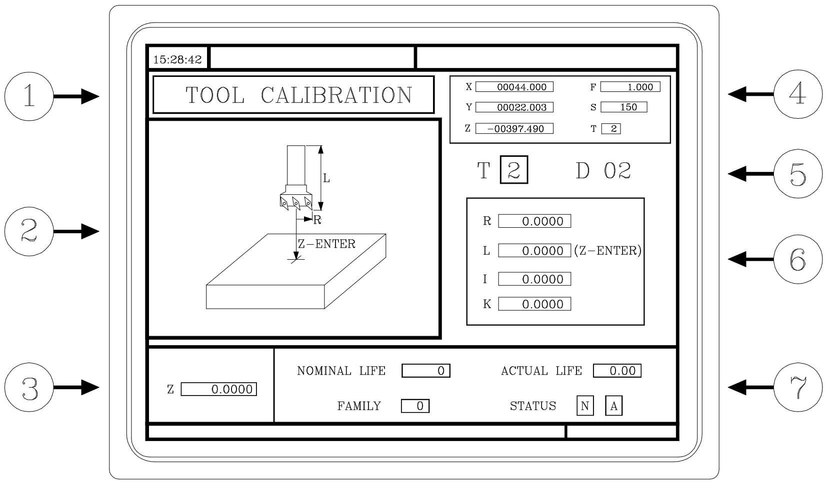
Press key [ENTER]

## 2.5.2 TOOL CALIBRATION

To access tool calibration mode press key




The CNC displays the following information:




1.- Header for the selected operating mode: «Tool calibration».

2.- Help graphics for the tool calibration.

3.- Window for tool calibration.

4.- Current machine status
Actual (real) X, Y, Z coordinates, actual axis feedrate "F", actual spindle speed "S" and "T" tool currently selected.

5.- Tool number and its Offset number.

6.- Length and offset values set in the tool offset table.

7.- Nominal life, real life, family and status of the table set in the tool table.

To calibrate the tool take the following steps:

1.- Define the tool in the tool table.
2.- Carry out tool calibration

## *2.5.2.1 DEFINE THE TOOL IN THE TOOL TABLE*

To define a tool in the tool table take the following steps:

***Select the tool number to be defined***

Press key T to select field «T»

Key in the tool number to be defined and press key RECALL

If the tool is defined, the CNC will display the values stored in the table.

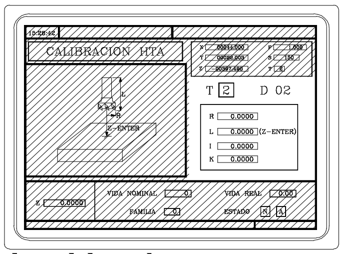
If the tool is not defined, the CNC will assign it a offset with the same number and all the data that define the geometry and lengths of the tool will be reset to value 0.

***Select the offset number to be associated with this tool***

The "D" field must be selected. If not, use the ↑ ↓ ← → keys.

Key in the offset number to be associated with the tool and press ENTER

***Define the tool dimensions***



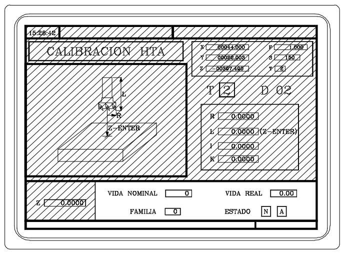

| | | | | |
|---|---|---|---|---|
| The tool data is: | R | Radius | I | Radius wear |
| | L | Length | K | Length wear |

Even if the tool length is known (L), it is recommended to measure it as indicated in the next section. Once it has been measured, the CNC updates the L and K fields.

The CNC assumes (R+I) as the real tool radius and (L+K) as the real tool length.

To set these values, select the corresponding field using the ↑ ↓ ← →, key in the desired value and press ENTER

***Define the rest of the data associated with the tool***




Nominal life. Machining time (in minutes) or number of operations that the tool may carry out.

Actual life. Machining time already elapsed or number of operations already carried out.

Family code. Used with automatic tool changer.
0 ... 199. normal tools, 200 ... 255 special tools.

When requesting a worn-out (expired) tool ("actual life" greater than "nominal life"), the CNC will select the next tool in the table belonging to the same family instead of the one requested.

Tool status. They are 2 fields for internal CNC data. They cannot be modified.

| | |
|---|---|
| N = Normal (family 0-199) | A = Available |
| S = Special (family 200-255) | E = Expired (“actual life” greater than “nominal life”) |
| | R = Rejected by the PLC |

To define these values, select the corresponding field using the ↑ ↓ ← → key in the desired value and press ENTER

## 2.5.2.2 TOOL MEASUREMENT

There are 2 ways of measuring a tool.

a) Using a tool calibrating table.

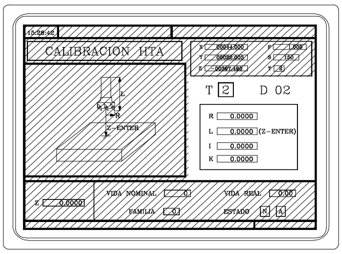
Using the window containing the tool dimensions to set that data.

b) Not using a tool calibrating table. The measurements are carried out with the CNC.

Use the Tool Calibration window.

***a) Set the tool length or modify the tool length offsets***

This window show the dimensions assigned to the selected tool.




"R" and "L" indicate the tool Radius and Length.

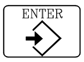
"I" and "K" indicate the offset the CNC has to apply to compensate for tool wear.

The CNC adds the "I" value to the radius "R" and the "K" value to the length "L" for calculating the real dimensions (R+I) and (L+K) to be used.

Every time the R or L value is defined, the CNC sets the I and K fields, respectively, to zero.

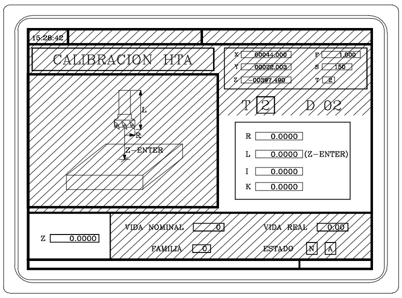
The "I" and "K" values are accumulative. That is, if the "I" value is 0.20 and a value of 0.05 is entered, the CNC assigns a value of 0.25 to the "I" field.

When defining I=0 or K=0, each one of them is reset to "0".

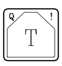
To change one of these values, select the corresponding field , key in the desired value and press 

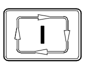

***b) Tool measurement***

The window on the right contains the tool dimensions and the one on the lower left-hand side the data necessary to measure it.




To access the tool calibration window (bottom left) and thus carry out tool calibration, the tool must be selected on the machine.

Otherwise, press 

key in the tool number and press 

Select the bottom left window using the

Key in the Z coordinate of the part used for calibration and press ENTER

Tool measurement. Length only.

Approach the tool to the part and touch it with it.

Press Z ENTER

The tool is now calibrated. The CNC assigns the length "L" corresponding to it and resets its "K" field to "0".

The tool radius "R" has to be entered manually.

To calibrate another tool:

Select at the machine: T number

Approach the tool to the part and touch it with it.

Then, Z ENTER

### *2.5.2.3 MODIFY VALUES WHILE IN EXECUTION*

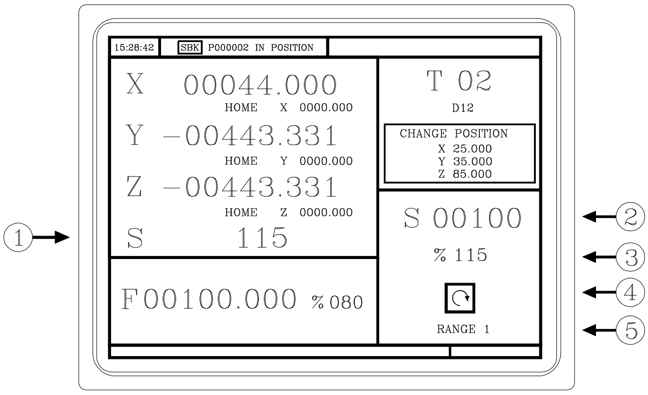
The tool values (dimensions and geometry) may be modified without having to interrupt program execution.

To do this, press , the CNC will show the Tool Calibration screen with all the data corresponding to the active tool being possible to change its data or that of any other tool.

To exit this screen, press ESC

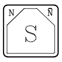
## 2.6 SPINDLE CONTROL

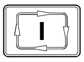
The standard MC work mode shows the following information about the spindle.



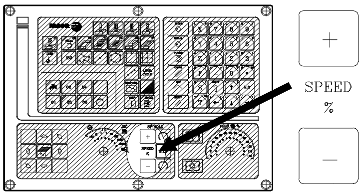

1.- Actual (real) spindle speed in rpm.

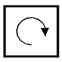
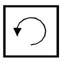
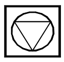
2.- Theoretical spindle speed in rpm.

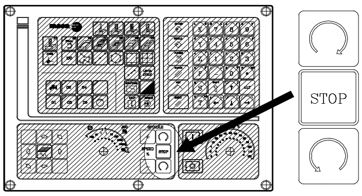
To select another speed, press  the CNC highlights the current value

Key in the new value and press  The CNC assumes that value and updates the real spindle speed.

3.- % of the theoretical spindle speed being applied.

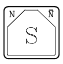
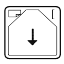
To change this percentage, press 

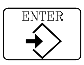
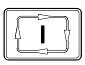

4.- Spindle status:  turning clockwise,  turning counterclockwise or  stopped.

To change the spindle status, press: 


5.- Currently selected spindle speed range.

When using an automatic tool changer, this value cannot be modified.

When NOT using an automatic tool changer, press  and then use the  key until the current value is highlighted.

Enter the range number to be selected and press  or 

Note: When the machine does not have spindle ranges, this message is useless. That is why the CNC does not show this message when text number 28 has not be defined in program 999997.

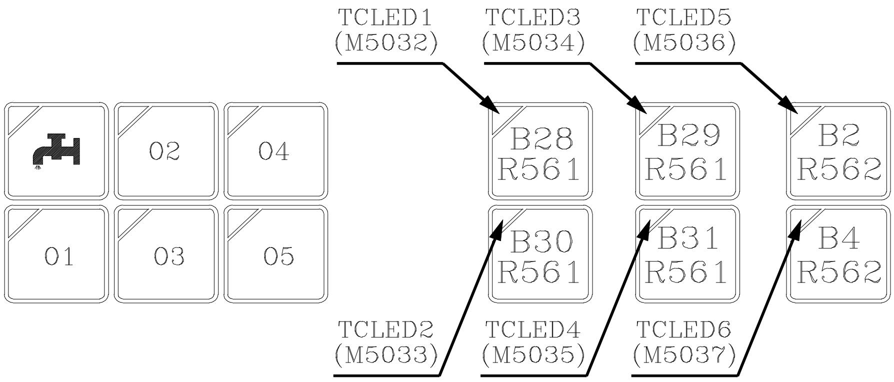
## 2.7 CONTROL OF EXTERNAL DEVICES

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 MCLED* input variables shown in the figure being available for this purpose.

Examples:

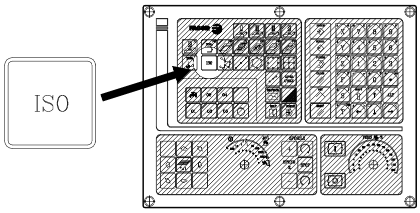
Control of the coolant: DFU B28R561 = CPL MCLED1
= CPL O33

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 MCLED2
= CPL O34

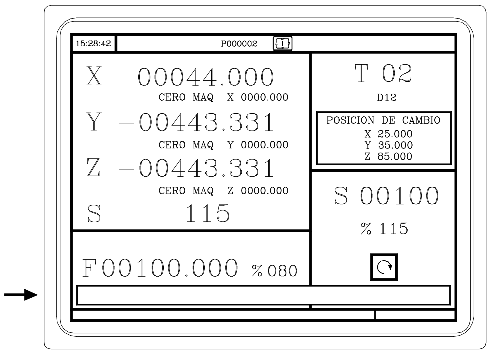
## 2.8 ISO CODE MANAGEMENT

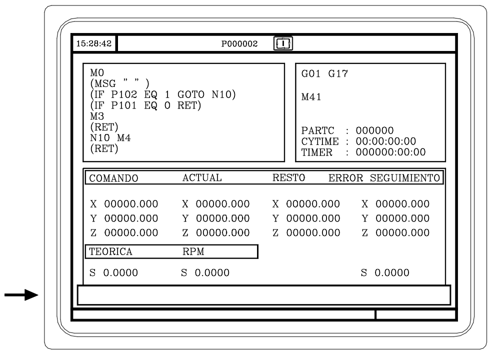
The ISO key gives access to the MDI mode or to the ISO work mode.




To access the MDI mode, the JOG mode must be selected and then press ISO

The CNC displays a window at the bottom of the standard (or special) screen.




In this window, an ISO-coded block may be edited and then executed just like in MDI mode of the "M model" work mode

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

When accessing the ISO mode, a special screen comes up where up to 6 program blocks may be edited in ISO code 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 display the symbol.

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

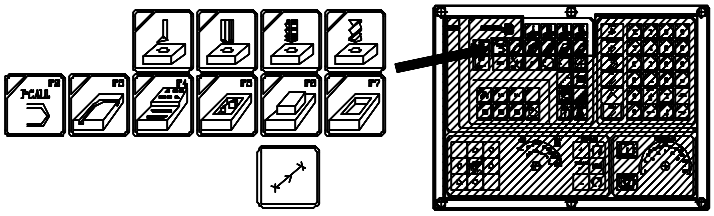
Press GRAPHICS to simulate and ENTER to execute.

It is possible to combine blocks edited in ISO code with machining cycles (standard and/or user defined) to make up part-programs. The chapter on "Program storage" describes how to do it and how to operate with them.

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

# *3. WORKING WITH OPERATIONS OR CYCLES*

The following keys of the CNC must be used to select the machining operations or cycles:



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

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

Once all the necessary data has been defined, the operator may Simulate or Execute the cycle just like any other standard cycle of the MC mode.

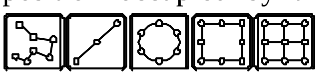
When pressing any other key, the CNC selects the corresponding machining operation or cycle changing the display and lighting up the indicator lamp of the key just pressed.

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

- Boring operation (2 levels)
- Reaming operation
- Tapping operation
- Drilling (3 levels) & center punching
- Profile milling operation (2 levels)
- Surface and slot milling
- Pocket with 2D and 3D profile
- Rectangular & circular boss
- Rectang. (2) & circular (2) pocket
- Positioning (2 levels)

When the machining operation or cycle has several levels, the [LEVEL CYCLE] key must be pressed to select the desired cycle level:

The Boring , Reaming, Tapping, Drilling and Center punching operations may be carried out at the position occupied by the tool or they could be associated with a positioning by means of



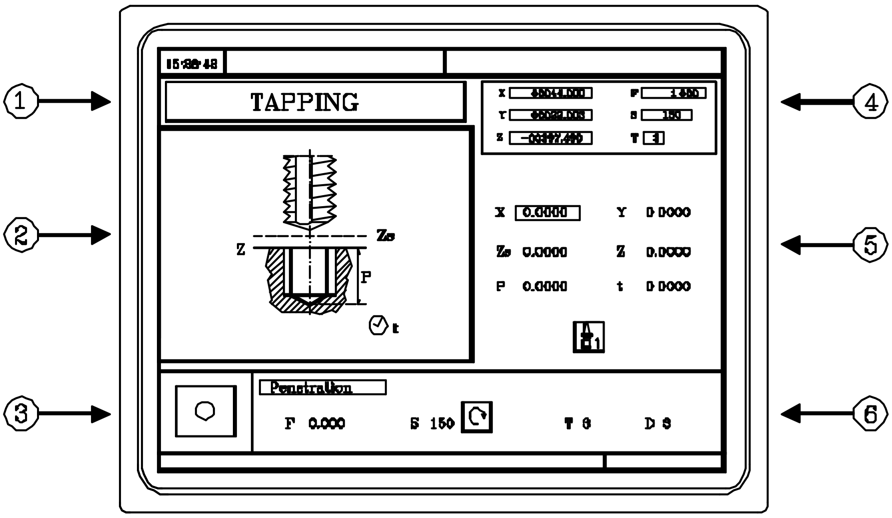
With this CNC, it is possible to combine ISO-coded blocks with standard and/or user-defined machining operations to create part-programs as described in the chapter on "Part-program storage".

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).

## 3.1 OPERATION EDITING MODE

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




1.- Name of the selected operation or cycle.

2.- Help graphics.

3.- When referred to positioning, it indicates the associated operation

4.- Current machine status. Coordinates and machining conditions.

5.- Data defining the geometry of the machining operation.

6.- Machining conditions for the operation.

The CNC will highlights an icon, a coordinate or one of the operation (or cycle) defining data.

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

a) Use the ← → ↑ ↓ keys, the CNC selects the previous one or the next one.

b) Press X Y or Z The CNC selects the first coordinate for that axis. By pressing that key again, it will select the next coordinate for that axis.

c) Press F or T The CNC selects the corresponding roughing data . By pressing that key again, the corresponding finishing data is selected.

d) Press S The CNC selects the "S" roughing data. By pressing that key again, the finishing "S" data is selected.

## *3.1.1 DEFINITION OF THE MACHINING CONDITIONS*

Some operations keep the same machining condition during the whole execution process (boring, reaming, etc.)

Other operations use certain machining conditions for roughing and other conditions for finishing (pockets, bosses, etc.)

This section describes how to define all this data.

***Axis feedrate (F)***

Place the cursor over this data, key in the desired value and press

***Spindle speed (S)***

Place the cursor over this data, key in the desired value and press

***Spindle turning direction***

Place the cursor over this data and press

***Machining tool (T)***

Place the cursor over this data, key in the desired value and press

It is also possible to access the Tool calibration mode to check or change the data corresponding to the selected tool. To do this, place the cursor over the "T" field and press

To quit the tool calibration mode and return to the cycle, press

***Roughing pass (D)***

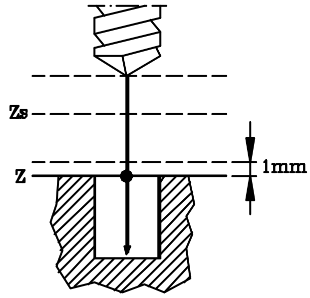
Place the cursor over this data, key in the desired value and press

***Finishing stocks (d, dz)***

Place the cursor over this data, key in the desired value and press

## 3.1.2 SAFETY PLANE

In all operations, there are four work planes.




**Starting plane** or tool position when calling the cycle. It does not have to be defined.

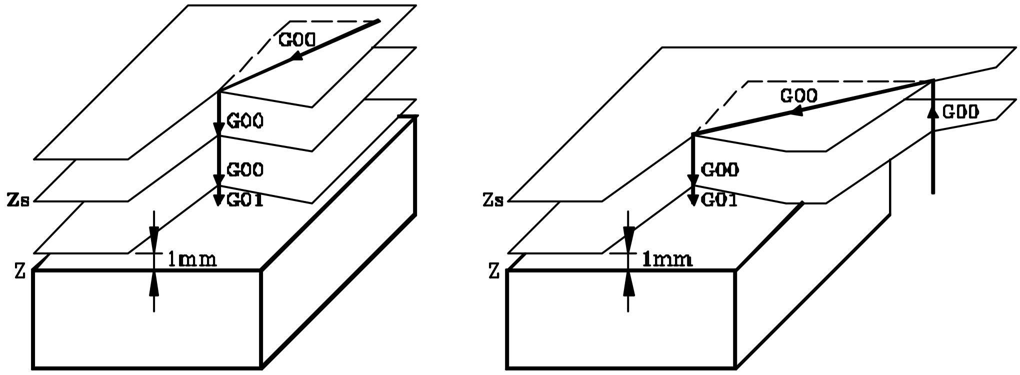
**Safety plane**. It is used for the first approach and for withdrawing the tool after the machining operation. It is defined with parameter **Zs**.

**Approach** (to the part) **plane**. It does not have to be defined. The CNC calculates ti, at 1 mm off the part surface.

**Part surface**. It is defined with parameter **Z**.

The tool moves in rapid (G00) to the safety plane (Zs), it keeps on going in rapid to the approach plane (up to 1mm off the part surface) and, finally, it moves at machining feedrate (G01) down to the part surface.

The approach to the part surface depends on the tool position.



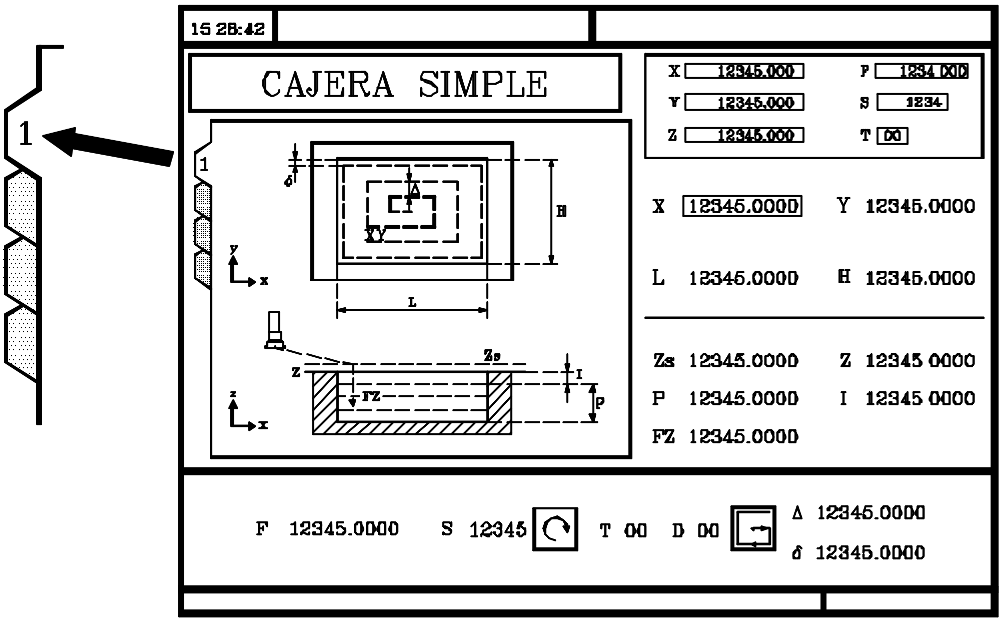

If it is above the safety plane (left drawing), it first moves on X and Y and then on Z.

If it is below the safety plane (right drawing), it first moves on Z up to the safety plane, then on X and Y and finally on Z down to the part surface.

## 3.1.3 CYCLE LEVEL

All the cycles have several editing levels.

Each level has its own screen and the main window of the cycle indicates (with tabs) the available levels and which one is currently selected.

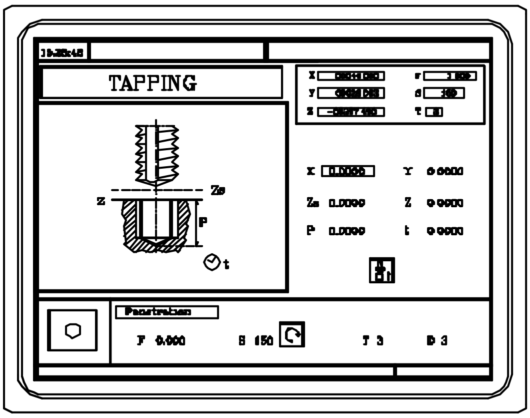



To change levels, use the 

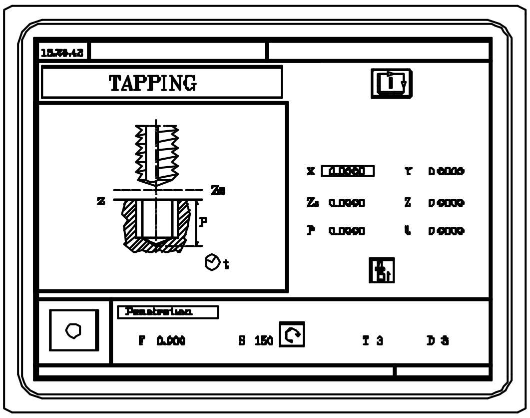
key or the "Page up" and "Page down" keys to scroll up and down through the different levels.

## 3.2 *SIMULATION AND EXECUTION OF THE OPERATION*

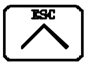
There are 2 ways to work with operations or cycles: Editing and Execution modes.



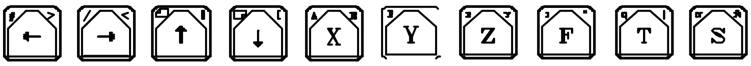
Editing mode



Execution mode

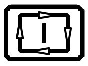
Press  to switch from the Editing mode to the Execution mode.

Press one of these keys to switch from the Execution mode to the Editing mode:




The operation or cycle can be simulated in any of the two modes. To do that, press 

For further information refer to the chapter on "Execution and Simulation".

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

For further information refer to the chapter on "Execution and Simulation".

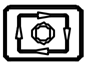
## *3.2.1 BACKGROUND CYCLE EDITING*

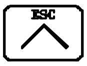
While executing a part-program, it is possible to edit an operation or cycle at the same time (background editing).

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

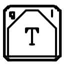
The operation being edited in the background cannot be executed or simulated nor the current axis position be assigned to a coordinate.

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

Press  => to interrupt execution and resume background editing.

Press 

=> To quit background editing.

Press 

=> To go into tool inspection.

Pressing the [T] key while in background editing, it selects the "T" field of the operation or canned cycle being edited.

**Warning**

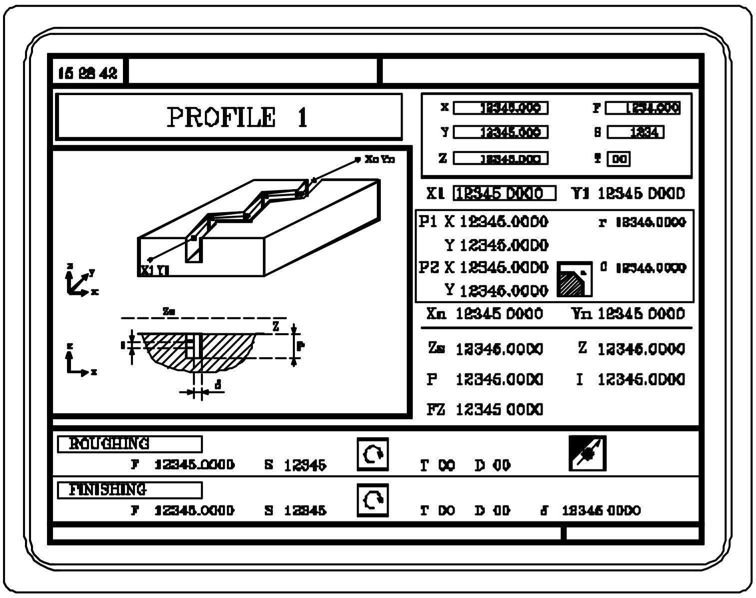
Background editing is not possible while executing an independent operation or cycle. It is only possible while executing a part-program.

## 3.3 PROFILE MILLING OPERATION

Press  to select the profile milling operation

This cycle may be defined in two ways:

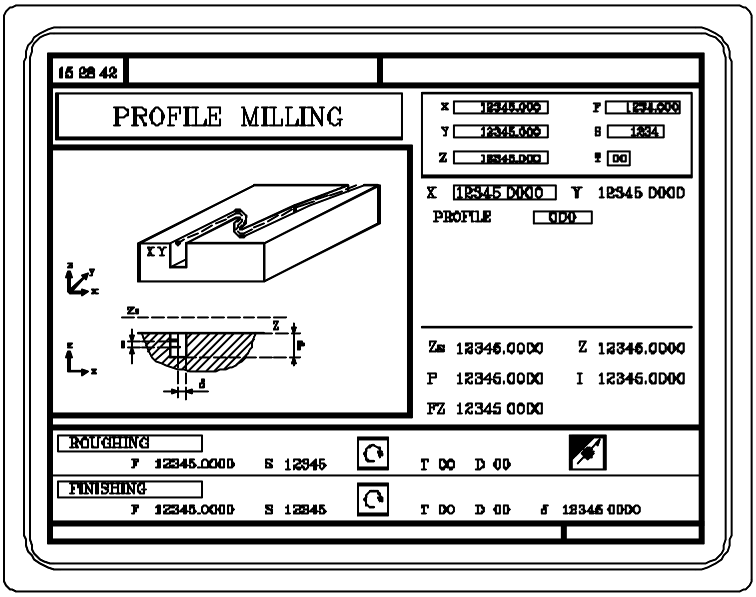
Level 1.




One must define: The starting point (X1, Y1), the intermediate points (P1 through P12), the end point (Xn, Yn) and the machining conditions in Z (Zs, Z, P, I, Fz)

On the other hand, in the data area for the roughing operation, one must define whether the milling operation is to be carried out with or without tool radius compensation.

Level 2.



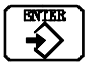

One must define The starting point (X, Y), the "Profile Program" number and the machining conditions in Z (Zs, Z, P, I, Fz)

On the other hand, in the data area for the roughing operation, one must define whether the milling operation is to be carried out with or without tool radius compensation.

### *3.3.1 DATA DEFINITION*

***Coordinates of the starting and end points***

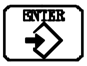
These coordinates are defined one at a time. Once the cursor is over the coordinates of the axis to be defined, one can:

a) Enter the value by hand. Key in the desired value and press 

b) Assign the current position of the machine.

Jog the axis, with the handwheel or the JOG keys up to the desired point. The upper right-hand window shows the tool position at all times.

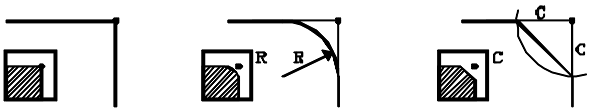
Press  for the selected data to assume the value appearing in the upper right-hand window.

Press 

***Intermediate points (Level 1)***

The intermediate points are defined one at a time. At each point, one must define:

The X, Y coordinates are defined one at a time like those for the starting end points.

The type of corner 

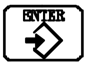

To select the type of corner, place the cursor over the icon and press 

When not using all 12 definition points, the first unused point must be defined with the same coordinates as those of the last point of the profile.

***Machining conditions in Z (Zs, Z, P, I, Fz)***

The machining conditions are defined one by one.

The Zs and Z values are defined like the coordinates of the starting and end points.

To define the rest of the values (P, I, Fz), place the cursor in the corresponding window, key in the desired value and press 

If the penetration step is programmed with a positive sign (I+), the cycle recalculates the step so allthe penetrations are the same with a value equal to or smaller than the one programmed.

If it is programmed with a negative sign (I-), the cycle machines with the given step except on the last one where it machines the rest.

***Milling with or without tool radius compensation***

 Without tool radius compensation

 With left-hand tool radius compensation

 With right-hand tool radius compensation

To select the type of tool compensation, place the cursor over the icon and press 

## 3.3.2 PROFILE DEFINITION (LEVEL 2)

To define the "Profile program" one can:

***Key in the "Profile Program" number directly.***

If the "Profile program" number is known, key it in and press ENTER

***Access the "Profile Program" directory to select one of them***

Press ↓ The cycle will show a window with the profile programs already defined.

To move around within a window, use ↑ ↓

Place the cursor over the desired program and press ENTER

To exit this window without selecting any program, use ← →

***Edit a new "Profile program"***

To edit a new "Program", key the program number (between 0 and 999) and press RECALL

The CNC will display the window for the profile editor (see Operating manual of the M and MC models, chapter 4 section "Profile Editor").

Once the profile has been edited, the CNC requests a comment to be associated with the "Profile Program" just edited.

Enter the desired comment and press ENTER

If no comment is desired, press ESC

***Modify an existing "Profile program".***

To modify a "Program", key in its number and press RECALL

The CNC will display the profile currently defined in the window for the profile editor.

One can: Add new elements at the end of the current profile.
Modify the data of any element.
Modify or insert chamfers, roundings, etc.
Delete elements from the element.

***Delete an existing "Profile Program".***

Press ↓ the cycle will show the profile programs already defined.

Place the cursor over the "Profile Program" to be deleted and press CLEAR

The CNC will request confirmation.

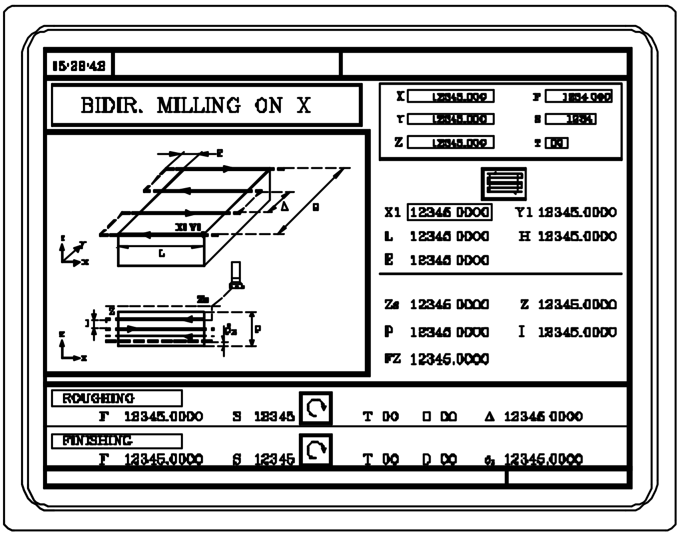
***Notes:*** The profile programs can also be accessed in the "M" mode because the CNC saves them internally as P 997xxx.
Example: Profile program 123 is internally stored as P997123.

When saving a part-program containing a level-2 profile cycle out to an external device, PC, floppy disk unit, etc. its associated profile program P997xxx must also be saved.

## *3.4 SURFACE AND SLOT MILLING MILLING OPERATIONS*

Press to select these operations.

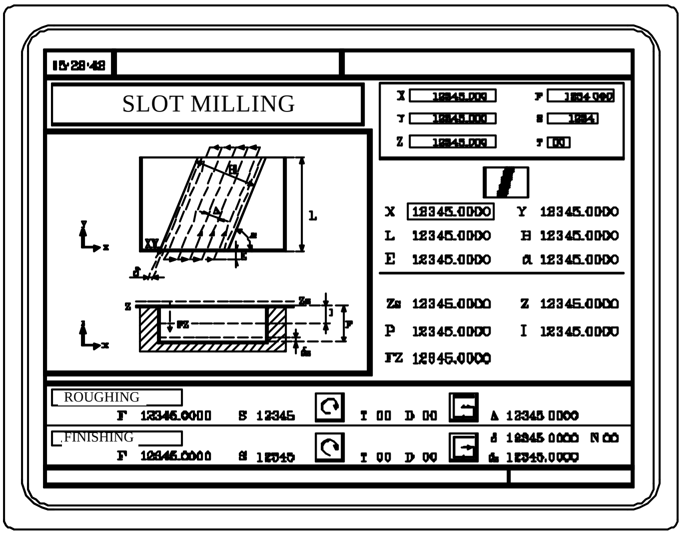
Surface milling operation:




One must define — The type of milling, the starting point (X1, Y1), the dimensions of the surface to be milled (L, H, E) and the machining conditions in Z (Zs, Z, P, I, Fz)

On the other hand, one must define the milling step (Δ) in the data area for the roughing operation and the finishing stock (δZ) in the data area for the finishing operation.

Slot milling operation:



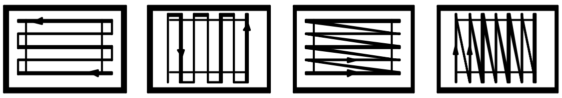

One must define — The type of slot milling, the starting point (X1, Y1), the dimensions of the slot to be milled (L, H, E) and the machining conditions in Z (Zs, Z, P, I, Fz).

In the roughing area, define the milling step (Δ) and the machining direction

In the finishing area, define the finishing stocks (δ and δZ), the number of finishing passes and the machining direction

## *3.4.1 SURFACE MILLING DATA DEFINITION*

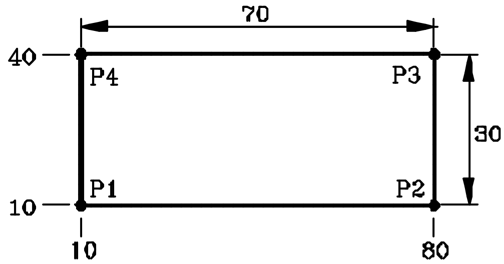
***Type of surface milling***



To select the type of surface milling, place the cursor over this icon and press 

***Surface to mill (X1, Y1, L, H, E, a)***

Define one of the corners of the surface to be milled (X1, Y1), the length (L) and the width (H). The sign of L and H indicates the orientation with respect to the point X1, Y1.



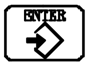

P1 X1=10 Y1=10
L=70 H=30

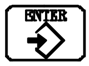
P2 X1=80 Y1=10
L=−70 H=30

P3 X1=80 Y1=40
L=−70 H=−30

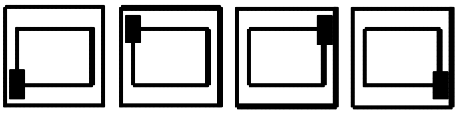
P4 X1=10 Y1=40
L=70 H=−30

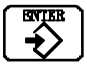
The X1, Y1 coordinates may be defined by:

a) Entering the value by hand. Key in the desired value and press 

b) Assigning the current position of the machine.
Move the axis to the desired point with the handwheel or the JOG keys. The top right window shows the tool position at all times.
Press  so the selected data assumes the value shown in the top right window and press 

Once the surface to be milled has been defined, the icon shown at the bottom right (area for roughing and finishing) allows selecting the corner where it will start milling. Possible values:

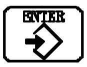


The "E" and "**a**" data are defined one by one. Go to the relevant window, key in the desired value and press 

When programming an "E" value smaller than the tool radius, the CNC mills with an "E" value equal to the tool radius.

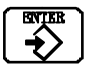
***Machining conditions in Z (Zs, Z, P, I, Fz)***

They are defined one by one. The Zs and Z values are defined like the starting and end points.

To define the rest of the values (P, I, Fz), place the cursor in the corresponding window, key in the desired value and press 

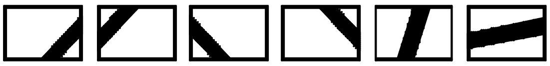
If the penetration step is programmed with a positive sign (I+), the cycle recalculates the step so allthe penetrations are the same with a value equal to or smaller than the one programmed.
If it is programmed with a negative sign (I-), the cycle machines with the given step except on the last one where it machines the rest.

***Milling step (D) and finishing stock (dz)***

They are defined one by one. Place the cursor in the corresponding window, key in the desired value and press 

## *3.4.2 SLOT MILLING DATA DEFINITION*

***Type of slot milling***



To select the type of slot milling, place the cursor over this icon and press 

***Coordinates of the starting point***

These coordinates are defined one by one. After placing the cursor over the axis coordinates to be defined, it is possible:

a) To enter the value by hand. Key in the value and press ENTER

b) Assign the current axis position.

Jog the axis to the desired point with the handwheel or with the JOG keys. The upper right-hand window shows the tool position at all times.

Press RECALL for the selected data to assume the value displayed in the upper right-hand window.

Press ENTER

***Slot dimensions (L, H, E, a)***

They are defined one by one. Place the cursor in the corresponding window, key in the desired value and press ENTER

When programming parameter «E» with a smaller value than the tool radius, the CNC executes the planning with an «E» value equal to the tool radius.

***Machining conditions in Z (Zs, Z, P, I, Fz)***

The machining conditions are defined one by one.

The Zs and Z values are defined like the starting and end points.

To define the rest of the values (P, I, Fz), place the cursor in the corresponding window, key in the desired value and press ENTER

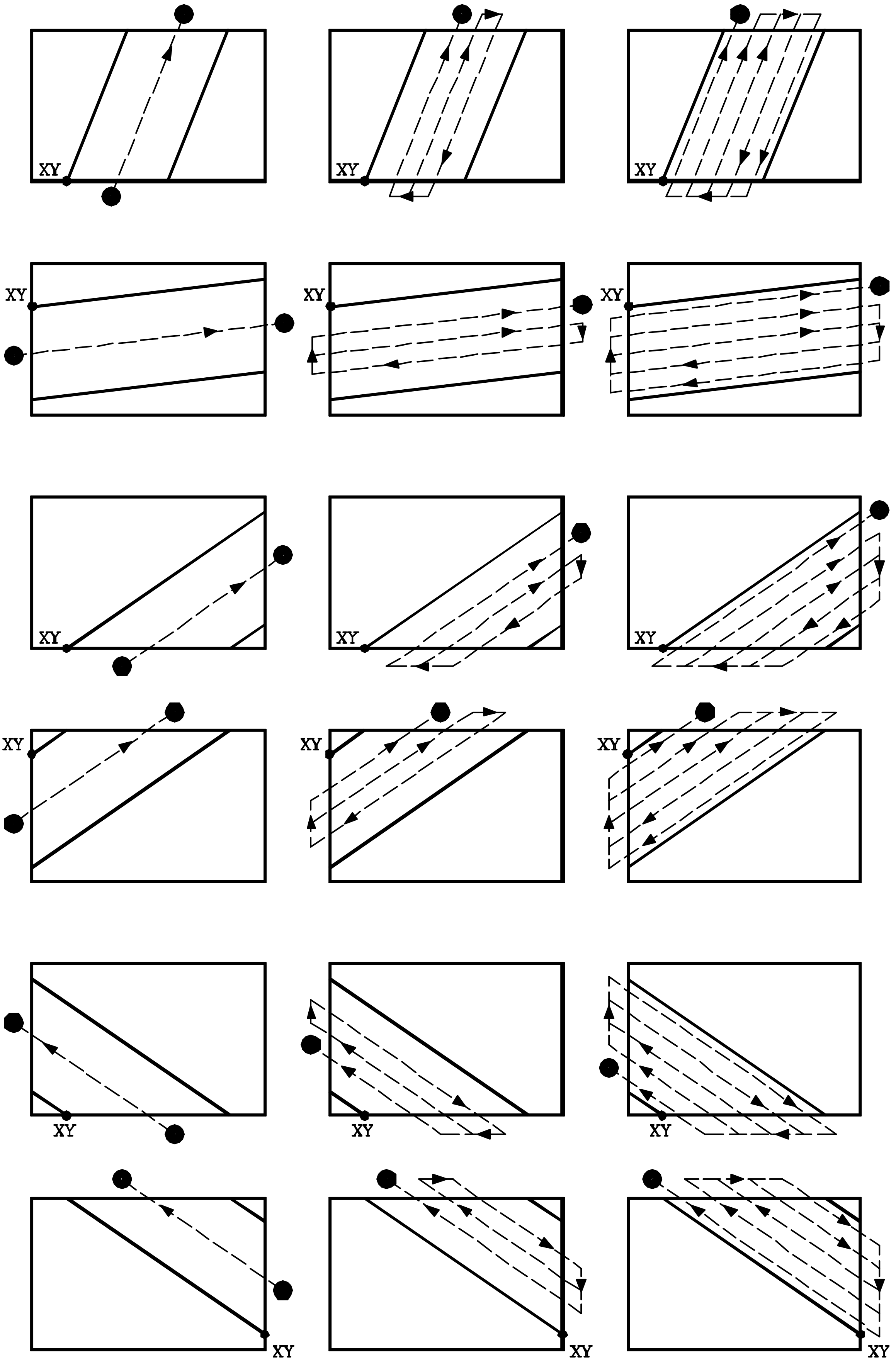
If the penetration step is programmed with a positive sign (I+), the cycle recalculates the step so allthe penetrations are the same with a value equal to or smaller than the one programmed.

If it is programmed with a negative sign (I-), the cycle machines with the given step except on the last one where it machines the rest.

***Milling step (D) and finishing stock (dz)***

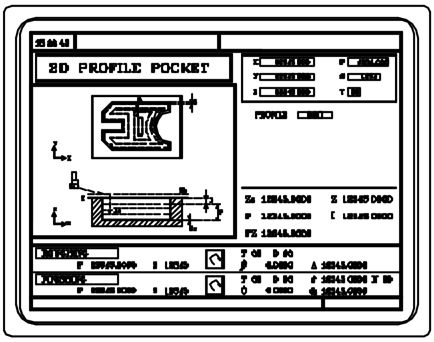
They are defined one by one. Place the cursor in the corresponding window, key in the desired value and press ENTER

Milling the different slot types, clockwise:

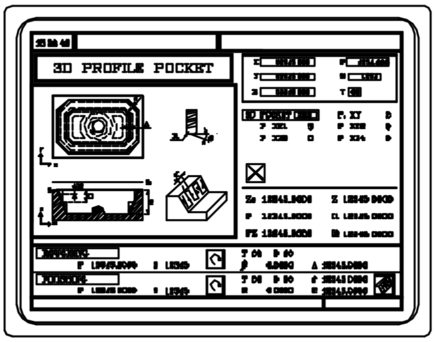


## *3.5 POCKET CYCLE WITH A PROFILE*

To select a profile milling operation, press 
This cycle may be defined in two different ways:

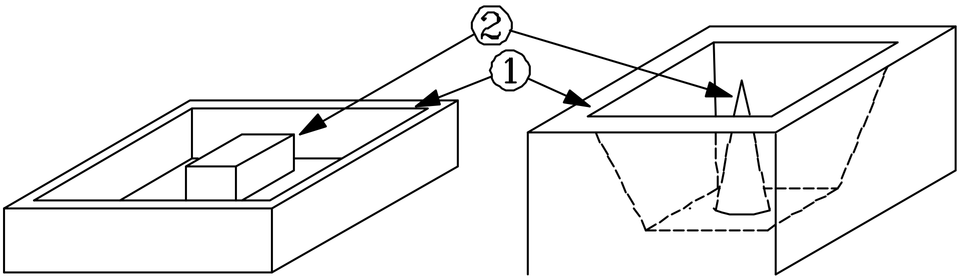



Pocket with 2D profile




Pocket with 3D profile

A pocket consists of contour or outside profile (1) and a series of contours or profiles internal to it. These inside profiles are referred to as islands.




2D pockets (upper left-hand figure) have all the walls of the outside profile plus those of the vertical islands. 3D pockets (upper right-hand figure) may have one, several or all the walls of the outside profile and/or those of the non-vertical islands (up to a maximum of 4).

### ***Programming pockets with 2D profiles***

When defining the Profile, one must specify the contour or contours of the islands besides the outside contour of the pocket.

The machining operation in Z is defined by means of :

| | |
|---|---|
| Coordinate of the safety plane ................ Zs | Coordinate of the part surface ..................... Z |
| Pocket depth ............................................ P | Step in Z ..................................................... I |
| Penetration feedrate in Z ........................ Fz | |

In the area for the roughing operation data, define:

| | |
|---|---|
| The lateral penetration angle ..................... β | The milling pass ......................................... Δ |

In the area for the finishing operation data, define:

| | |
|---|---|
| The lateral penetration angle ..................... θ | Finishing stock on the walls ......................... δ |
| Finishing stock at the bottom ................... δz | Number of finishing passes in Z ................. N |

***Programming pockets with 3D profiles***

Pocket ID number. (3D POCKET)

It is possible to have several 3D pockets. The CNC associates with each 3D pocket all its data (surface profile, depth profile, machining conditions, etc.)

Surface profile or profile in the X,Y plane. Profile (P. XY).
It must indicate the contour or contours of the possible islands besides the outside contour of the pocket.

Depth profile corresponding to the profile defined first. Profile (P. Z1)
It usually corresponds to the outside contour of the pocket.

Depth profile corresponding to the profile defined second . Profile (P. Z2)
It usually corresponds to the contour of the island defined first.

Depth profile corresponding to the profile defined third. Profile (P. Z3)
It usually corresponds to the contour of the island defined second.

Depth profile corresponding to the profile defined fourth. Profile (P. Z4)
It usually corresponds to the contour of the island defined third.

Once all the profiles have been defined, the 3D pocket configuration must be validated. To do so, place the cursor over the icon and press ENTER

The cycle will show the icon.

The machining operation in Z is defined by means of:

| | |
|---|---|
| The coordinate of the safety plane .......... Zs | Coordinate of the part surface ..................... Z |
| Pocket depth ........................................... P | Roughing pass in Z .................................... I1 |
| Penetration feedrate in Z ........................ Fz | Semi-finishing pass in Z ............................. I2 |

In the area for roughing data, the following must be defined:

| | |
|---|---|
| Lateral penetration angle .......................... β | Milling pass ............................................... Δ |

In the area for finishing data, the following must be defined:

| | |
|---|---|
| Tool tip radius ......................................... R | Finishing stock on walls ............................... δ |
| Finishing pass ....................................... (e) | |

Direction of the finishing passes on the walls

***Notes:*** The pocket configuration program and the profile programs can also be accessed in the "M" mode since the CNC stores them internally as:

P995xxx 3D pocket configuration
P998xxx The XY plane profiles in 2D and 3D pockets
P996xxx Depth profiles in 3D pockets

## *3.5.1 DATA DEFINITION*

***Machining conditions in Z (Zs, Z, P, Fz, I, I1, I2)***

The machining conditions must be defined one by one.

To define the values (P, Fz, I, I1, I2), place the cursor in the corresponding window, key in the desired value and press ENTER

If the penetration step is programmed with a positive sign (I+), the cycle recalculates the step so allthe penetrations are the same with a value equal to or smaller than the one programmed.
If it is programmed with a negative sign (I-), the cycle machines with the given step except on the last one where it machines the rest.

To define the values (Zs and Z), after placing the cursor in the corresponding window, one may:

a) Enter the value by hand. Key in the desired value and press ENTER

b) Assign the current axis position.

Jog the axis to the desired point with the handwheel or the JOG keys. The upper right-hand window shows the tool position at all times.

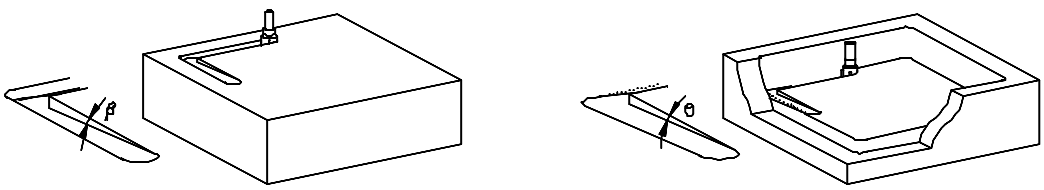
Press RECALL for the selected data to assume the value shown in the upper right-hand window.

Press ENTER

### ***Milling pass (D) and finishing passes (e)***

Place the cursor in the corresponding window, roughing or finishing operation, key in the desired value and press ENTER

### ***Lateral penetration angles (b, q)***



Place the cursor in the corresponding window, roughing or finishing operation, key in the desired value and press ENTER

### ***Finishing stocks: on the walls (d) and at the bottom (dz)***

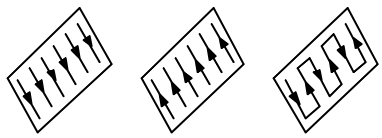
Place the cursor in the corresponding window, finishing operation, key in the desired value and press ENTER

### ***Finishing tool tip radius (R)***

Place the cursor in the corresponding window, finishing operation, key in the desired value and press ENTER

### ***Direction of the finishing passes on the walls***

To select the direction of the finishing passes on the walls, place the cursor over this icon and press



## 3.5.2 PROFILE DEFINITION

To define a Profile, after selecting the corresponding window, one can:

***Key in the "Profile Program" number directly.***

If the "Profile program" number is known, key it in and press ENTER

***Access the "Profile Program" directory to select one of them***

Press ↓ The cycle will show a window with the profile programs already defined. There are 3 directories, one for the pocket configuration profiles, another one for the XY plane profile and a third one for the depth profile of the pocket.

To move around within a window, use ↑ ↓

Place the cursor over the desired program and press ENTER

To exit this window without selecting any program, use ← →

***Edit a new "Profile program"***

To edit a new "Program", key the program number (between 0 and 999) and press RECALL

The CNC will display the window for the profile editor (see Operating manual of the 8055 M CNC, chapter 4 section "Profile Editor").

Once the profile has been edited, the CNC requests a comment to be associated with the "Profile Program" just edited.

Enter the desired comment and press ENTER

If no comment is desired, press ESC

***Modify an existing "Profile program".***

To modify a "Program", key in its number and press RECALL
The CNC will display the profile currently defined in the window for the profile editor.

One can: Add new elements at the end of the current profile.
Modify the data of any element.
Modify or insert chamfers, roundings, etc.
Delete elements from the element.

***Delete an existing "Profile Program".***

Press ↓ the cycle will show the profile programs already defined.

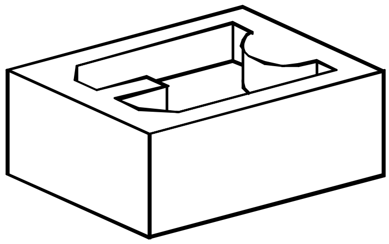
Place the cursor over the "Profile Program" to be deleted and press CLEAR
The CNC will request confirmation.

***Notes:*** Profile programs can also be accessed in the "M" mode since the CNC stores them internally as:
P995xxx The 3D configuration profiles
P998xxx The XY plane profiles in 2D and 3D pockets.
P996xxx The depth profiles of 3D pockets

When saving a part-program containing a pocket cycle with profile out to an external device, PC, floppy disk unit, etc. its associated profile programs must also be saved.

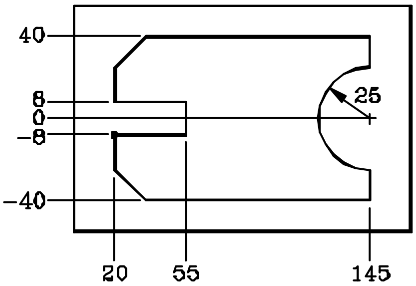
## *3.5.3 EXAMPLES OF PROFILE DEFINITION*

***Example of how to define a 2D pocket without islands:***



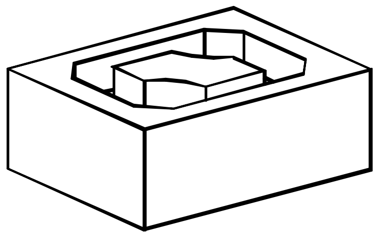
2D Pocket Profile 1 Recall
Configuration
Abscissa axis: X   Ordinate axis: Y
Autozoom: Yes   Validate
Profile

| | | | | |
|---|---|---|---|---|
| Starting Point | X 20 | Y -8 | Validate | |
| Straight | X 20 | Y -40 | Validate | |
| Straight | X 145 | Y -40 | Validate | |
| Straight | X 145 | Y -25 | Validate | |
| Clockwise arc | Xf 145 | Yf 25 | R 25 | Validate |
| Straight | X 145 | Y 40 | Validate | |
| Straight | X 20 | Y 40 | Validate | |
| Straight | X 20 | Y 8 | Validate | |
| Straight | X 55 | Y 8 | Validate | |
| Straight | X 55 | Y -8 | Validate | |
| Straight | X 20 | Y -8 | Validate | |



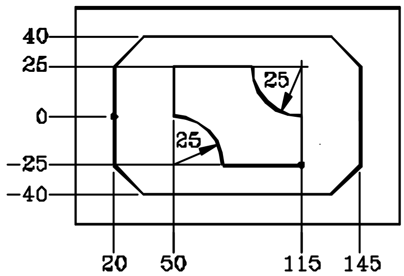

Corners   Chamfer
Select lower left-hand corner   Enter   Chamfer 15 Enter
Select upper left-hand corner   Enter   Chamfer 15 Enter
Escape
Finish   Save Profile

***Example of how to define a 2D profile with islands:***



2D Pocket Profile 2 Recall
Configuration
Abscissa axis: X   Ordinate axis: Y
Autozoom: Yes   Validate
Profile (outside profile)

| | | | |
|---|---|---|---|
| Starting Point | X 20 | Y 0 | Validate |
| Straight | X 20 | Y -40 | Validate |
| Straight | X 145 | Y -40 | Validate |
| Straight | X 145 | Y 40 | Validate |
| Straight | X 20 | Y 40 | Validate |
| Straight | X 20 | Y 0 | Validate |



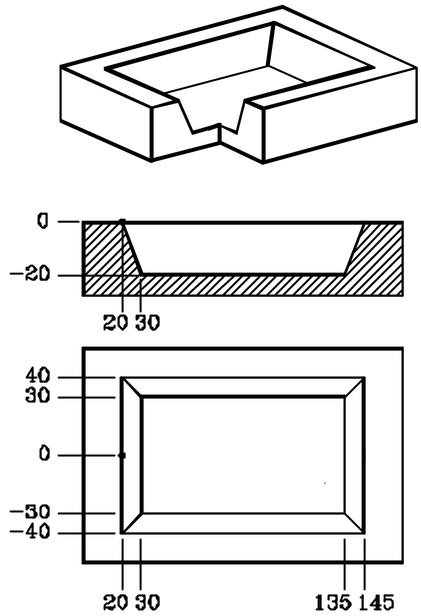

Corner   Chamfer
Select lower left-hand corner   Enter
Chamfer 15   Enter
Select lower right-hand corner   Enter
Chamfer 15   Enter
Select upper right-hand corner   Enter
Chamfer 15   Enter
Select upper left-hand corner   Enter
Chamfer 15   Enter
Escape
New Profile   (island)
Profile

| | | | | | | |
|---|---|---|---|---|---|---|
| Starting Point | X 115 | Y -25 | Validate | | | |
| Straight | X 115 | Y 0 | Validate | | | |
| Clockwise arc | Xf 90 | Yf 25 | Xc 115 | Yc 25 | R 25 | Validate |
| Straight | X 50 | Y 25 | Validate | | | |
| Straight | X 50 | Y 0 | Validate | | | |
| Clockwise arc | Xf 75 | Yf -25 | Xc 50 | Yc -25 | R 25 | Validate |
| Straight | X 115 | Y -25 | Validate | | | |

Finish   Save Profile

***Example of how to define a 3D pocket without islands:***

3D Pocket= 1




P.XY= 3 Recall
Configuration
Abscissa axis: X    Ordinate axis: Y
Autozoom: Yes    Validate
Profile (outside profile)

| | | | |
|---|---|---|---|
| Starting Point | X 20 | Y 0 | Validate |
| Straight | X 20 | Y -40 | Validate |
| Straight | X 145 | Y -40 | Validate |
| Straight | X 145 | Y 40 | Validate |
| Straight | X 20 | Y 40 | Validate |
| Straight | X 20 | Y 0 | Validate |

Finish    Save Profile

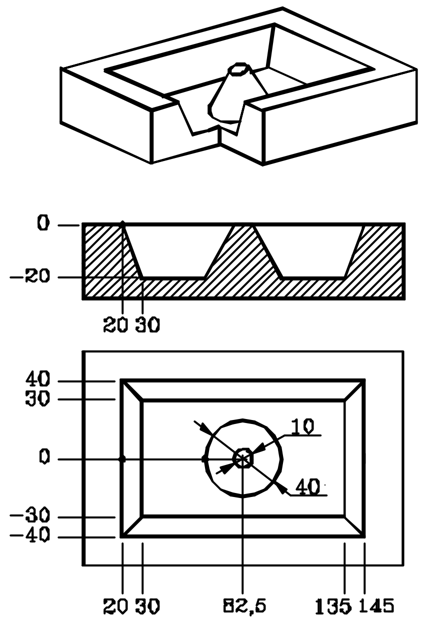
P.Z1= 1 Recall
Configuration
Abscissa axis: X    Ordinate axis: Z
Autozoom: Yes    Validate
Profile (depth profile)

| | | | |
|---|---|---|---|
| Starting Point | X 20 | Z 0 | Validate |
| Straight | X 30 | Z -20 | Validate |

Finish    Save Profile

***Example of how to define a 3D pocket with islands:***

3D Pocket= 2




P.XY= 4 Recall
Configuration
Abscissa axis: X    Ordinate axis: Y
Autozoom: Yes    Validate
Profile (outside profile)

| | | | |
|---|---|---|---|
| Starting Point | X 20 | Y 0 | Validate |
| Straight | X 20 | Y -40 | Validate |
| Straight | X 145 | Y -40 | Validate |
| Straight | X 145 | Y 40 | Validate |
| Straight | X 20 | Y 40 | Validate |
| Straight | X 20 | Y 0 | Validate |

New Profile (island)
Circle    X 62.5    Y0    Xc 82.5    Yc 0    Validate
Finish    Save Profile

P.Z1= 2 Recall
Configuration
Abscissa axis: X    Ordinate axis: Z
Autozoom: Yes    Validate
Profile (outside depth profile)

| | | | |
|---|---|---|---|
| Starting Point | X 20 | Z 0 | Validate |
| Straight | X 30 | Z -20 | Validate |

Finish    Save Profile

P.Z2= 3 Recall
Configuration
Abscissa axis: X    Ordinate axis: Z    Autozoom: Yes    Validate
Profile (depth profile of the island)

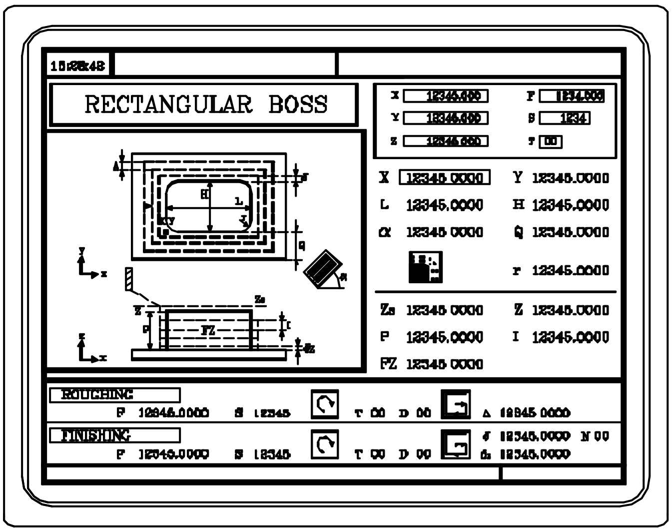
| | | | |
|---|---|---|---|
| Starting Point | X 77,5 | Z 0 | Validate |
| Straight | X 62,5 | Z -20 | Validate |

Finish    Save Profile

## *3.6 RECTANGULAR AND CIRCULAR BOSS CYCLES*

To select the boss cycles, press

Rectangular Boss cycle

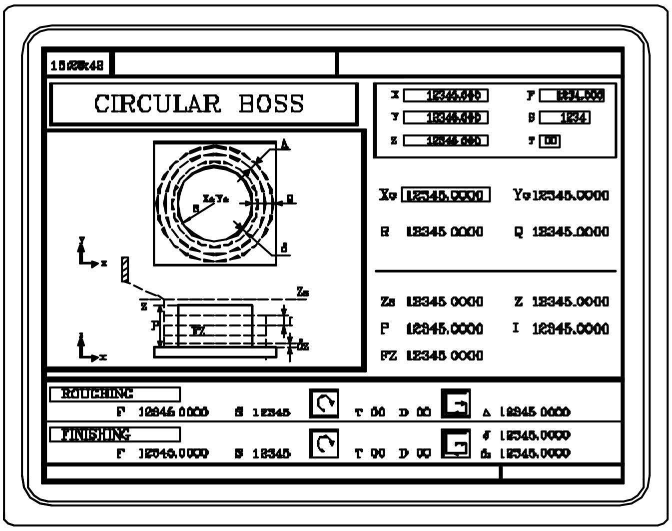



One must define The starting point (X,Y), the dimensions of the boss (L,H), the inclination angle (a), the amount of material to be removed (Q), the type of corner and the machining conditions in Z (Zs, Z, P, I, Fz)

In the roughing area, define the milling step ($\Delta$) and the machining direction

In the finishing area, define the finishing stocks ($\delta$ and $\delta z$), the number of finishing passes and the machining direction

Circular boss cycle




One must define The center coordinates (Xc, Yc), the boss radius (R), the amount of material to be removed (Q) and the machining conditions in Z (Zs, Z, P, I, Fz)

In the roughing area, define the milling step ($\Delta$) and the machining direction

In the finishing area, define the finishing stocks ($\delta$ and $\delta z$), the number of finishing passes and the machining direction

## *3.6.1 DATA DEFINITION*

***Coordinates of the starting point***

These coordinates are defined one by one. After placing the cursor over the axis coordinates, one can:

a) Enter the value by hand. Key in the desired value and press

b) Assign the position of the axis.

Jog the axis to the desired point with the handwheel or the JOG keys. The upper right-hand window shows the tool position at all times.

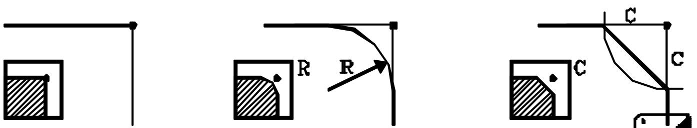
Press for the selected data to assume the value appearing in the upper right-hand window.

Press

***Rectangular Boss: Dimensions, inclination angle and material to be removed***

They are defined one by one. Place the cursor in the corresponding window, key in the desired value and press

***Type of corner***




To select the type of corner, place the cursor over this icon and press

***Circular: Center coordinates, radius and material to be removed***

They are defined one by one.

The center coordinates (Xc, Yc) are defined like the starting and end points.

To define the rest of the values (R, Q) place the cursor in the corresponding window, key in the desired value and press

***Machining conditions in Z (Zs, Z, P, I, Fz)***

They are defined one by one.

The Zs and Z values are defined like the starting and end points.

To define the rest of the values (P, I, Fz), place the desired value and press

If the penetration step is programmed with a positive sign (I+), the cycle recalculates the step so allthe penetrations are the same with a value equal to or smaller than the one programmed.
If it is programmed with a negative sign (I-), the cycle machines with the given step except on the last one where it machines the rest.

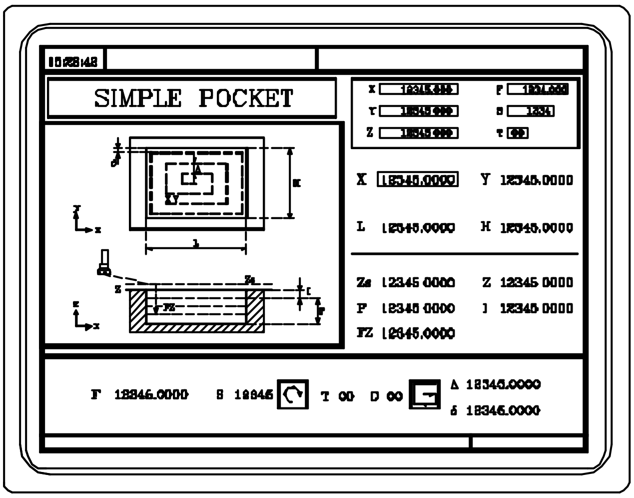
***Milling pass (D)***
***Finishing stocks: on the walls (d) and at the bottom (dz)***
***Number of finishing passes (N)***

Place the cursor in the window for the finishing operation, key in the desired value and press

## *3.7 RECTANGULAR AND CIRCULAR POCKET CYCLES*

To select these cycles press

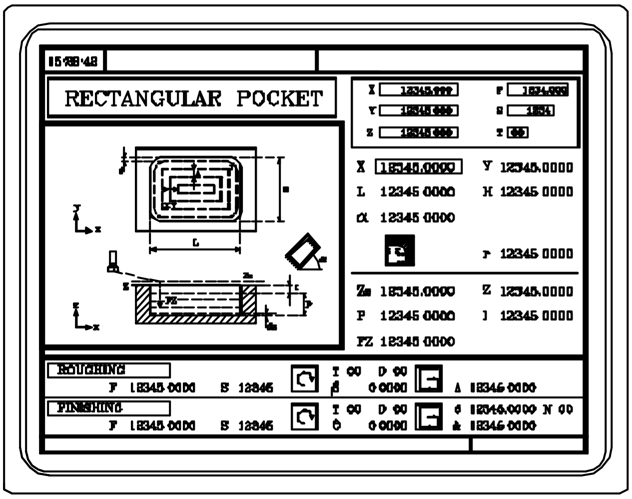
Rectangular pocket cycle (Level 1)




One must define: The starting point (X, Y), the pocket dimensions (L, H) and the machining conditions in Z (Zs, Z, P, I, Fz)

One must also define the milling pass (Δ), the finishing stock (δ) and the machining direction

Rectangular pocket cycle (Level 2)

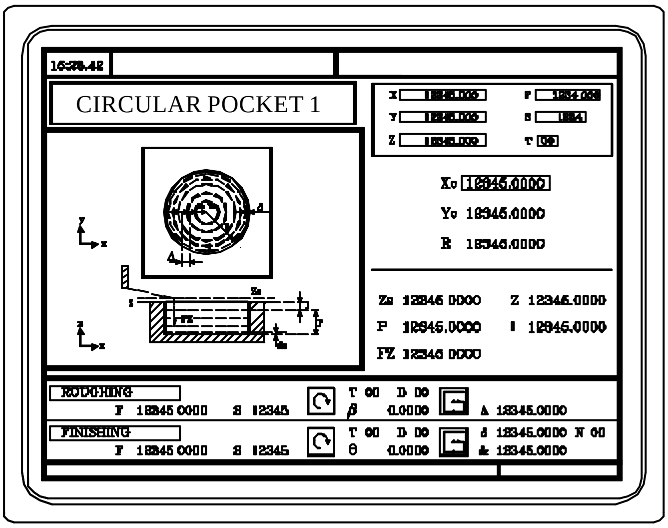



One must define: The starting point (X, Y), the dimensions of the pocket (L, H), the inclination angle (a), the type of corner and the machining conditions in Z (Zs, Z, P, I, Fz)

In the area for roughing data, define the lateral penetration angle (β), the milling pass (Δ) and the machining direction

In the area for finishing data, define the lateral penetration angle (θ), the finishing stocks (δ and δz), the number of finishing passes (N) and the machining direction

Circular pocket cycle (level 1)




One must define

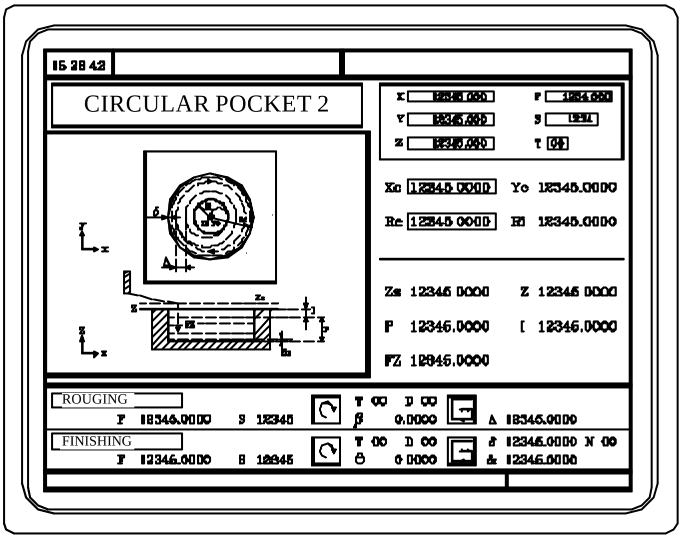
The center coordinates (Xc, Yc), the pocket radius (R) and the machining conditions in Z (Zs, Z, P, I, Fz)

In the area for roughing data, define the lateral penetration angle (β), the milling pass (Δ) and the machining direction

In the area for finishing data, define the lateral penetration angle (θ), the finishing stocks (δ and δz), the number of finishing passes (N) and the machining direction

Circular pocket cycle (level 2)

Useful for machining pre-emptied pockets or rings




One must define

The center coordinates (Xc, Yc), the inside radius (Ri), the outside radius (Re) and the machining conditions in Z (Zs, Z, P, I, Fz)

In the area for roughing data, define the lateral penetration angle (β), the milling pass (Δ) and the machining direction

In the area for finishing data, define the lateral penetration angle (θ), the finishing stocks (δ and δz), the number of finishing passes (N) and the machining direction

## *3.7.1 DATA DEFINITION*

***Coordinates of the starting point***

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

a) Enter the value by hand. Key in the desired value and press ENTER

b) Assign the current axis position.

Jog the axis to the desired point with the handwheel or the JOG keys. The upper right-hand window shows the tool position at all times.

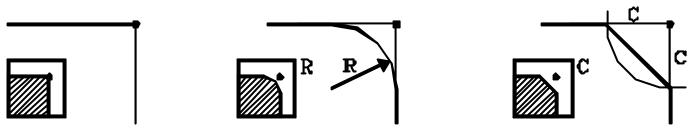
Press RECALL for the selected data to assume the value appearing in the upper right-hand window.

Press ENTER

***Rectangular Pocket: Dimensions and inclination angle***

They are defined one by one. Place the cursor in the corresponding window, key in the desired value and press ENTER

***Type of corner***




To select the type of corner, place the cursor over this icon and press

If, due to the dimensions of the pocket, the specified rounding or chamfer is not possible, the CNC will make a chamfer or rounding with the largest possible value.

***Circular pocket: Center coordinates and radii***

They are defined one by one.

The center coordinates (Xc, Yc), are defined like the starting and end point.

To define the radii (R, Ri, Re), place the cursor in the corresponding window, key in the desired value and press ENTER

***Machining conditions in Z (Zs, Z, P, I, Fz)***

They are defined one by one.

The Zs and Z values are defined like the starting and end point.

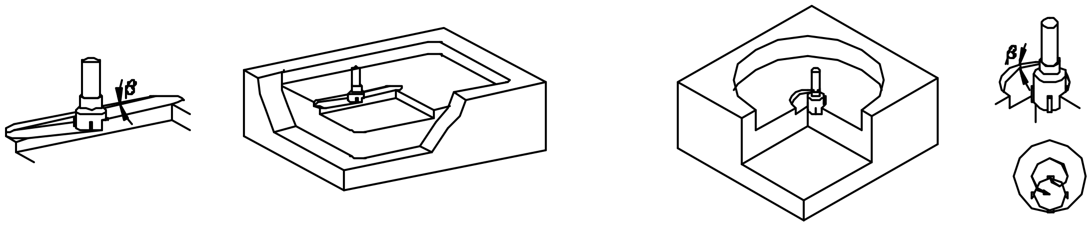
To define the rest of the values (P, I, Fz), place the cursor in the corresponding window, key in the desired value and press ENTER

If the penetration step is programmed with a positive sign (I+), the cycle recalculates the step so allthe penetrations are the same with a value equal to or smaller than the one programmed.
If it is programmed with a negative sign (I-), the cycle machines with the given step except on the last one where it machines the rest.

***Milling pass (D)***
***Finishing stocks: on walls (d) and at the bottom (dz)***
***Number of finishing passes (N)***

Place the cursor in the window for finishing operation, key in the desired value and press ENTER

***Lateral penetration angle (b, q)***




In the rectangular pocket, the penetration is carried out from the center of the pocket out and following the first machining path. The passes are carried out as often as needed and the operation always concludes at the center of the pocket.

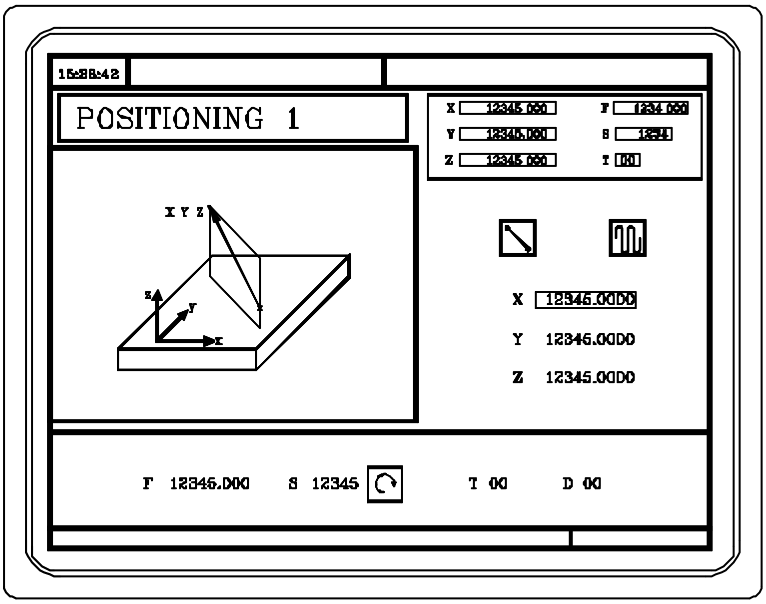
In the circular pocket, the penetration is carried out from the center of the pocket following a helical path with a radius equal to the that of the tool while keeping the machining direction. The penetration always ends at the center of the pocket.

Place the cursor in the corresponding window, roughing or finishing operation, key in the desired value and press ENTER

## *3.8 POSITIONING (2 LEVELS)*

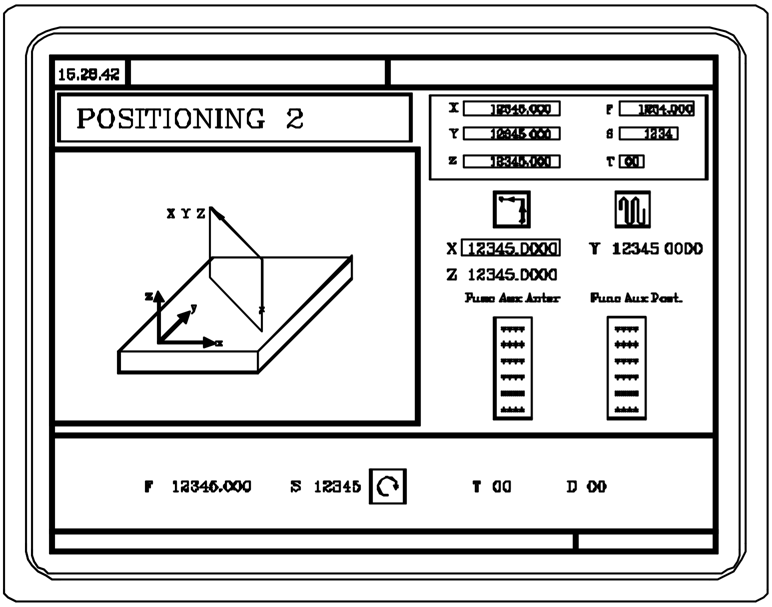
To select the positioning cycle, press 
This cycle may be defined in two different ways:

Level 1.




One must define — The target point (X, Y, Z), the axes movement sequence and the type of feedrate

Level 2.




One must define — The target point (X, Y, Z), the axes moving sequence, the type of feedrate and the auxiliary functions "M" to be executed before and after the movement.

## 3.8.1 DATA DEFINITION

### *Coordinates of the target point*

These coordinates are defined one by one. After placing the cursor over the axis coordinates, on can:

a) Enter the value by hand. Key in the desired value and press

b) Assign the current axis position.

Jog the axis to the desired point with the handwheel or the JOG keys. 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

### *Axes moving sequence*

All three axes at the same time

First the Z axis and then in the plane (X and Y axis at the same time)

Firs in the plane (X and Y at the same time) and then the Z axis

To select the moving sequence, place the cursor over this icon and press

### *Type of axis feedrate*

Programmed feedrate

Rapid feed

To select the type of feedrate, place the cursor over this icon and press

### *Auxiliary "M" functions*

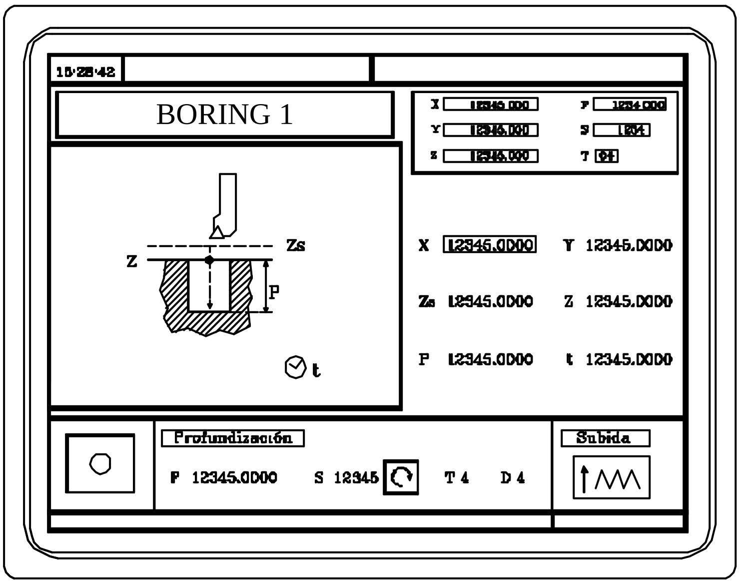
Select the corresponding window with

Use to move around inside the window
The functions will be executed in the same order as inserted in the list.
To delete a function, select it and press

## 3.9 BORING OPERATION

To select the boring operation, press 

Boring operation (Level 1)

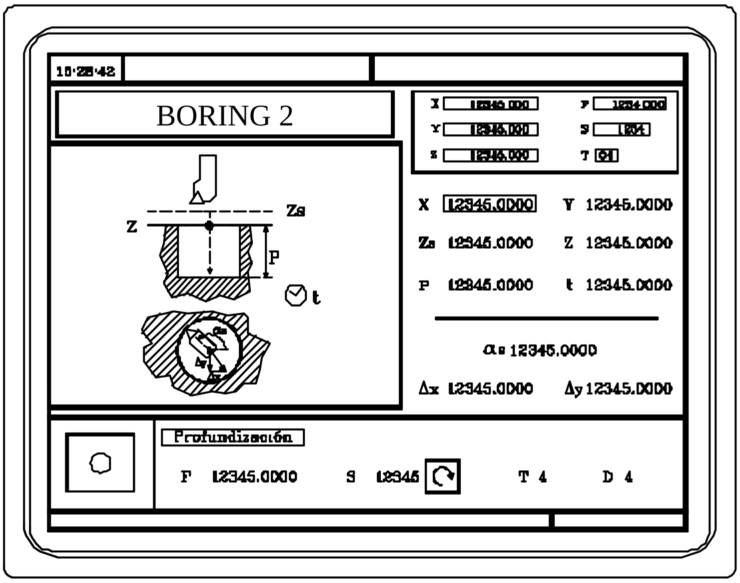


One must define The machining point (X, Y), the coordinate of the safety plane (Zs), the coordinate of the part surface (Z), The total machining depth (P) and the dwell at the bottom (t)

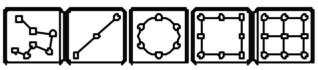
Boring operation (level 2)

Available when working with spindle orientation.

After penetrating the quill, it is possible to orient the spindle and retract the quill before the exit movement, thus avoiding scratching the part.



One must define The machining point (X, Y), the coordinate of the safety plane (Zs), the coordinate of the part surface (Z), The total machining depth (P), the dwell at the bottom (t) and the orientation and quill retraction data (αs, Δx, Δy)

The Boring operation may be carried out in the indicated position (X,Y) or a positioning may be associated with it by means of the  keys as described later on.

## 3.9.1 DATA DEFINITION

***Coordinates of the machining point***

These coordinates are defined one by one. After placing the cursor over the axis coordinates, one can:

a) Enter the value by hand. Key in the desired value and press ENTER

b) Assign the current axis position.

Jog the axis to the desired point with the handwheel or the JOG keys. The upper right-hand window shows the tool position at all times.

Press RECALL for the selected data to assume the value appearing in the upper right-hand window.

Press ENTER

***Machining conditions in Z (Zs, Z, P, t)***

The machining conditions are defined one by one.

The Zs and Z values are defined like those of the machining point.

To define the rest of the values (P, t), place the cursor in the corresponding window, key in the desired value and press ENTER

***Type of exit at level 1***

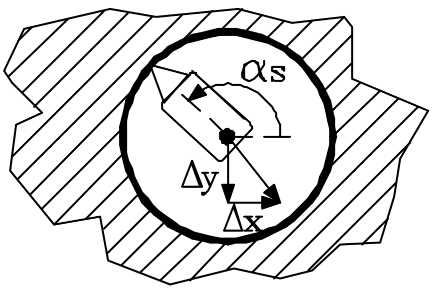
The exit movement may be carried out in two ways:

At machining feedrate (G01) and with the spindle turning

In rapid (G00) and with the spindle stopped

To select the type of exit, place the cursor over this icon and press

***Data for orienting and retracting the quill (as, Dx, Dy)***



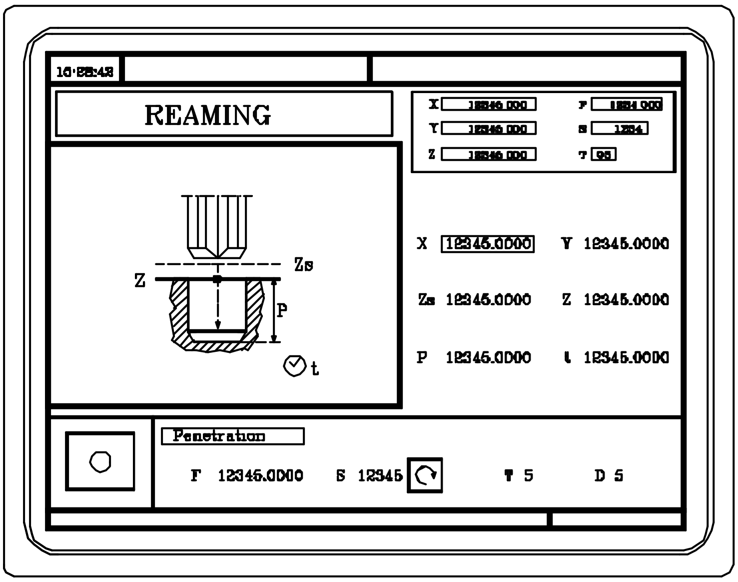

They are defined one by one.

Go to the relevant window, key in the desired value and press ENTER

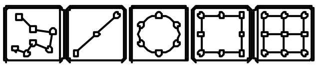
The quill first orients, it then retracts in XY and finally goes up in rapid (G00) with the spindle stopped.

## 3.10 REAMING OPERATION

To select the Reaming operation, press 



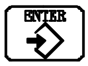

One must define: The machining point (X, Y), the coordinate of the safety plane (Zs), the coordinate of the part surface (Z), the total machining depth (P) and the dwell at the bottom (t)

The Reaming operation can be carried out in the indicated position (X,Y) or a positioning may be associated with it by means of the  keys as described later on.

### 3.10.1 DATA DEFINITION

***Coordinates of the machining point***

These coordinates are defined one by one. After placing the cursor over the axis coordinates, one can:

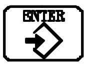
a) Enter the value by hand. Key in the desired value and press 


b) Assign the current axis position.

Jog the axis to the desired point with the handwheel or the JOG keys. The upper right-hand window shows the tool position at all times.

Press 

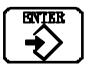
for the selected data to assume the value appearing in the upper right-hand window.

Press 


***Machining conditions in Z (Zs, Z, P, t)***

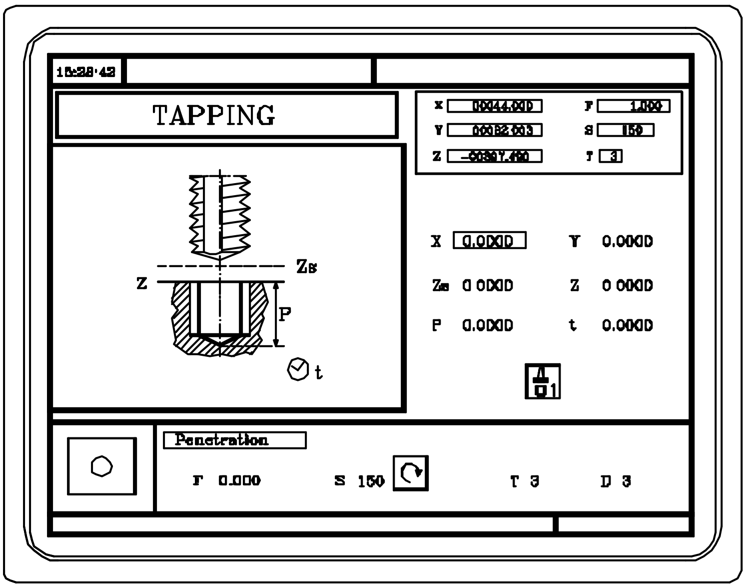
The machining conditions are defined one by one.

The Zs and Z values are defined like those of the machining point.

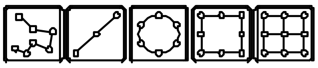
To define the rest of the values (P, t), place the cursor in the corresponding window, key in the desired value and press 

## 3.11 TAPPING OPERATION

To select the tapping operation, press 




One must define — The machining point (X, Y), the coordinate of the safety plane (Zs), the coordinate of the part surface (Z), the total machining depth (P) and the dwell at the bottom (t) and the type of Tapping.

The Tapping operation can be carried out in the indicated position (X,Y) or a positioning may be associated with it by means of the  keys as described later on.

## *3.11.1 DATA DEFINITION*

***Coordinates of the machining point***

These coordinates are defined one by one. After placing the cursor over the axis coordinates, one can:

a) Enter the value by hand. Key in the desired value and press ENTER

b) Assign the current axis position.

Jog the axis to the desired point with the handwheel or the JOG keys. The upper right-hand window shows the tool position at all times.

Press RECALL for the selected data to assume the value appearing in the upper right-hand window.

Press ENTER

***Machining conditions in Z (Zs, Z, P, t)***

The machining conditions are defined one by one.

The Zs and Z values are defined like those of the machining point.

To define the rest of the values (P, t), place the cursor in the corresponding window, key in the desired value and press ENTER

***Type of tapping***

Rigid tapping ...................Without clutch

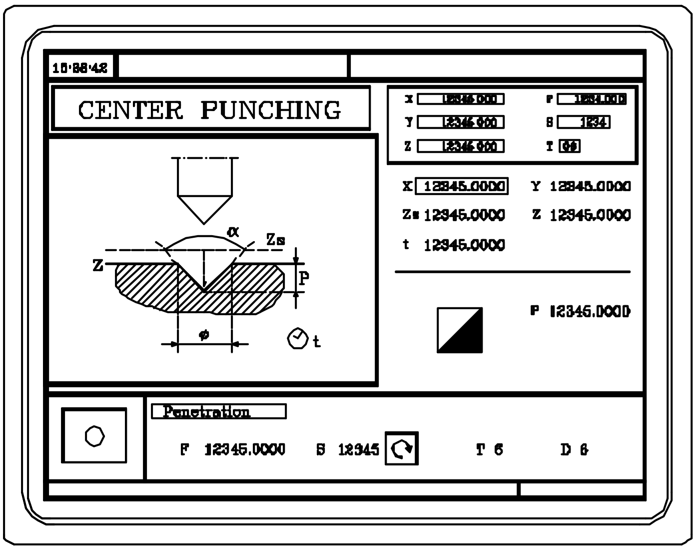
Regular tapping ...............With a clutch

To select the type of tapping, place the cursor over this icon and press

## 3.12 DRILLING AND CENTER PUNCHING OPERATIONS

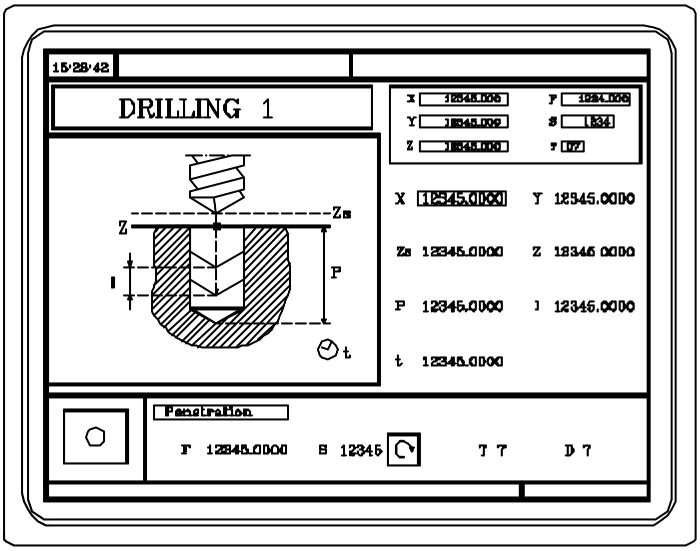
To select the Drilling and Center Punching operations, press

Center punching operation




One must define — The punch point (X, Y), the coordinate of the safety plane (Zs), the coordinate of the part surface (Z), the total machining depth (P) and the dwell at the bottom (t) and the type of center punching.

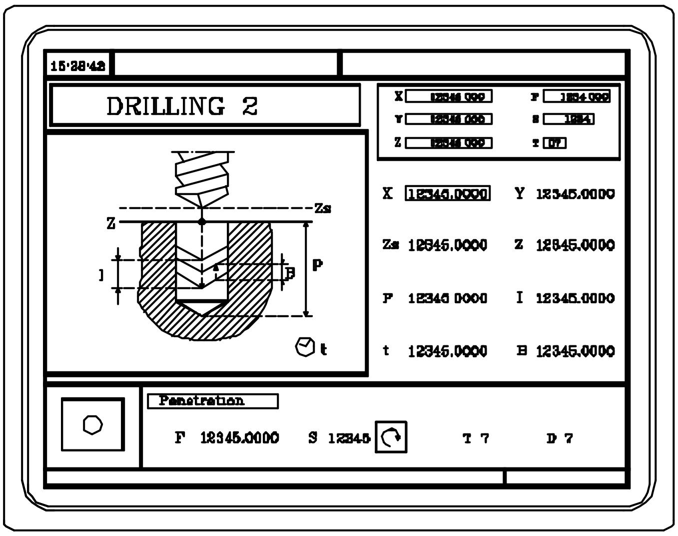
Drilling operation. Level 1




One must define — The drilling point (X, Y), the coordinate of the safety plane (Zs), the coordinate of the part surface (Z), the total machining depth (P), the drilling peck and the dwell at the bottom (t).

Drilling operation. (Level 2)

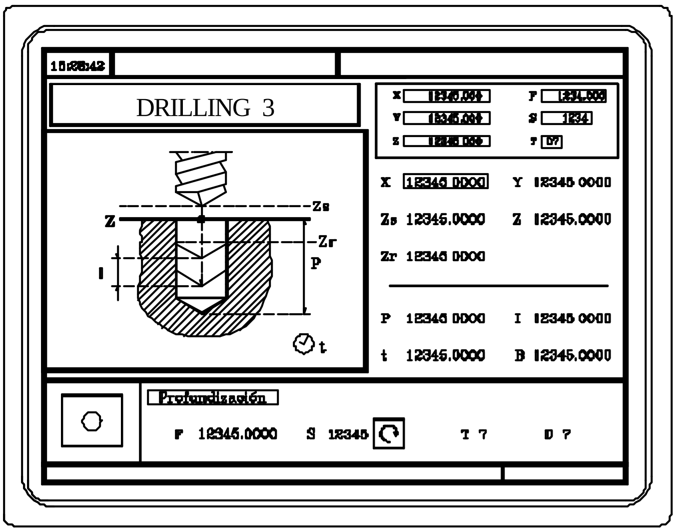
It is possible to set the withdrawal distance (B) after each penetration.



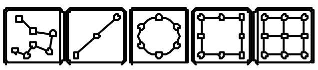

One must define The machining point (X, Y), the coordinate of the safety plane (Zs), the coordinate of the part surface (Z), the total machining depth (P), the drilling peck (I), the dwell at the bottom (t) and the withdrawal distance after each penetration (B)

Drilling operation. (Level 3)

It is possible to set the withdrawal position (Zr) after each penetration.




One must define The machining point (X, Y), the coordinate of the safety plane (Zs), the coordinate of the part surface (Z), the withdrawal position (Zr) the total machining depth (P), the drilling peck (I), the dwell at the bottom (t).

The Center Punching and Drilling operations may be carried out in the indicated position (X,Y) or may be associated with a positioning using the  keys as described later on.

## *3.12.1 DATA DEFINITION*

***Coordinates of the machining point***

These coordinates are defined one by one. After placing the cursor over the axis coordinates, one can:

a) Enter the value by hand. Key in the desired value and press

b) Assign the current axis position.

Jog the axis to the desired point with the handwheel or the JOG keys. 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

***Machining conditions in Z (Zs, Z, Zr, P, I, t, B)***

The machining conditions are defined one by one.

The Zs and Z values are defined like the coordinates of the machining point.

To define the rest of the values (Zr, P, I, t, B), place the cursor in the corresponding window, key in the desired value and press

One must define — The machining point (X, Y), the machining conditions in Z (Zs, Z, t) and the type of center punching.

***Type of center punching***

The type of center punching may be defined in two ways:

a) By defining the total machining depth (P)
b) By defining the punch angle ($\alpha$) and the diameter of the point ($\Phi$)

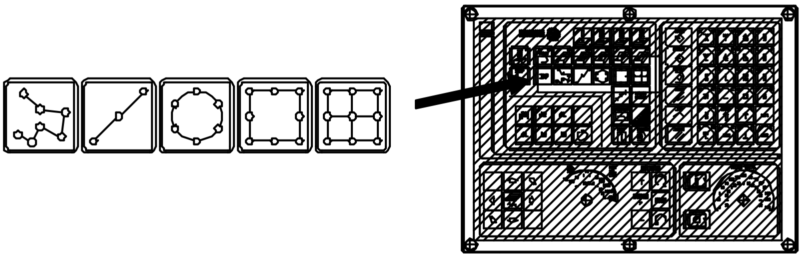
To select the type of center punching, place the cursor over this icon and press

To define the "P, $\alpha$, $\Phi$" values, place the cursor in the corresponding window, key in the desired value and press

## 3.13 MULTIPLE POSITIONING

With this CNC, it is possible to associate multiple positioning with Boring, Reaming, Tapping, Drilling and Center Punching operations.

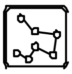
The following keys must be used to select this feature.

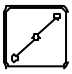


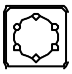
When pressing one of these keys, the CNC selects the corresponding type of positioning and it changes the display.

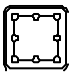
It keeps the lamp ON of the key corresponding to the selected operation (Boring, Reaming, etc.) and the bottom of the screen shows the data for that operation.

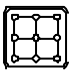
The types of multiple positioning that can be selected with each key are

 At random points

 In a straight line

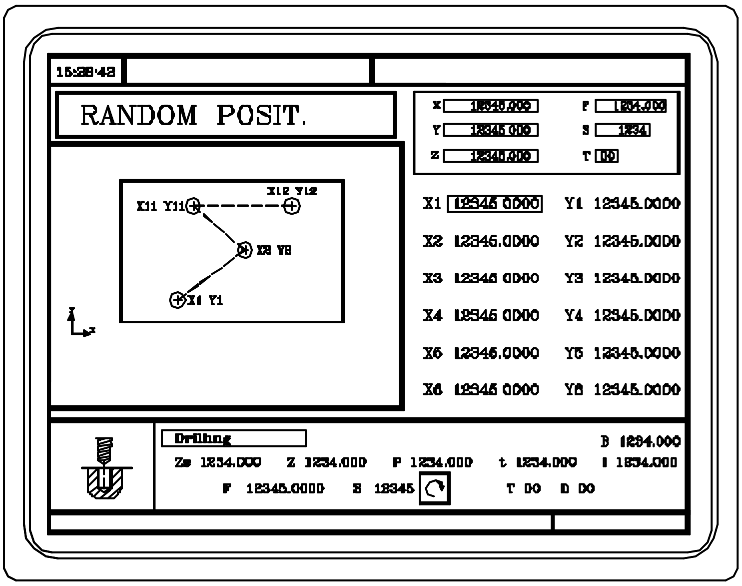
 In a bolt-hole pattern

 In a parallelogram pattern

 In a grid pattern

## 3.13.1 MULTIPLE POSITIONING AT RANDOM POINTS

To associate this positioning with an operation, press 

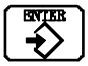



Up to 12 points can be defined. Coordinates (X1, Y1) .... (X12, Y12)

When not using all 12 points, the first unused point must be defined with the same coordinates as those of the last point.

### *Point definition*

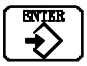
The coordinates are defined one by one. After placing the cursor over the axis coordinates, one can:

a) Enter the value by hand. Key in the desired value and press 

b) Assign the current axis position.

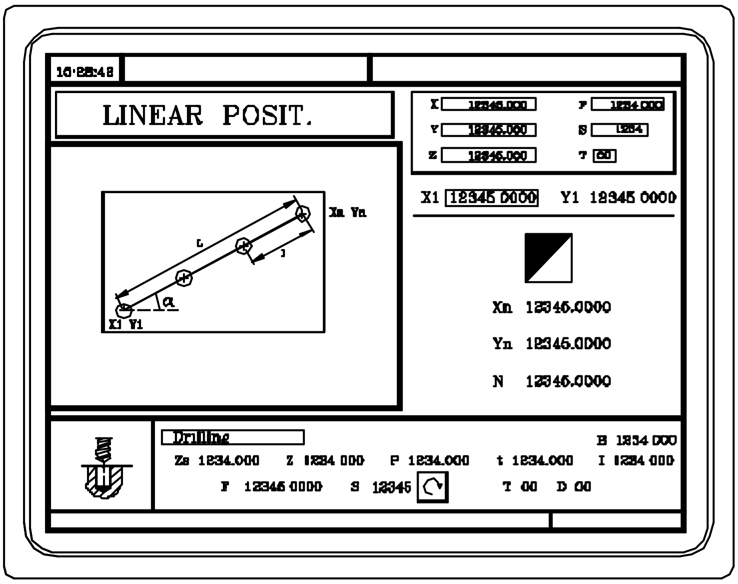
Jog the axis to the desired point with the handwheel or the JOG keys. The upper right-hand window shows the tool position at all times.

Press  for the selected data to assume the value shown in the upper right-hand window.

Press 

## *3.13.2 MULTIPLE POSITIONING IN A STRAIGHT LINE*

To associate this positioning with an operation, press




This may be defined in 5 different ways:

1) The coordinates of the first point ........................................ (X1, Y1)
The coordinates of the last point ........................................ (Xn, Yn)
Distance between points ........................................................... (I)

2) The coordinates of the first point ........................................ (X1, Y1)
The coordinates of the last point ........................................ (Xn, Yn)
Number of points ........................................................................(N)

3) The coordinates of the first point ........................................ (X1, Y1)
The inclination angle ................................................................ ($\alpha$)
Total distance from first to last point .......................................... (L)
Distance between points ........................................................... (I)

4) The coordinates of the first point ........................................ (X1, Y1)
The inclination angle ................................................................ ($\alpha$)
Total distance from first to last point .......................................... (L)
Number of points ........................................................................(N)

5) The coordinates of the first point ........................................ (X1, Y1)
The inclination angle ................................................................ ($\alpha$)
Number of points ........................................................................(N)
Distance between points ........................................................... (I)

To select the desired one, place the cursor over the icon and press

***Point definition***

The coordinates are defined one by one. After placing the cursor over the axis coordinates, one can:

a) Enter the value by hand. Key in the desired value and press ENTER

b) Assign the current axis position.

Jog the axis to the desired point with the handwheel or the JOG keys. The upper right-hand window shows the tool position at all times.

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

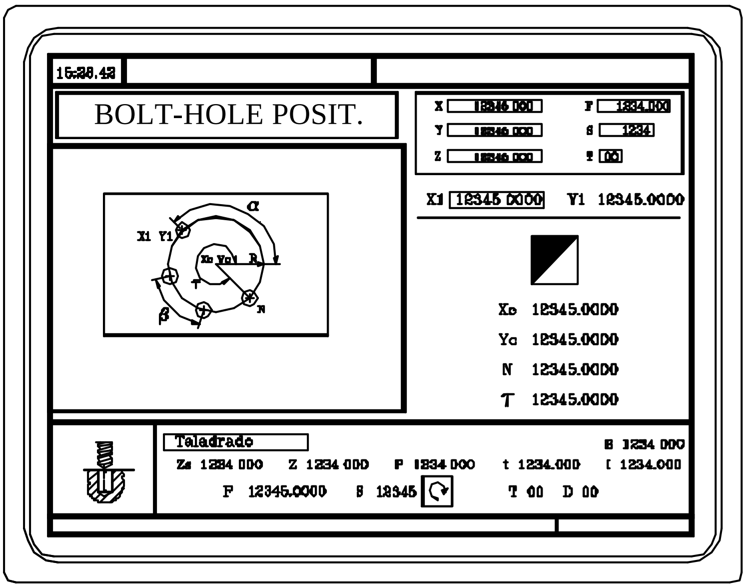
Press ENTER

To define the rest of the values (I, N, $\alpha$, L), place the cursor in the corresponding window, key in the desired value and press ENTER

## *3.13.3 MULTIPLE POSITIONING IN AN ARC (BOLT-HOLE PATTERN)*

To associate this positioning with an operation, press

Multiple bolt-hole pattern positioning (Level 1)




This could be defined in 6 different ways:

1) The coordinates of the first point ........ (X1, Y1)
   The center coordinates ........ (Xc, Yc)
   The angle of the last point ........ $(\tau)$
   The angular distance between points ........ $(\beta)$

2) The coordinates of the first point ........ (X1, Y1)
   The radius ........ (R)
   The angle of the first point ........ $(\alpha)$
   The angle of the last point ........ $(\tau)$
   The angular distance between points ........ $(\beta)$

3) The coordinates of the first point ........ (X1, Y1)
   The center coordinates ........ (Xc, Yc)
   The number of points ........ (N)
   The angle of the last point ........ $(\tau)$

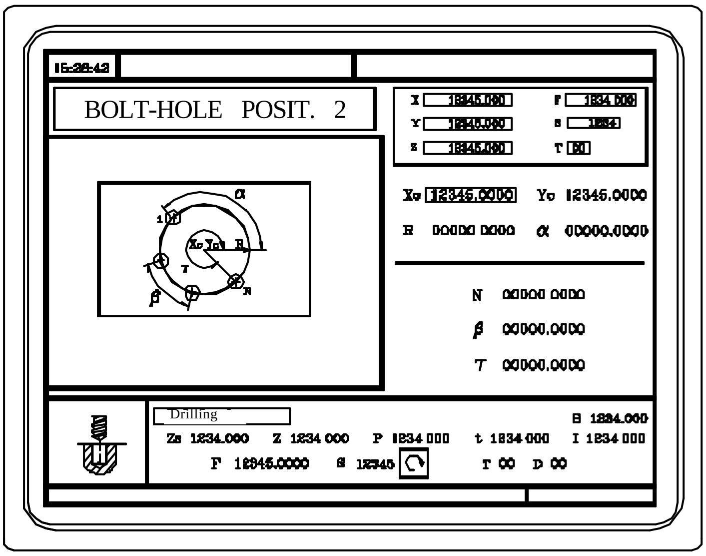
4) The coordinates of the first point ........ (X1, Y1)
   The center coordinates ........ (Xc, Yc)
   The number of points ........ (N)
   The angular distance between points ........ $(\beta)$

5) The coordinates of the first point ........ (X1, Y1)
   The radius ........ (R)
   The angle of the first point ........ $(\alpha)$
   The number of points ........ (N)
   The angle of the last point ........ $(\tau)$

6) The coordinates of the first point ........ (X1, Y1)
   The radius ........ (R)
   The angle of the first point ........ $(\alpha)$
   The number of points ........ (N)
   The angular distance between points ........ $(\beta)$

To select the desired one, place the cursor over the icon and press

Multiple bolt-hole pattern positioning (level 2)




One must define:

The center coordinates ........................................................(Xc, Yc)
The starting point in polar coordinates:......... Radius (R) and angle (α)

2 of the following data must be defined.
When all 3 are defined (if they are other than 0) the cycle assumes (N) and (β)

The number of points to machine ................................................ (N)
Angular distance between points .................................................. (β)
The angle of the end point .......................................................... (τ)

***Data definition***

The coordinates are defined one by one. After placing the cursor over the axis coordinates, one can:

a) Enter the value by hand. Key in the desired value and press ENTER

b) Assign the current axis position.

Jog the axis to the desired point with the handwheel or the JOG keys. The upper right-hand window shows the tool position at all times.

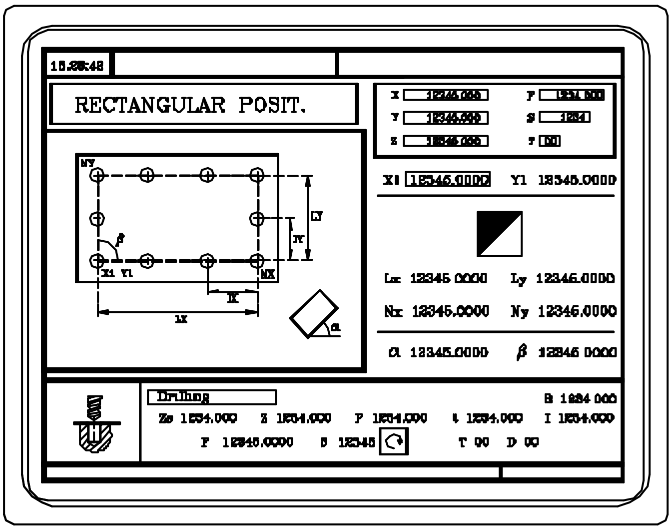
Press RECALL for the selected data to assume the value shown in the upper right-hand window.

Press ENTER

To define the rest of the values (R, N, α, β, τ), place the cursor in the corresponding window, key in the desired value and press ENTER

## *3.13.4 MULTIPLE POSITIONING IN A PARALLELOGRAM PATTERN*

To associate this positioning with an operation, press




This could be defined in 3 different ways:

1) The coordinates of the first point .......................................... (X1, Y1)
The lengths in X, Y ............................................................ (Lx, Ly)
The X and Y distances between points .................................. (Ix, Iy)
The rotation angle ........................................................................ (α)
The angle between the sides ....................................................... (β)

2) The coordinates of the first point .......................................... (X1, Y1)
The lengths in X and Y ....................................................... (Lx, Ly)
The number of points in X and Y ......................................... (Nx, Ny)
The rotation angle ........................................................................ (α)
The angle between the sides ....................................................... (β)

3) The coordinates of the first point .......................................... (X1, Y1)
The X and Y distances between points .................................. (Ix, Iy)
The number of points in X and Y ......................................... (Nx, Ny)
The rotation angle ........................................................................ (α)
The rotation angle ........................................................................ (β)

To select the desired one, place the cursor over the icon and press

### ***Point definition***

The coordinates are defined one by one. After placing the cursor over the axis coordinates, one can:

a) Enter the value by hand. Key in the desired value and press ENTER

b) Assign the current axis position.

Jog the axis to the desired point with the handwheel or the JOG keys. The upper right-hand window shows the tool position at all times.

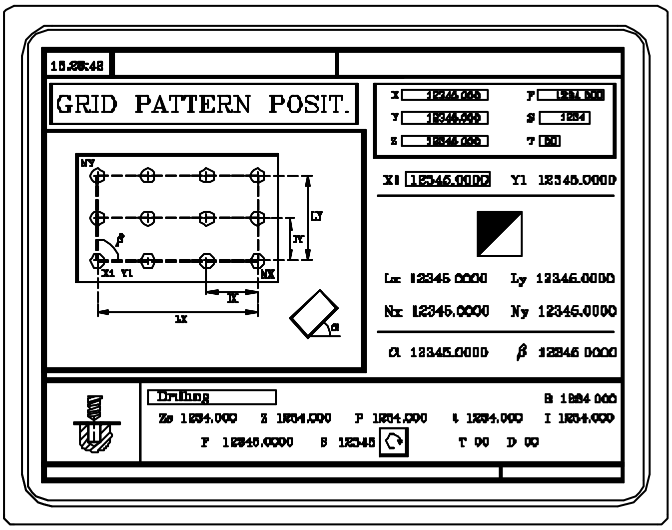
Press RECALL for the selected data to assume the value shown in the upper right-hand window.

Press ENTER

To define the rest of the values (Lx, Ly, Ix, Iy, α, β, Nx, Ny), place the cursor in the corresponding window, key in the desired value and press ENTER

## 3.13.5 MULTIPLE POSITIONING IN A GRID PATTERN

To associate this positioning with an operation, press




This could be defined in 3 different ways:

1) The coordinates of the first point .......................................(X1, Y1)
The lengths in X, Y ........................................................... (Lx, Ly)
The X and Y distances between points .................................. (Ix, Iy)
The rotation angle ....................................................................(α)
The angle between the sides ...................................................... (β)

2) The coordinates of the first point .......................................(X1, Y1)
The lengths in X and Y ...................................................... (Lx, Ly)
The number of points in X and Y ...................................... (Nx, Ny)
The rotation angle ....................................................................(α)
The angle between the sides ...................................................... (β)

3) The coordinates of the first point .......................................(X1, Y1)
The X and Y distances between points .................................. (Ix, Iy)
The number of points in X and Y ...................................... (Nx, Ny)
The rotation angle ....................................................................(α)
The rotation angle ..................................................................... (β)

To select the desired one, place the cursor over the icon and press

***Point definition***

The coordinates are defined one by one. After placing the cursor over the axis coordinates, one can:

a) Enter the value by hand. Key in the desired value and press

b) Assign the current axis position.

Jog the axis to the desired point with the handwheel or the JOG keys. The upper right-hand window shows the tool position at all times.

Press for the selected data to assume the value shown in the upper right-hand window.

Press

To define the rest of the values (Lx, Ly, Ix, Iy, α, β, Nx, Ny), place the cursor in the corresponding window, key in the desired value and press

# 4. STORAGE OF PROGRAMS

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 chapter on "working with operations or cycles".

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

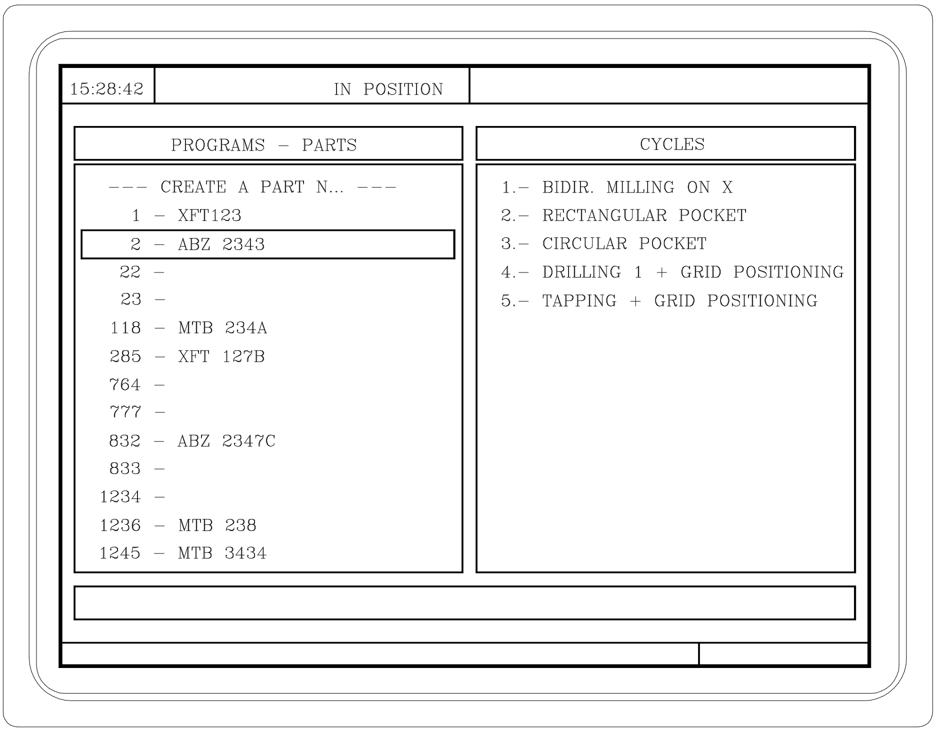
Modify a part-program.................................... Erase an operation
Move an operation to another position
Add or insert to a new operation
Modify an already existing operation

## *4.1 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 ISO-coded 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:

ESC

the key for an operation

or ISO

## 4.2 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 [↓]

If the part-program is formed on an MC mode cycle basis, the right -hand column will display the cycles which said part consists of:

If you press [ENTER] or [←] or [→] the pointer goes 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

### 4.2.1 SEEING THE OPERATIONS IN DETAIL

After selecting the operation required, with the pointer, press [RECALL]

The CNC will display all the data for said operation.

Now you can:

Simulate the operation. (See the chapter on "Execution and Simulation").
Execute the operation. (See the chapter on "Execution and Simulation").
Modify the operation
Store the operation. Replace the previous one or including this as a new one.

## 4.3 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.

### 4.3.1 STORAGE OF AN OPERATION OR CYCLES

A block or cycle can be added at the end of the program, after the last operation, or inserted between 2 existing operations.

To store the block or cycle, follow these steps:

* Define the desired block or cycle, assigning the relevant data to it.
* 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 operation is to be stored and press [ENTER]

Example:

| You have | You want |
|---|---|
| 1.- Bidirectional surface milling in X | 1.- Bidirectional surface milling in X |
| 2.- Rectangular Pocket | 2.- Rectangular Pocket |
| 3.- Circular Pocket | 3.- Circular Pocket |
| | ***4.- Drilling 1 + Positioning in Line*** |
| 4.- Drill 1 + Grid pattern positioning | 5.- Drill 1 + Grid pattern positioning |
| 5.- Tapping + Grid pattern positioning | 6.- Tapping + Grid pattern positioning |
| | ***7.- Profile 1*** |

***4.- Drilling 1 + Positioning in Line*** Once the operation is defined, place the cursor over "operation 3.- Circular Pocket" and press [ENTER].

***7.- Profile 1*** Once the operation is defined, place the cursor over "5.- Tapping + Grid pattern positioning" and press [ENTER].

## *4.4 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.

## *4.5 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.

## *4.6 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.

### *4.6.1 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.

### *4.6.2 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 | Your want |
|---|---|
| 1.- Bidirectional Surface Milling in X | 1.- Bidirectional Surface Milling in X |
| 2.- Rectangular Pocket | 2.- Circular Pocket |
| 3.- Circular Pocket | 3.- Drilling 1 + Positioning in line |
| 4.- Drilling 1 + Positioning in line | 4.- Drilling 1 + Positioning in Grid |
| 5.- Drilling 1 + Positioning in Grid | 5.- Tapping + Positioning in Grid |
| 6.- Tapping + Positioning in Grid | 6.- Rectangular Pocket |
| 7.- Profile 1 | 7.- Profile 1 |

Select the "Rectangular Pocket" and press

Move the cursor onto the "Tapping + Positioning in Grid" and press ENTER

### *4.6.3 ADDING OR INSERTING A NEW OPERATION*

To add or insert an operation take the same steps as to store an operation.

* Define the operation or cycle required, assigning this the relevant data.
* Press P.PROG to access the list of part-programs stored.
* Move over the operation after which the operation is to be stored and press ENTER

### *4.6.4 MODIFYING AN ALREADY EXISTING OPERATION*

To modify an operation take the following steps:

* Use the pointer to select, in the right-hand column, the operation required for modification.
* 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 keys ↑ ↓. 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 "Rectangular Pocket" operation is modified

| You have | "Replace" option | "Insert" option |
|---|---|---|
| ***1.- Rectangular Pocket*** | ***1.- Rectangular Pocket*** | ***1.- Rectangular Pocket*** |
| 2.- Circular Pocket | 2.- Circular Pocket | ***2.- Rectangular Pocket*** |
| | | 3.- Circular Pocket |

Note: One can select an existing operation, modify this and then insert this somewhere else and even in another part-program.

# 5. 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:

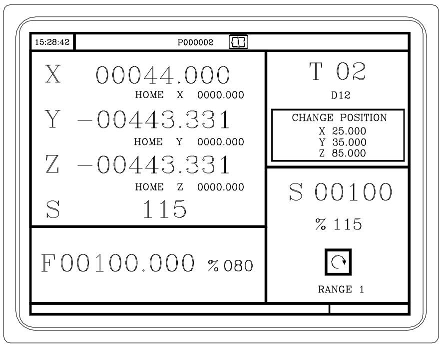
The CNC allows a part-program or any operation to be executed or simulated. This simulation or execution can be done from beginning to end or alternatively press key SINGLE for this to be executed or simulated step by step.

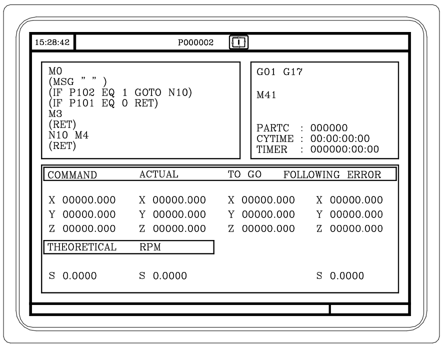
The CNC enables execution or simulation of:

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.



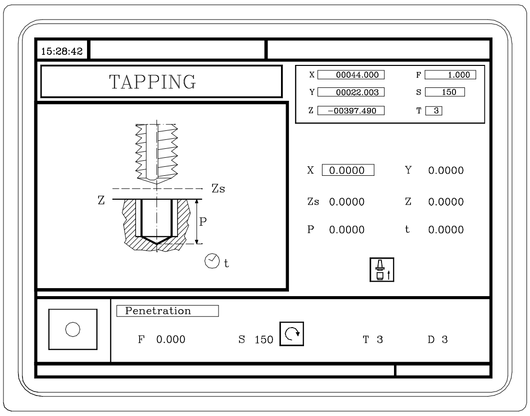

It then acts as follows:

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

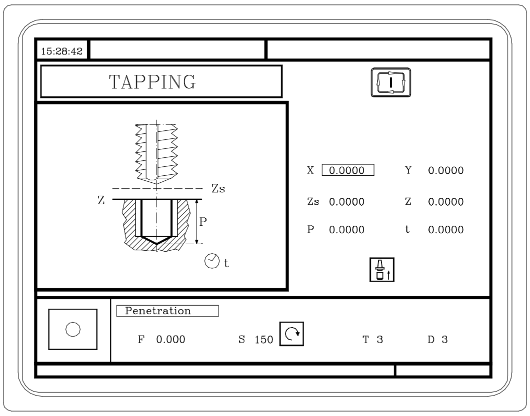
if CLEAR is pressed, the part-program is de-selected and the CNC deletes it from the top center window.

## *5.1 SIMULATING OR EXECUTING AN OPERATION OR CYCLE*

All the operations or cycles have 2 operating modes: Execution mode and Edition Mode




Editing mode




Execution Mode

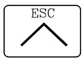
**Simulation**

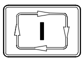
The operation or cycle can be simulated in both operating modes. To do this, press 

The CNC will display the graphic representation page for the M model.

**Execution**

An operation or cycle can only be executed in the cycle execution mode.
The operation or cycle cannot be executed when the cycle editing mode is selected.

To exit the edition mode and go on to execution mode press 


To execute an operation or cycle, press 

## *5.2 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

## *5.2.1 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 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.

## *5.3 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.

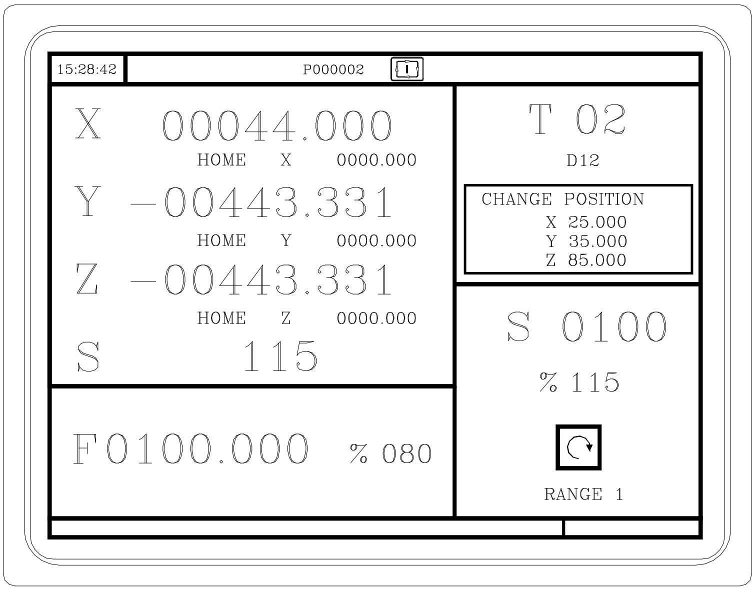
* Select the program which contains this from the left-hand column and the operation required to be simulated or executed from the right-hand column.

* Press RECALL

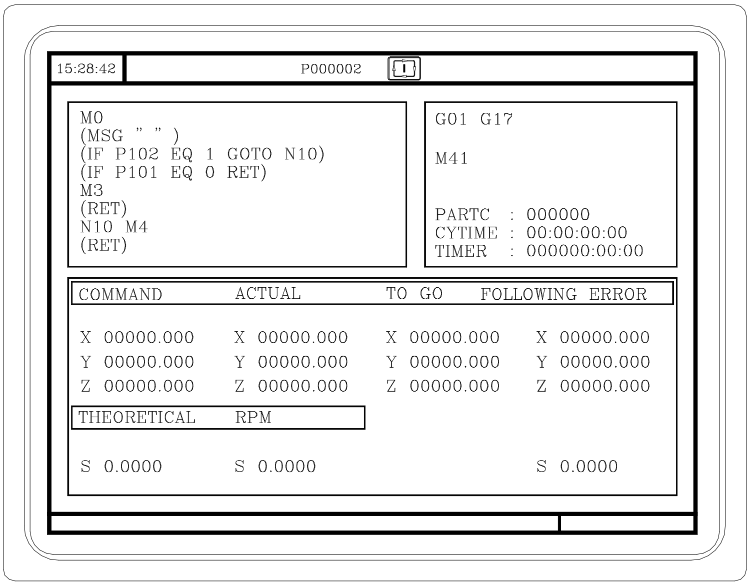
To simulate the operation press GRAPHICS and to execute this press

## *5.4 EXECUTION MODE*

When you press to execute an operation or part-program, the CNC displays the standard MC operating mode screen.




If you press the CNC displays the special MC 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 GRAPHICS 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.

## 5.4.1 TOOL INSPECTION

The PLC mark M5050, general CNC logic input "TOOLINSP", determines when tool inspection is enabled.

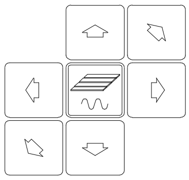
TOOLINSP=0 Tool inspection is possible after pressing

TOOLINSP=1 When pressing program execution is interrupted.

Once program execution is interrupted, press to move the axes and proceed with tool inspection.

Once Tool Inspection has been selected, it is possible to:

***Move the axes to the tool change position***

Move them using the handwheels or the  keys

*Select another tool*

To be able to make a tool change the standard MC operating mode screen must be selected.

Press . The CNC will frame the tool number.

Key the tool number required for selection and press for the CNC to select the new tool.

The CNC will process the tool change.

***Modify tool values (dimensions and geometry)***

Press The CNC displays the Tool Calibration screen.

It is possible to change the tool dimensions (offsets I,K to compensate 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

***Resume program execution.***

To resume program execution, press

The CNC will reposition the tool moving it to the point where tool inspection started. Two cases are possible:

1.- Only one axis has been moved.
The CNC repositions it and resumes execution.

2.- Two or more axes have been moved.
The CNC shows a window with the following options to choose the repositioning order (sequence) of the axes.

| | |
|---|---|
| PLANE | The axes forming the main plane (X-Y) move at the same time. |
| Y-X | When moving the main plane axes, the Y axis moves first and then the X axis. |
| X-Y | When moving the main plane axes, the X axis moves first and then the Y axis. |
| Z | Move the Z axis. |

For example, to move the Z axis first, then the Y and finally the X: [Z] [Y-X]

## 5.5 GRAPHIC REPRESENTATION

When you press [GRAPHICS] the CNC displays the M mode graphic representation page.

To leave the graphic representation mode press [GRAPHICS] or [ESC]

In the Operation Manual, M -MC models, 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» or «Solid X-Z»

The «X-Z» graphic is a line graphic which uses colored lines to describe tool tip movement.

The «Solid X-Z» graph starts 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 [ENTER]

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 [STOP]

**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 the [←] [→] [↑] [↓] keys to increase or reduce its size use "+" "-", and for the CNC to assume these values press [ENTER]

Each time a new display zone is selected the CNC keeps the present graphic representation. It does not erase this.

When you press [START] 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, stop execution or simulation by pressing [STOP]

## 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 → ←, for the CNC to assume said value, press ENTER

*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 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 ENTER

*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 ENTER

## 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 GRAPHICS or ESC


# Self-teaching Manual
# (MC option)

Ref. 0112-ing

FAGOR

# INDEX

Chapter 4 Automatic operations



Chapter 5 Summary of work cycles



Capítulo 6 Conversational part-programs

Chapter 1

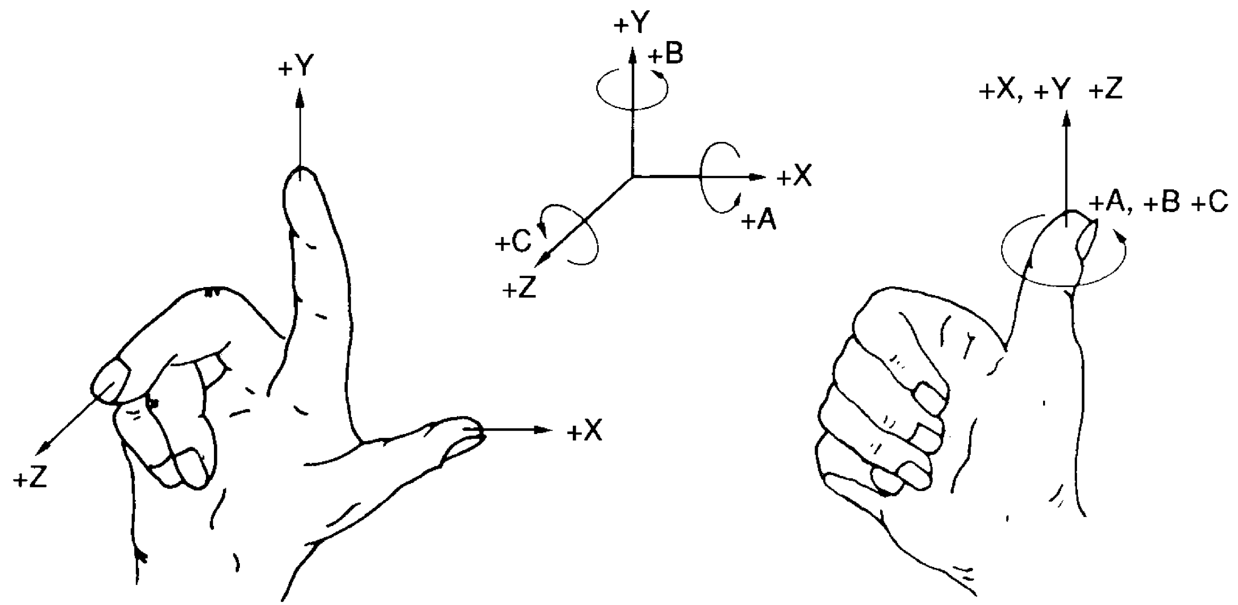
# Theory on CNC machines

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 work units.
  - > Programming units
  - > Spindle speed.
  - > Axis feedrate.

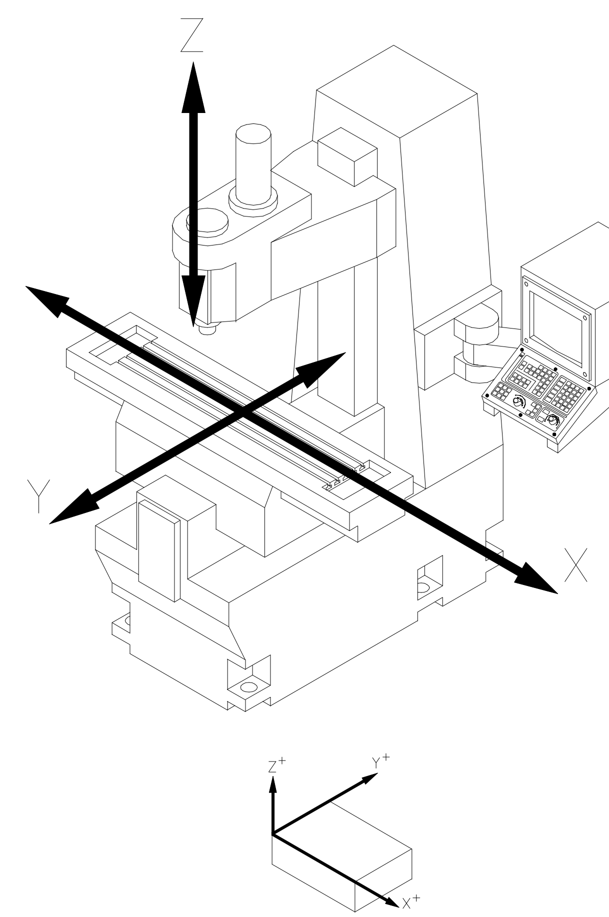
# 1.1 Machine Axes.

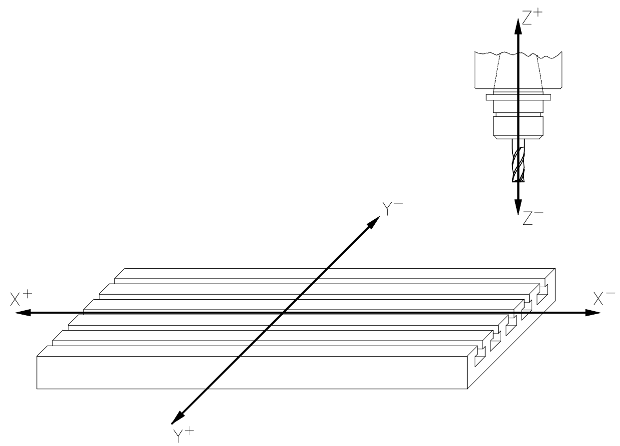
The orientation of the axes depends on the type of machine and are established by the “rule of the right hand”.



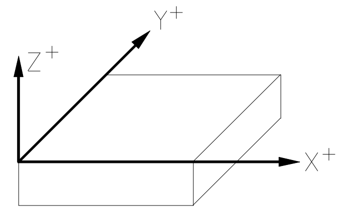

Axes orientation. Rotary axes.

This manual uses the following axes configuration.



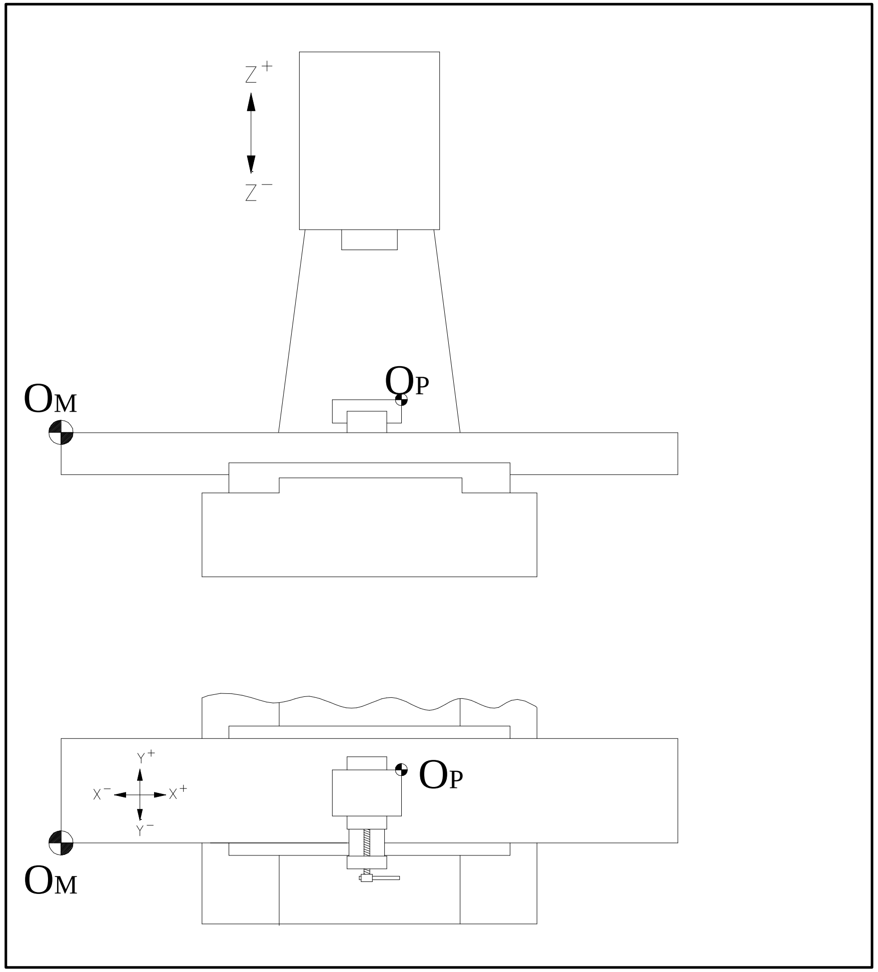




Two types of movements can be distinguished on a machine, those of the machine (X, Y) and that of the tool itself (Z). But for programming them, let us assume the movements of the tool with respect to the machine. Thus, the axes would be:



# 1.2 Machine reference zero and part zero.

They are the references the machine needs in order to work:




<u>Machine ref. zero ($O_M$)</u>

It is set by the manufacturer and it is the origin point for the axes.

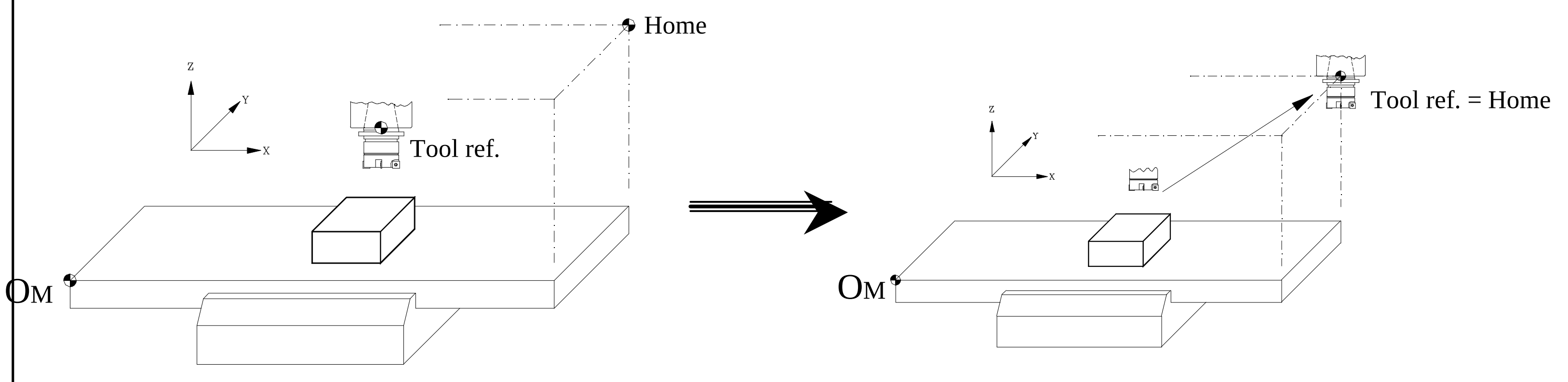
<u>Part zero ($O_P$)</u>

It 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.



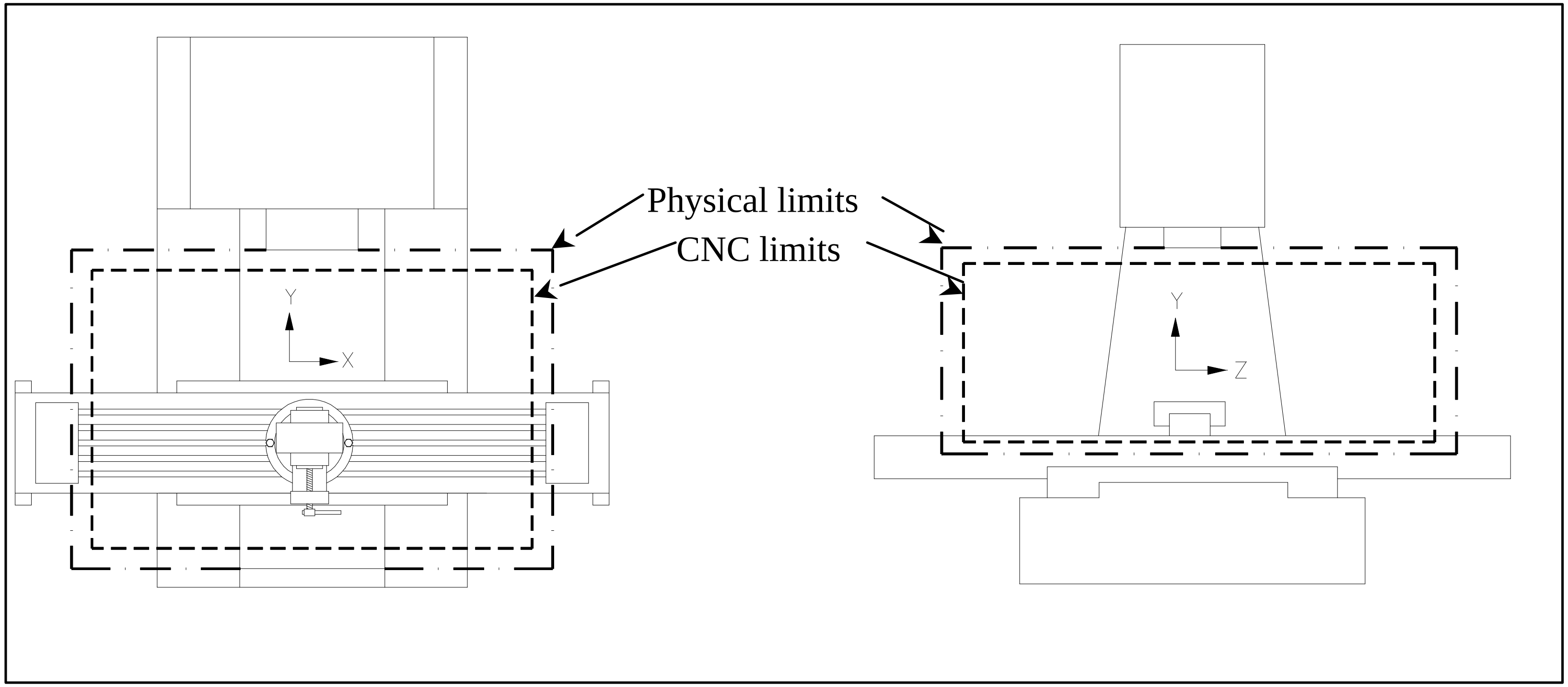

Machine Ref. point: Is the physical reference mark location to where the axes move during “home search”.

NOTE: With the new feedback systems (with distance coded reference marks), it is possible to know the position of the axes by moving them a short distance. This way, the concept of Machine Reference (home) no longer applies.

# 1.4 Travel limits.

There are two types of limits:

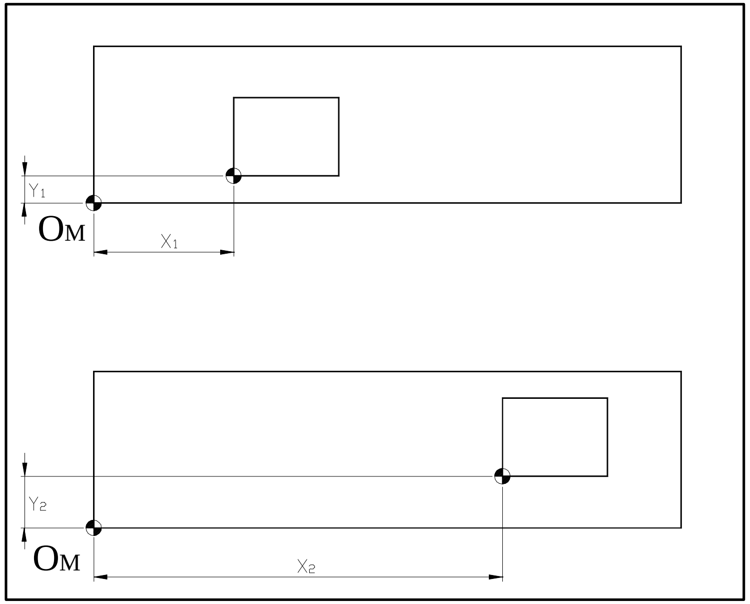
- Hard limits: Mechanical limits set on the machine to prevent the carriage from moving beyond the ways (cams and hardstops).
- CNC limits: Set at the CNC by the manufacturer to prevent the carriage from running into the machine's hard limits.



# 1.5 Part zero setting.

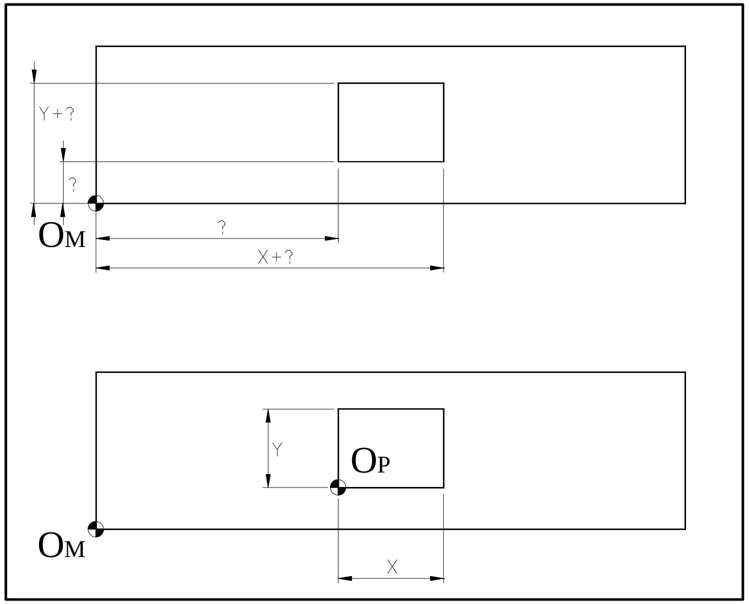
The Part zero is set on all three axes.

When machining several parts, the distance from Machine ref. zero ($O_M$) to the part is different for each one. A different program would be needed for each part. By programming from Part zero ($O_P$), it would be independent of the actual location of the part.




Programming gets complicated when done from Machine ref. zero ($O_M$) and it is only good for that part in that particular location.

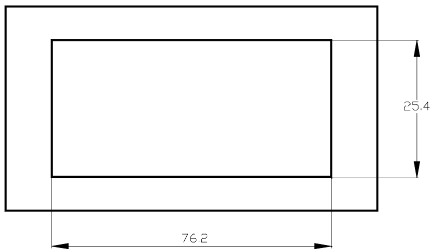
By programming from Part zero ($O_P$), the part dimensions may be taken straight from the blueprint.



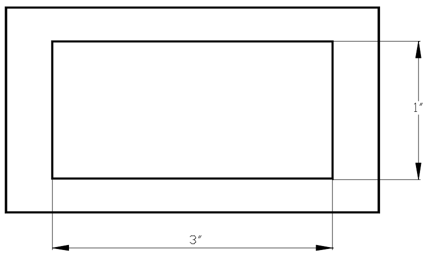
# 1.6 Work units.

Programming units

They are set by the manufacturer and may be in millimeters or inches.



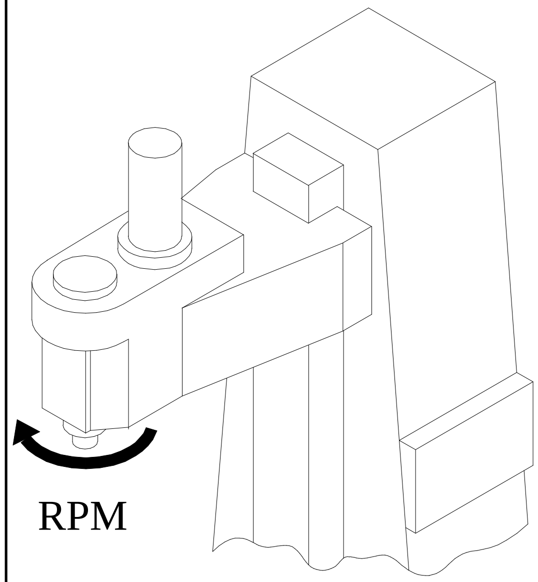

millimeters




inches

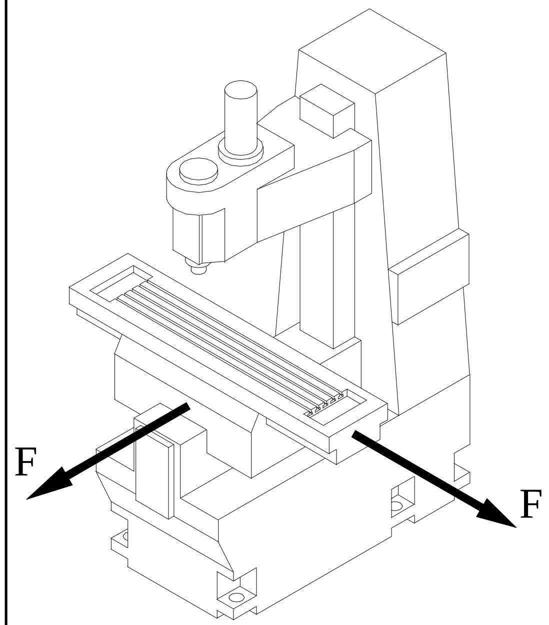
Sindle speed

The spindle turning speed is programmed in RPM.




Axis feedrate

The feedrate of the axes (F) is programmed in m/min.



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 radius values.

# 2.1 Tool management.

The tools to be used with this CNC may be placed in a tool magazine. Depending on whether the machine has or not a tool magazine, the tool change may be carried out as follows:

– If the machine does not have a tool magazine, the tool change is manual (like on a conventional machine).

– If the machine has a tool magazine, the CNC manages the tool change automatically.

# 2.2 Tool table.

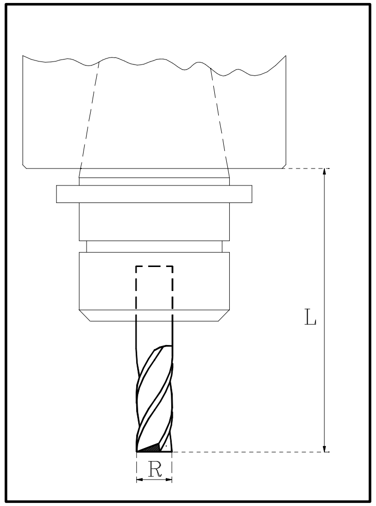
The tool data is stored in the tool table. When a tool change takes place, the CNC assumes the data set for that tool.

The data shown in the table is:

**T: TOOL NUMBER**

**D: OFFSET ASSOCIATED WITH THE TOOL**

It defines the tool dimensions.




L: Tool length.

R: Tool radius.

I: Radius wear.

K: Length wear.

**NOMINAL LIFE**

Machining time or number of operations that could be carried out with the tool.

**REAL LIFE**

Machining time or number of operations carried out.

**FAMILY**

Tools with similar characteristics.

**STATUS**

Tool type:

- N: Normal.
- S: Special.

Tool status:

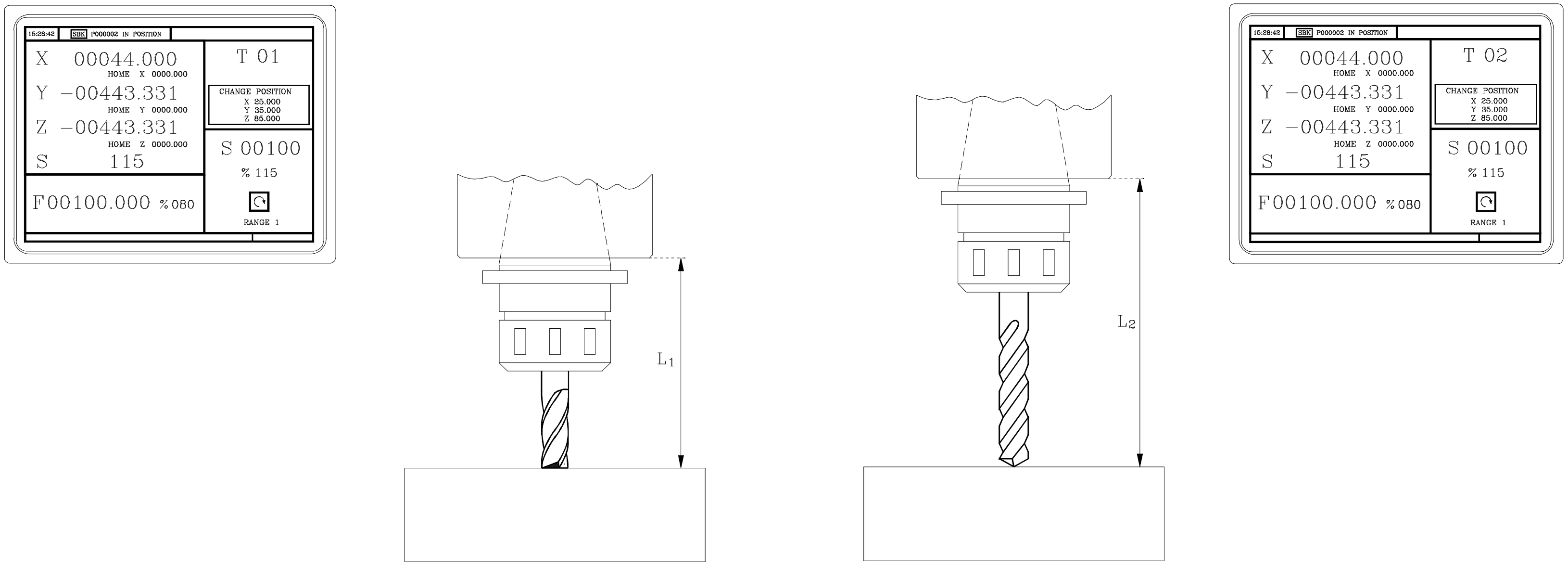
- A: Available.
- E: Expired. (Real life > Nominal life).
- R: Rejected by the PLC.

This data is updated by the CNC. The operator cannot change them.

When requesting an expired or rejected tool, the CNC looks for a tool of the same family. If there is one, it will select it; if not, it will issue the corresponding error message.

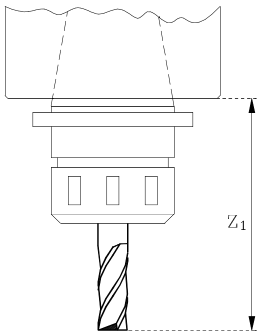
# 2.3 Tool calibration.

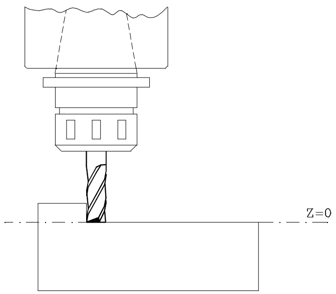
Tool calibration refers to the operation used to indicate to the CNC the length of the tool. This operation must be carried out properly so the parts come out with the right dimensions and the same point is controlled after a tool change.




Different tool dimensions, same point.

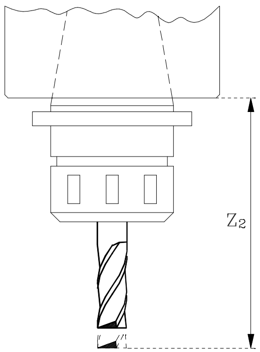
# DEFECTS DUE TO WRONG LENGTH CALIBRATION

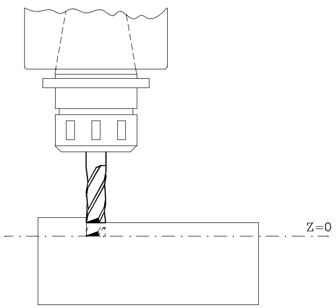



Right part dimensions

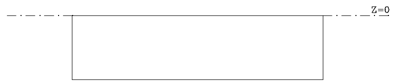
Z1: Real dim.



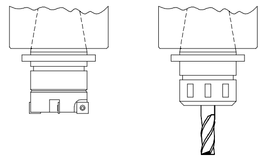




Z2:  Wrong dim.
Z2 > Real dim.

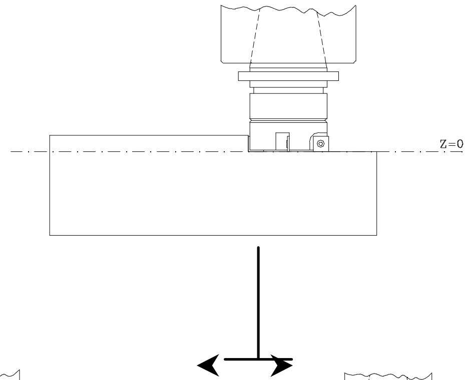
Wrong part dimensions

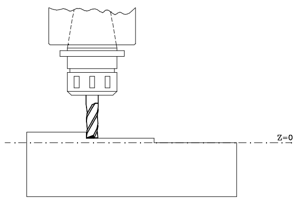



Part to be machined

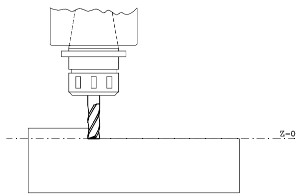


Tools



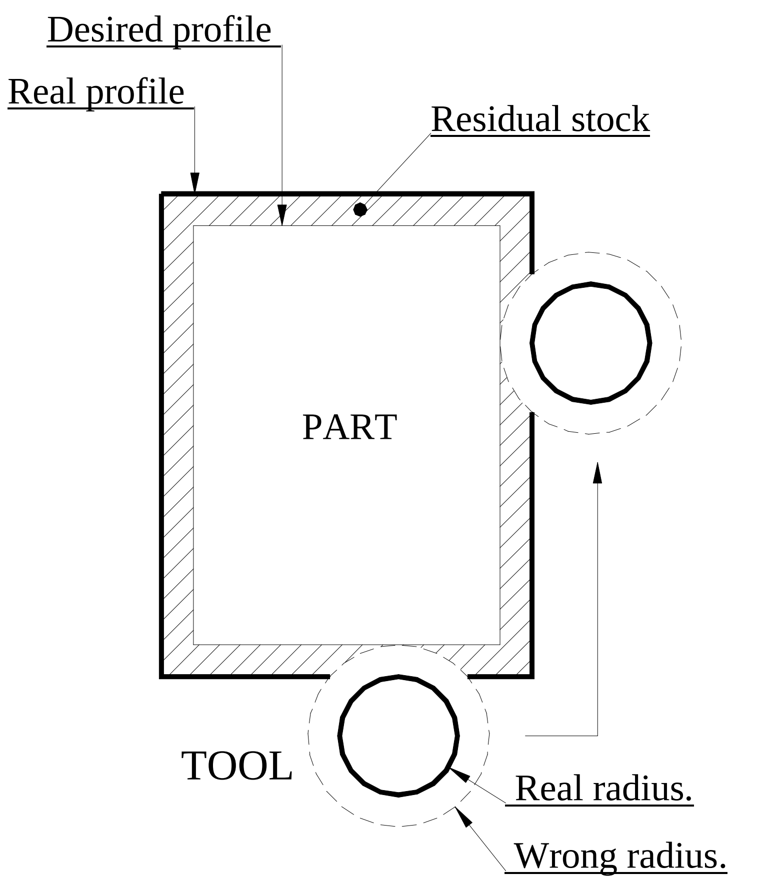



Wrong machining
Tools calibrated wrong



Proper machining
Tools calibrated right

DEFECTS DUE TO WRONG RADIUS VALUES




There is a residual stock due to different radii.

Chapter 3

# Hands-on training

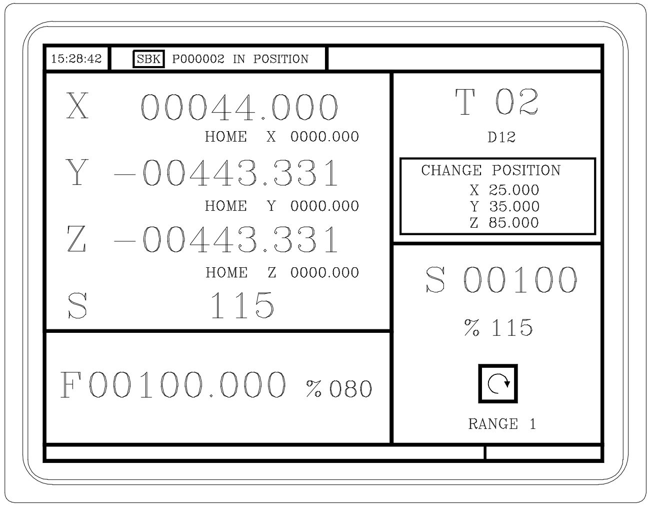
This chapter describes:

- 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 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.

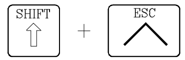
## 3.1.1 Power-up.

On power-up, the CNC will display the following screen.



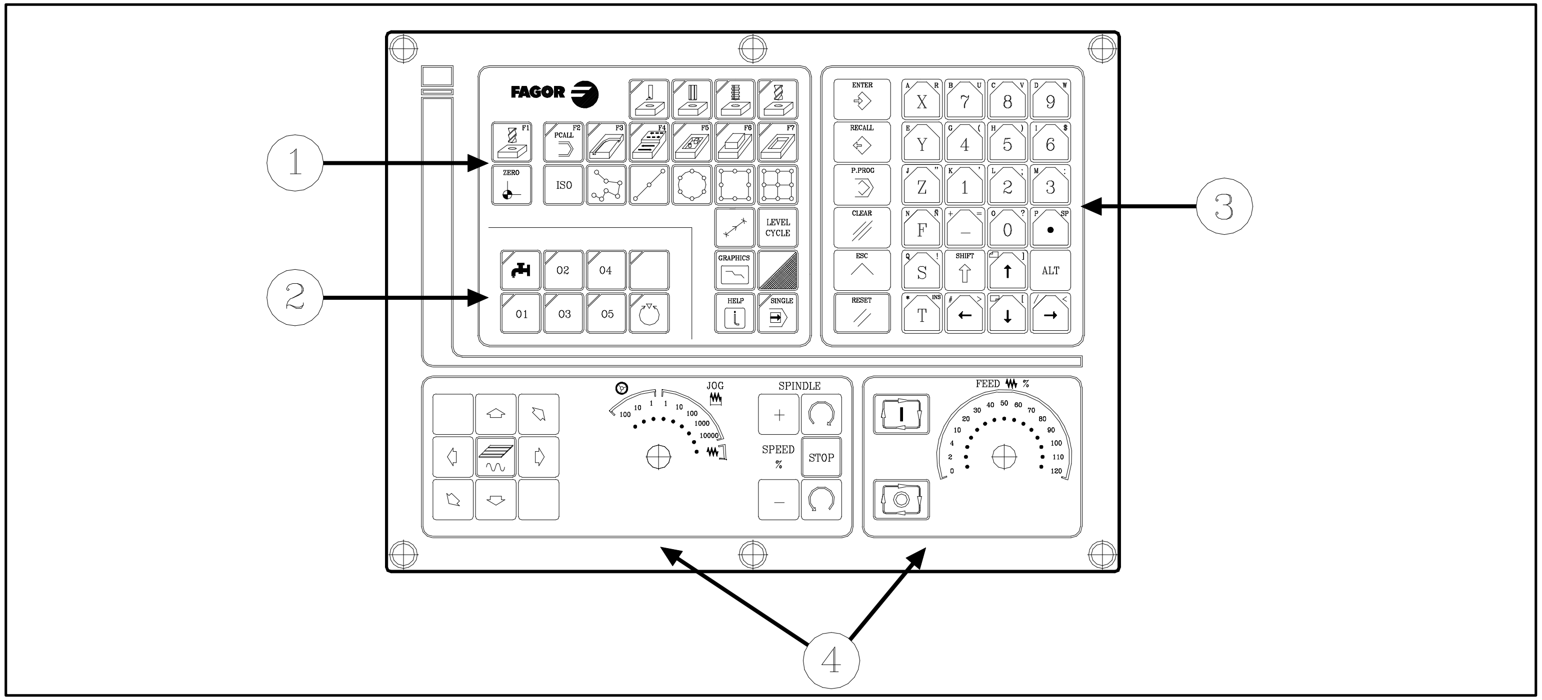

Screen for the MC mode.

If this screen is not displayed, it is because the CNC is in M mode. To enter in MC 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.

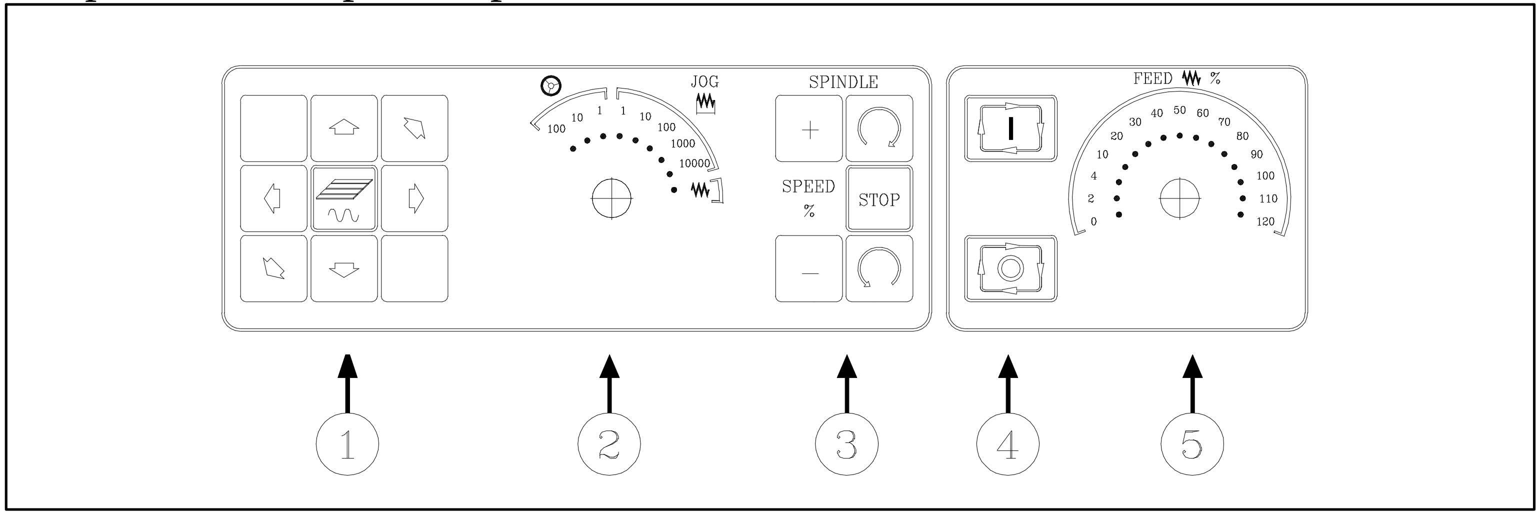
2.- Keys for external devices.

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 ( )).

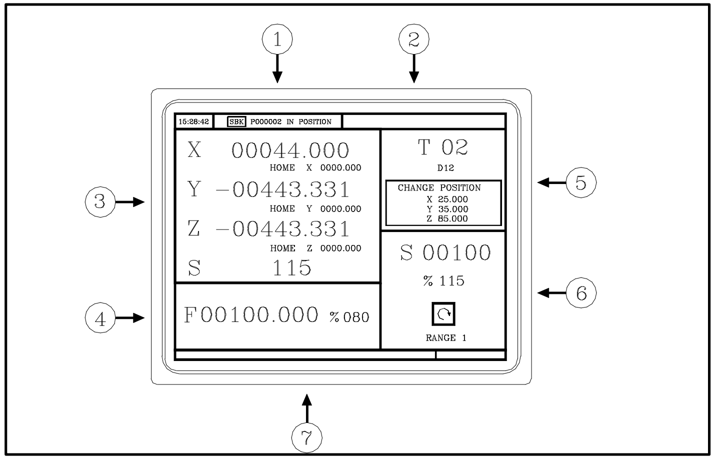
3.- Selection of spindle turning direction ( ) and start-up. Spindle speed override percentage ( + – ).

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 messages.

2.- CNC messages.

3.- Tool position referred to part zero and to home. Actual (real) spindle rpm.

4.- Selected axis feedrate and applied override %.

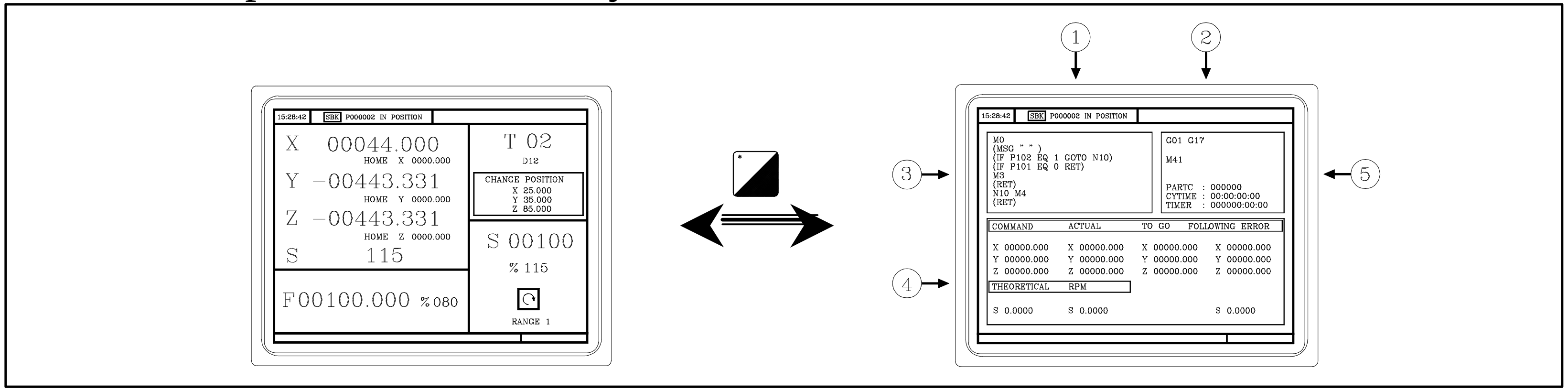
5.- Tool information.

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 messages.

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).

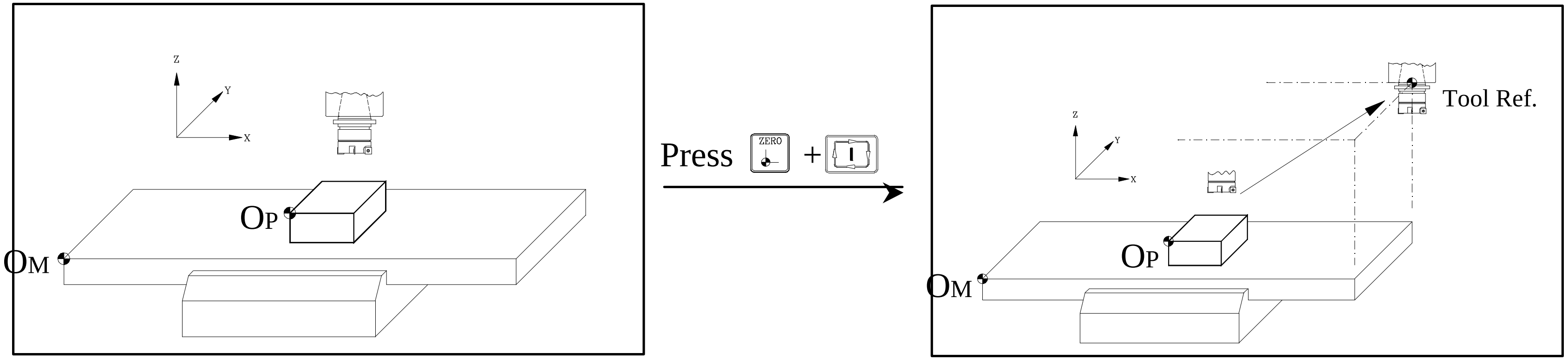
NOTE: Refer to the Operation Manual Chapter 3 Section 3.1

# 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 the three axes at the same time.




The CNC does not know the position of the axes.

The CNC shows the coordinates referred to the OP considering the tool dimensions.

NOTE: Refer to the Operation Manual Chapter 3 Section 3.3

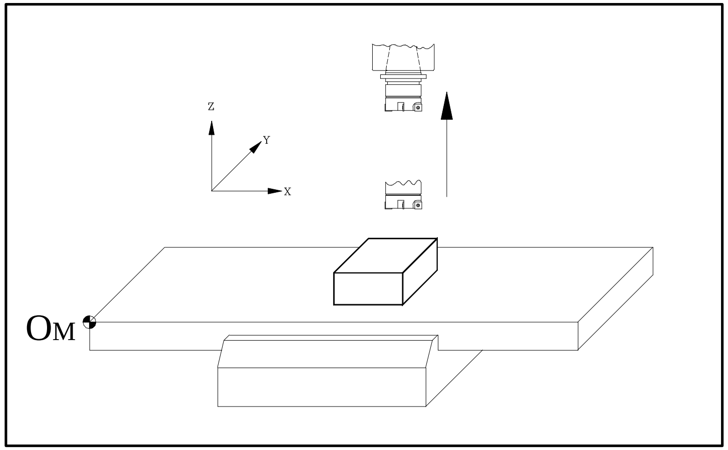
## 3.2.2 Without maintaining the part zero.

The "Home Search" is carried out on one axis at a time.

The CNC does not know the position of the axes.

Home search on the Z axis

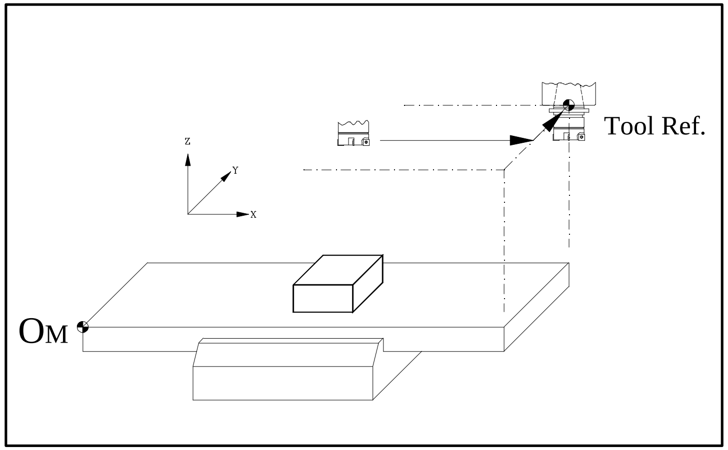
Press Z + ZERO + I




Home search on the X and Y axes.

Press X + ZERO + I

Press Y + ZERO + I



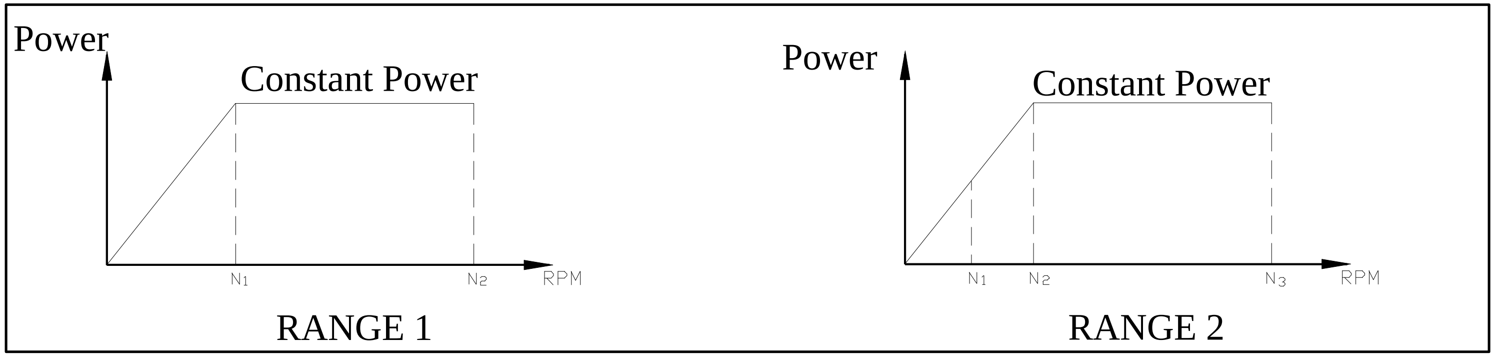

The CNC shows the coordinates referred to OM, considering the tool dimensions.

NOTE: Refer to the Operation Manual Chapter 3 Section 3.3

# 3.3 Spindle.

## 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.



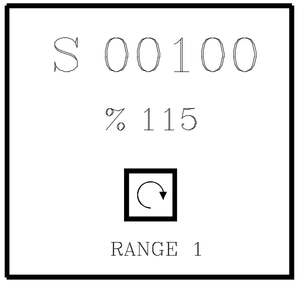

If the work speed is between $N_1$ and $N_2$, RANGE 1 should be used and if between $N_2$ and $N_3$, RANGE 2. Always try to work at constant power in order to extend tool life.

## 3.3.2 Spindle control.

To select the work speed (in rpm), press:

S + (turning speed) + 

The CNC shows the following information:




Selected speed.
Applied percentage.
Turning direction.
Active spindle range.

Use the following keys of the operator panel to start the spindle.

Start the spindle clockwise.

STOP Stop the spindle.

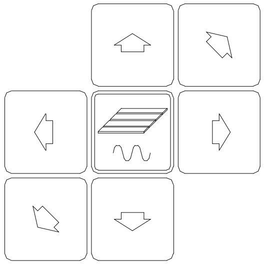
Start the spindle counter-clockwise.

\+ – Increase or decrease the override percentage applied to the spindle turning speed.

NOTE: Refer to the Operation Manual Chapter 3 Section 3.6.1

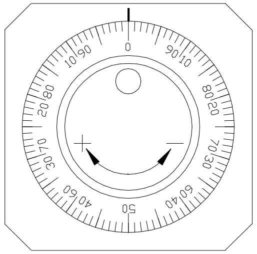
# 3.4 Axis jog.

To jog the axes, we will use:



Each key is used for moving the axis in one direction according to the axes of the machine. (Section 1.1)

JOG keys

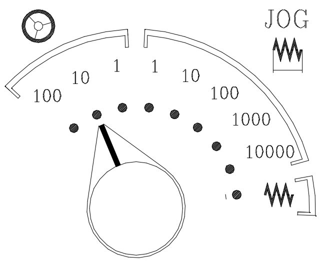


It can have one, two or three handwheels. The axes move in the turning direction of the handwheels.

Handwheel

To select the jog mode, use the selector switch:

Handwheel jog

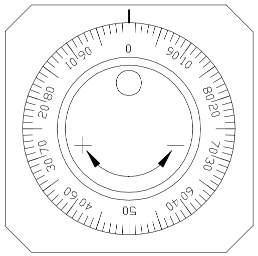



Incremental movement

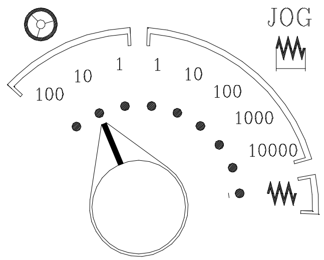
Continuous movement

## 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.

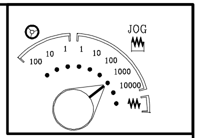
  The machine moves the axis as the handwheel is being turned.

- If the machine has 2 or more handwheels:

  The machine moves an axis with each handwheel.

NOTE: Refer to the Operation Manual Chapter 3 Section 3.4.3

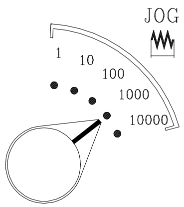
## 3.4.2 JOG.




### Incremental JOG.

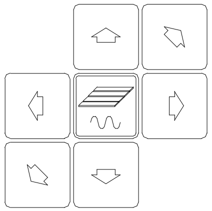
Every time a JOG key is pressed, the axis will move the selected increment at the programmed feedrate. (in rapid, if F=0).

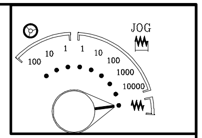
– Select the distance to move at the selector. ( position).




| | | |
|---|---|---|
| (1) | : | 0.001 mm |
| (10) | : | 0.010 mm |
| (100) | : | 0.100 mm |
| (1000) | : | 1.000 mm |
| (10000) | : | 10.000 mm |

– Move the axes with the JOG keys.





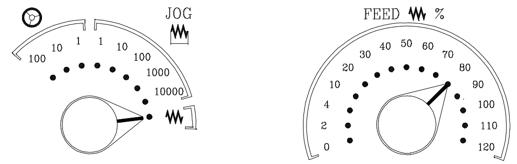

### Continuous JOG

When pressing a JOG key, the axis moves at the feedrate of the selected feedrate "F" override percentage (0% to 120%).

– Enter the feedrate value:

F + <Feedrate value> + ENTER

– Modify the percentage of the programmed feedrate.




Applied percentage

– Jog the axes with the JOG keyboard.

– If is pressed while the axes are moving, they will move at the fastest feedrate possible (set by the machine manufacturer).

NOTE: Refer to the Operation Manual Chapter 3 Section 3.4.1/3.4.2

### 3.4.3 Automatic axis movement to a particular position.

By means of the key, an axis may be moved to a particular coordinate.

Follow these steps:

– Select the axis to be moved at the stantard screen. X Y Z

– Enter the value of the destination point.

– Press .

The axis will move to the programmed point at the selected feedrate.

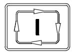
# 3.5 Tools.

## 3.5.1 Tool selection.

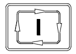
Depending on the machine, there are two possibilities:

### Machine with manual tool changer.

The tool change is carried out like on a conventional machine:

- Change the tool on the machine.
- Press .
- Enter the tool number so the CNC assumes the values of the corresponding tool table.
- Press .

### Machine with automatic tool changer.

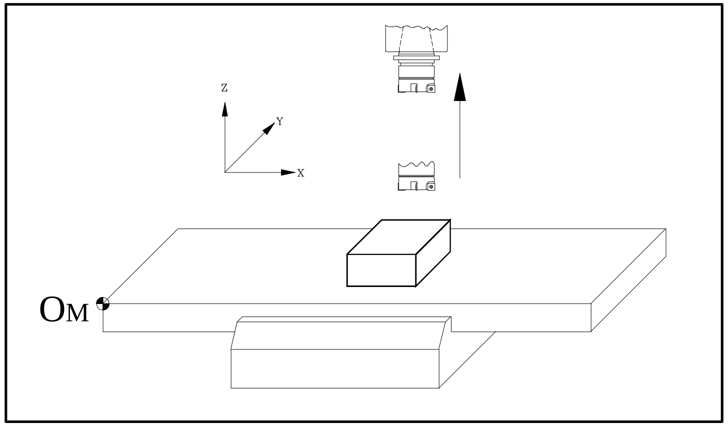
- Press .
- Enter the tool number.
- Press .
- The CNC manages the tool change.

NOTE: Refer to the Operation Manual Chapter 3 Section 3.5.1

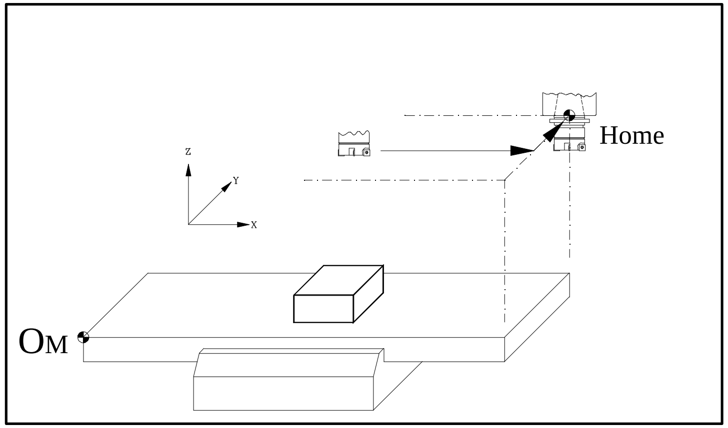
## 3.5.2 Tool calibration.

– Just before calibrating the tools, a “Home Search” must be carried out on all axes.

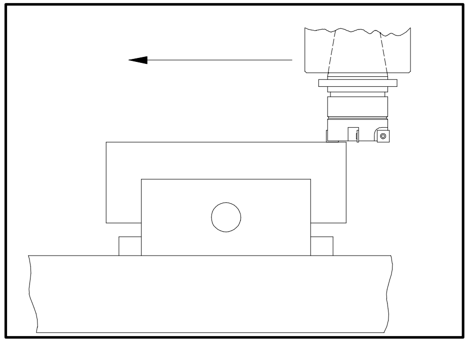
Home search on the Z axis.




Home search on the X and Y axes.




– A flat surface is needed for calibrating the tools. Use continuous JOG or handwheels for level milling the surface.

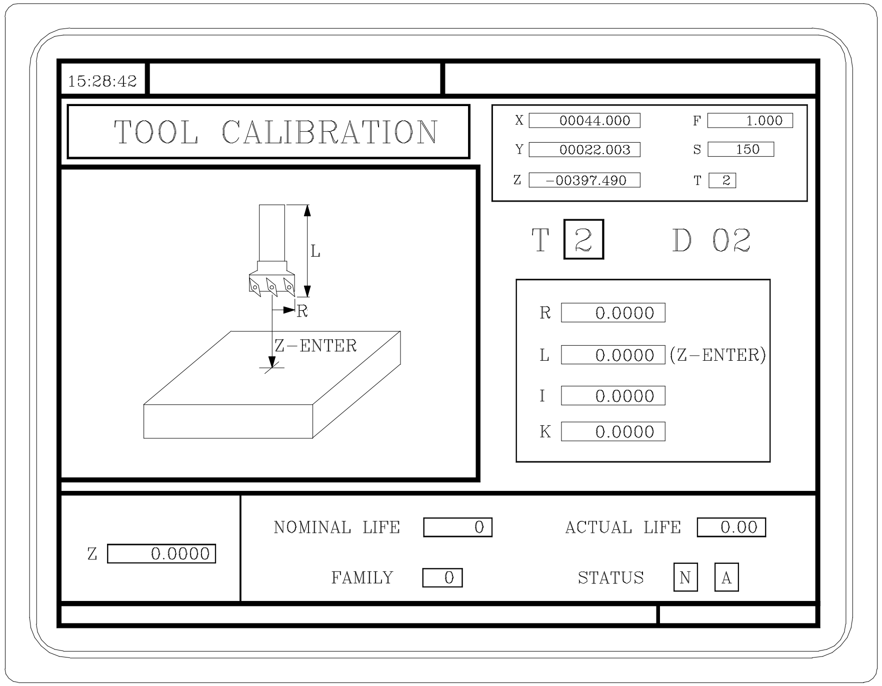


– Enter in the calibration mode. Press [F1]. The CNC displays the tool calibration screen.

Work mode.

Help graphics.

Height of the part used for tool calibration.




Actual position of the axes and cutting conditions.

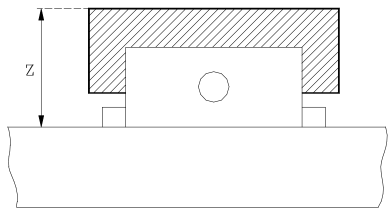
Tool number.
Tool dimensions.

Data on current tool status.

NOTE
Use ↑ ↓ ← → to move the cursor around

NOTE: Refer to the Operation Manual Chapter 3 Section 3.5.2

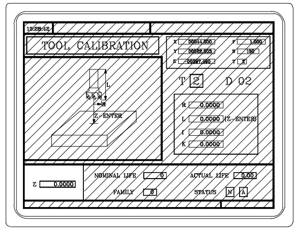
1.- Measure the part.




Part dimensions

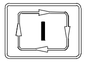
– Go to the tool calibration window.

– Enter the Z value.

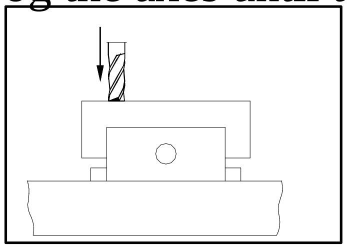


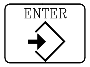

2.- Start the spindle.

3.- Select the tool to be calibrated. The CNC will assign the same tool offset number (D).

+ (tool number) +


5.- Jog the axes until touching the part along the Z axis. Press:



+



The CNC calculates the length and assigns it to the tool.

6.- Enter the rest of the data (Radius, Nominal life, Real life and family code). The K value is set to zero when calibrating.

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

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, R, L, I, K, Nominal Life, Real Life or Family), enter in the calibration mode and press:

[T] + (Tool number) + [RECALL]

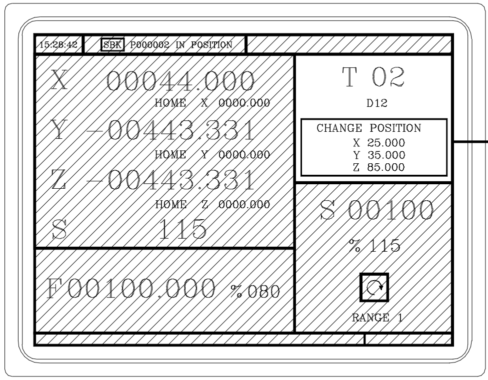
The CNC shows the data for that tool. To change it, place the cursor over the value to be modified, key in the new value and press [ENTER].

To quit the calibration mode, press [ESC].

NOTE: Refer to the Operation Manual Chapter 3 Section 3.5.2.1

## 3.5.4 Tool change point.

The machine manufacturer may allow selecting the tool change position.




Tool change position referred to home.

Enter the X, Y and Z values of the point chosen as the tool change position.

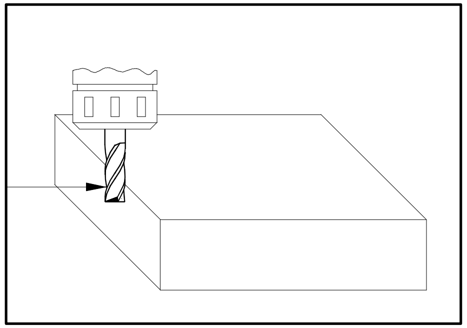
- T + X + (X value) + ENTER
- T + Y + (Y value) + ENTER
- T + Z + (Z value) + ENTER

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.

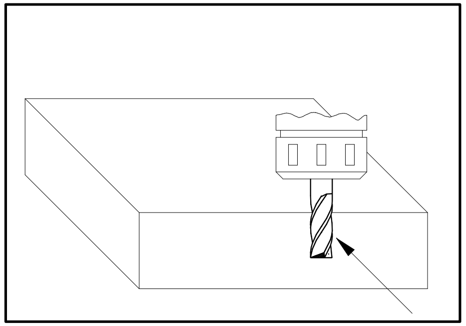
NOTE: Refer to the Operation Manual Chapter 3 Section 3.5.1.1

# 3.6 Checking for proper calibration.

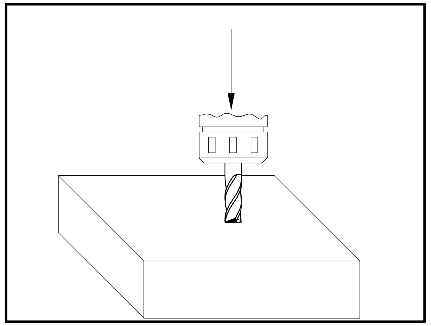
– Preset the part zero.



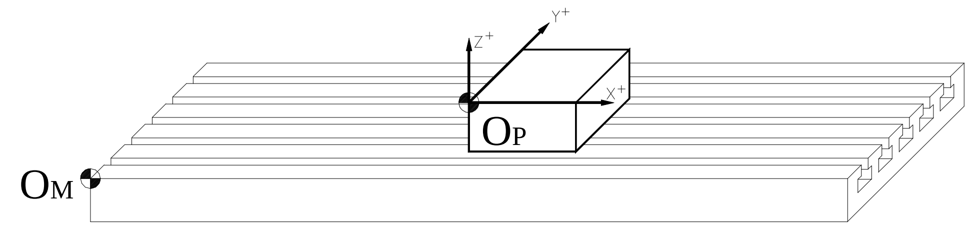
Approach the tool along X.
Press X + 0 + ENTER



Approach the tool along Y.
Press Y + 0 + ENTER



Approach the tool along Z.
Press Z + 0 + ENTER




Withdraw the tool.
Part Zero position.

– Start the spindle, touch the part surface with several tools and check the values on the screen.

– The tools are different, but the values on the screen must be the same.

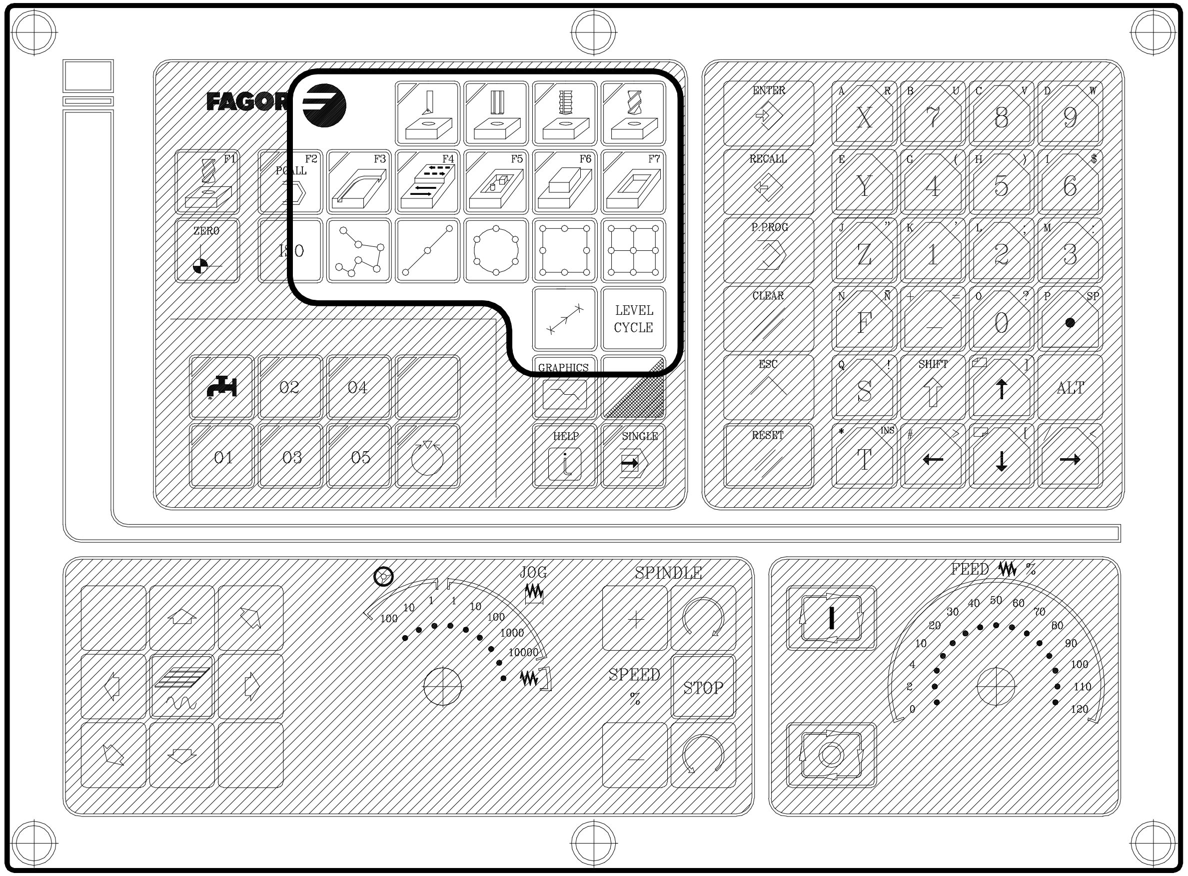
Chapter 4

# Automatic Operations

This chapter describes:

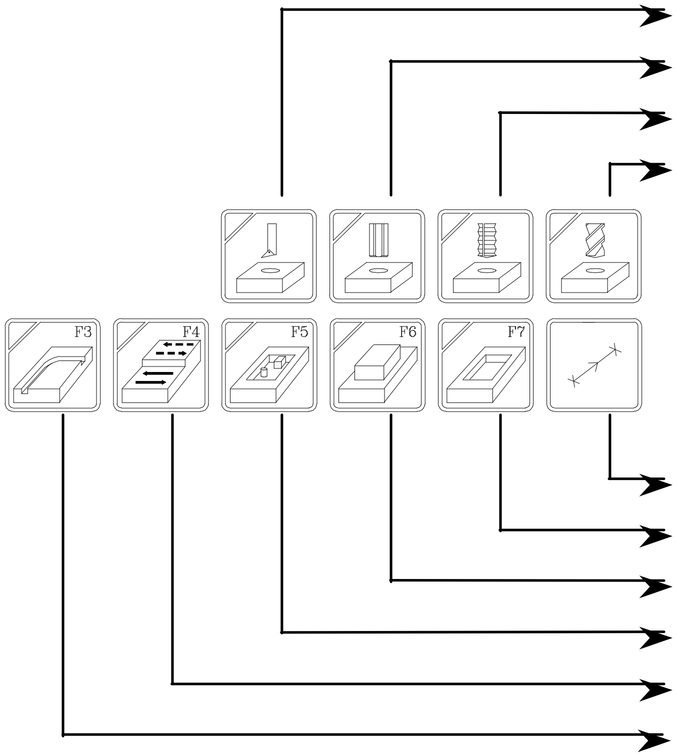
- Which are the keys associated with the automatic operations.
- Which are the various work modes.
- Example of an operation and a positioning cycle.
  - > How to edit the parameters of the operation and what they mean.
  - > How to simulate an operation and which are the graphic parameters.
  - > How to execute an operation.
    - – Tool inspection.
    - – Tool wear compensation.

# 4.1 Operation keys.



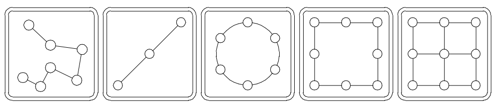

Layout of the automatic function keys.

# Operation keys:





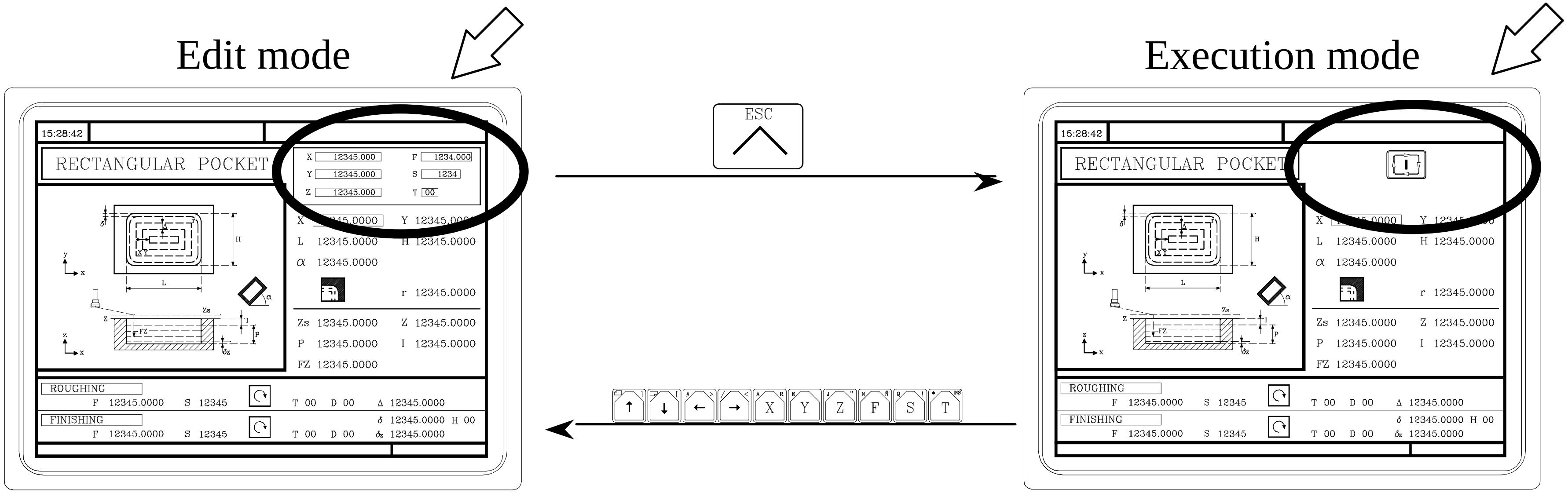
Selection of the cycle level within an operation



Used to associate a positioning cycle with Boring, Reaming, Threading, Drilling and Center punching operations.

# 4.2 Work modes.

There are 2 work modes:




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. ( )

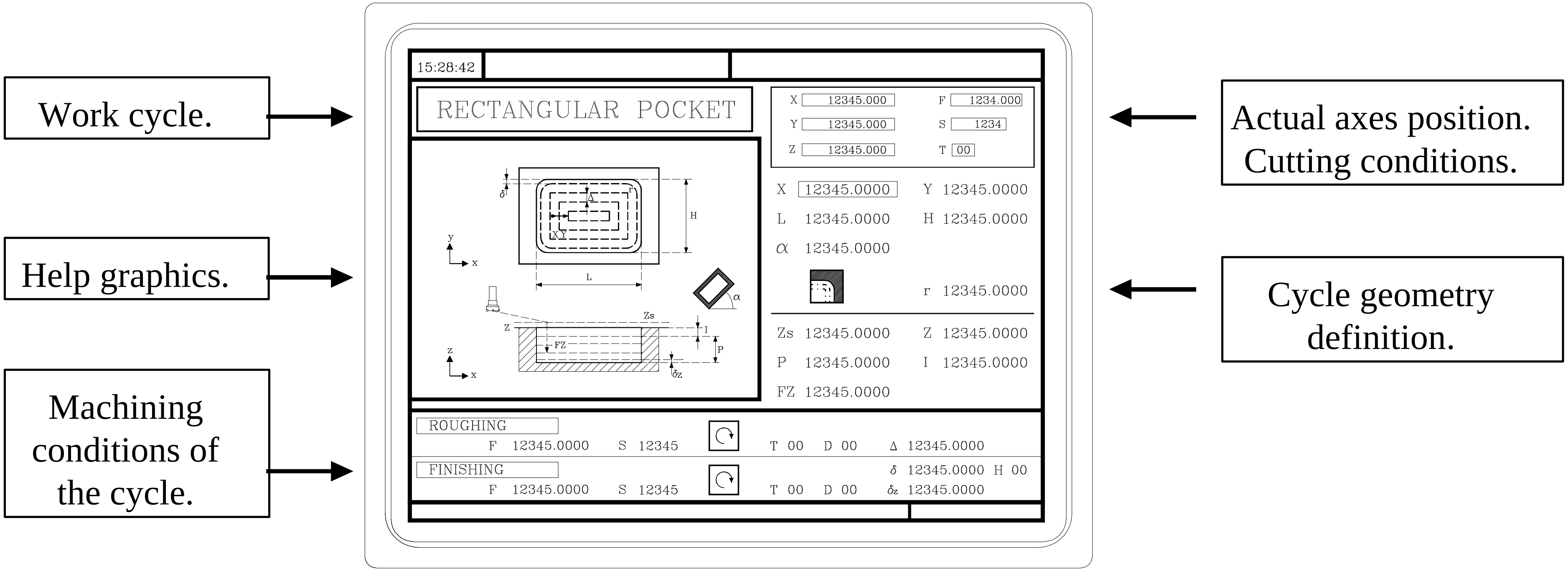
NOTE: Refer to the Operation Manual Chapter 4 Section 4.2

# 4.3 Example of an automatic operation.

## 4.3.1 Edit an operation.

### 4.3.1.1 Rectangular pocket.

– Select the Rectangular pocket operation. Press F7.




– Use the LEVEL CYCLE 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 Y or Z. The CNC selects the first coordinate of the axis. Press it again to select the second coordinate.
- 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.

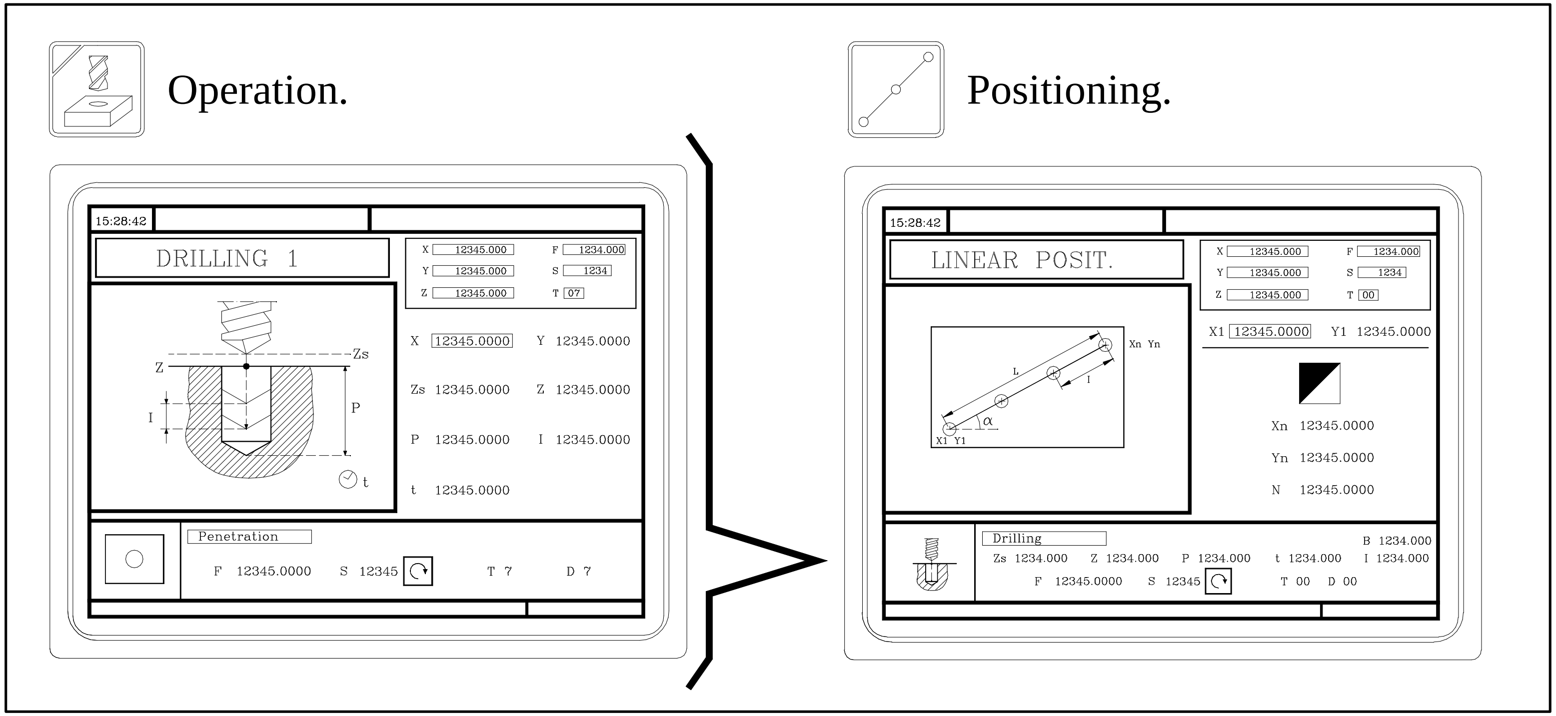
After making this selection:

- If it is a data, key in the new value and press ENTER.
- If it is an icon, press until the desired one is selected and then ENTER.
- If it is a coordinate, there are two possibilities:
  –Key in the new value and press ENTER.
  –Press RECALL + ENTER. The CNC will take the current position of the axes as the value.

NOTE: Refer to the Operation Manual Chapter 4 Section 4.1

## 4.3.1.2 Associate a positioning with an operation.

If it is a Boring, Reaming, Threading, Drilling or Center Punching operation, a positioning cycle may be associated with it. After setting the operation, choose the type of positioning. ( ).




Each positioning can be defined in several ways. To choose the right group of data, place the cursor over the icon and press .

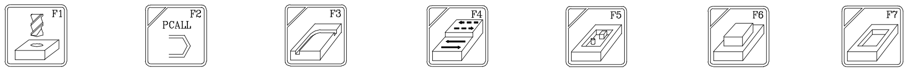
## 4.3.2 Simulate an operation.

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:

| | F1 | F2 | F3 | F4 | F5 | F6 | F7 |
|---|---|---|---|---|---|---|---|
| Function: | TYPE OF GRAPHIC | DISPLAY AREA | ZOOM | | GRAPHIC PARAMETERS | CLEAR SCREEN | + |

Key:




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.
  - “3D” Graphics.
    The three-dimensional tool path is represented by color lines.
  - “XY, XZ, YZ” Graphics.
    Color lines represent the tool path in the selected plane.
  - “Combined” Graphics.
    The screen is divided into four quadrants showing the XY, XZ, YZ planes and the 3D view.
  - Top view.
    It displays a solid XY plane indicating the depth of the part with different gray tones. It also shows two sections (XZ and YZ) of the part.
  - “Solid” Graphics.
    It shows a three-dimensional part. Starting at an initial block. During simulation, the tool can be seen removing material as well as the resulting part.

• <u>Display area.</u>

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 ENTER.

• <u>ZOOM.</u>

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 ENTER.

–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 “3D”, “XY, XZ, YZ” and “Top view” graphics.

Colors for solid graphics: For changing the colors of the tool and the part on “Top view” and “Solid” graphics.

• Clear screen.

It clears the screen. While in “Solid” graphics mode, it shows 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 SINGLE.

Once the data has been entered, press ESC. The CNC screen shows the Cycle Start key (I) and lets execute the operation.

To start the execution, press I.

Once execution has started:

O : Interrupts the execution. While interrupted, if we press:

I : Resumes the execution.

RESET : Cancels the execution.

GRAPHICS : 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 thread.

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 (JOG keys and handwheels), check or change the tool, stop or start the spindle, change the tool wear value, etc.

– Press to reposition the axes. If more than one axis was moved, the CNC will request the repositioning order (sequence).

– 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. This option may be executed during tool inspection or while the machine is running.

– 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) +

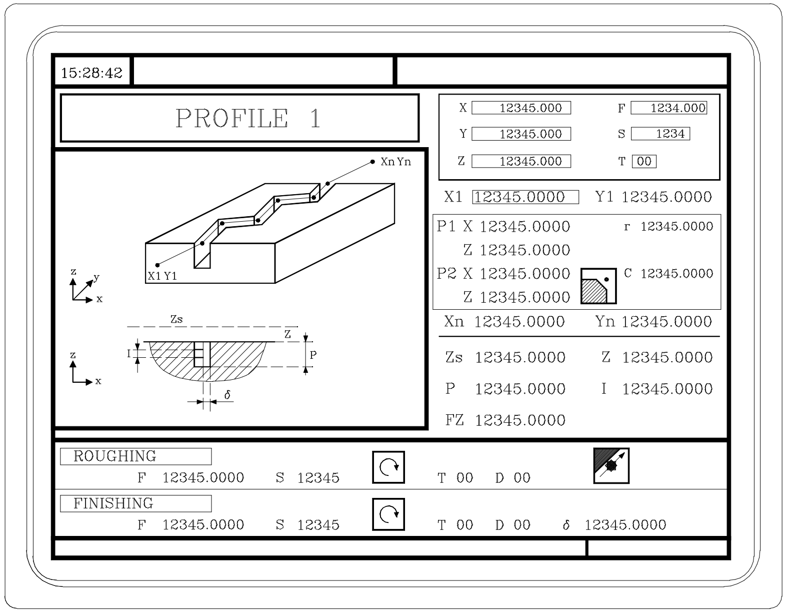
– To finish, press .

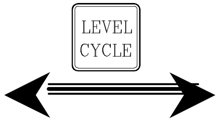
NOTE: The modifications are not assumed until the tool is selected.

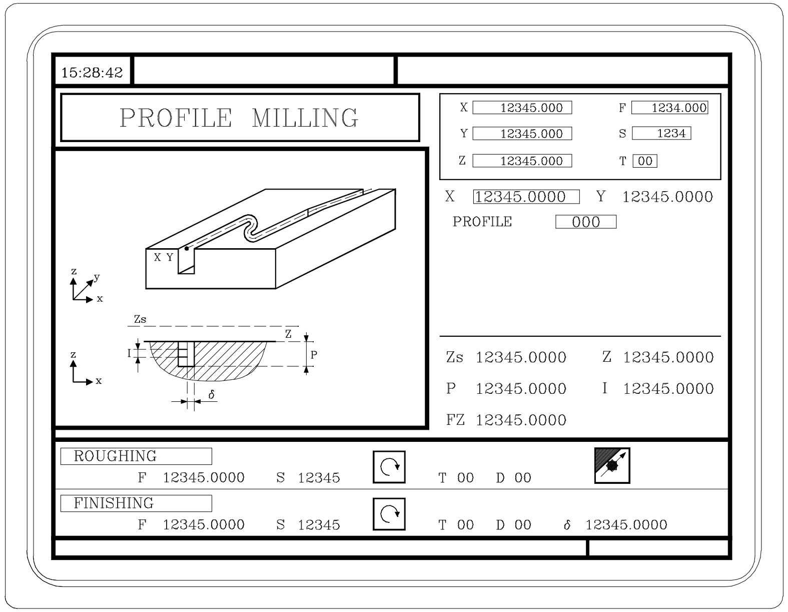
Chapter 5

# Summary of work cycles

# 5.1 Profile milling operation.



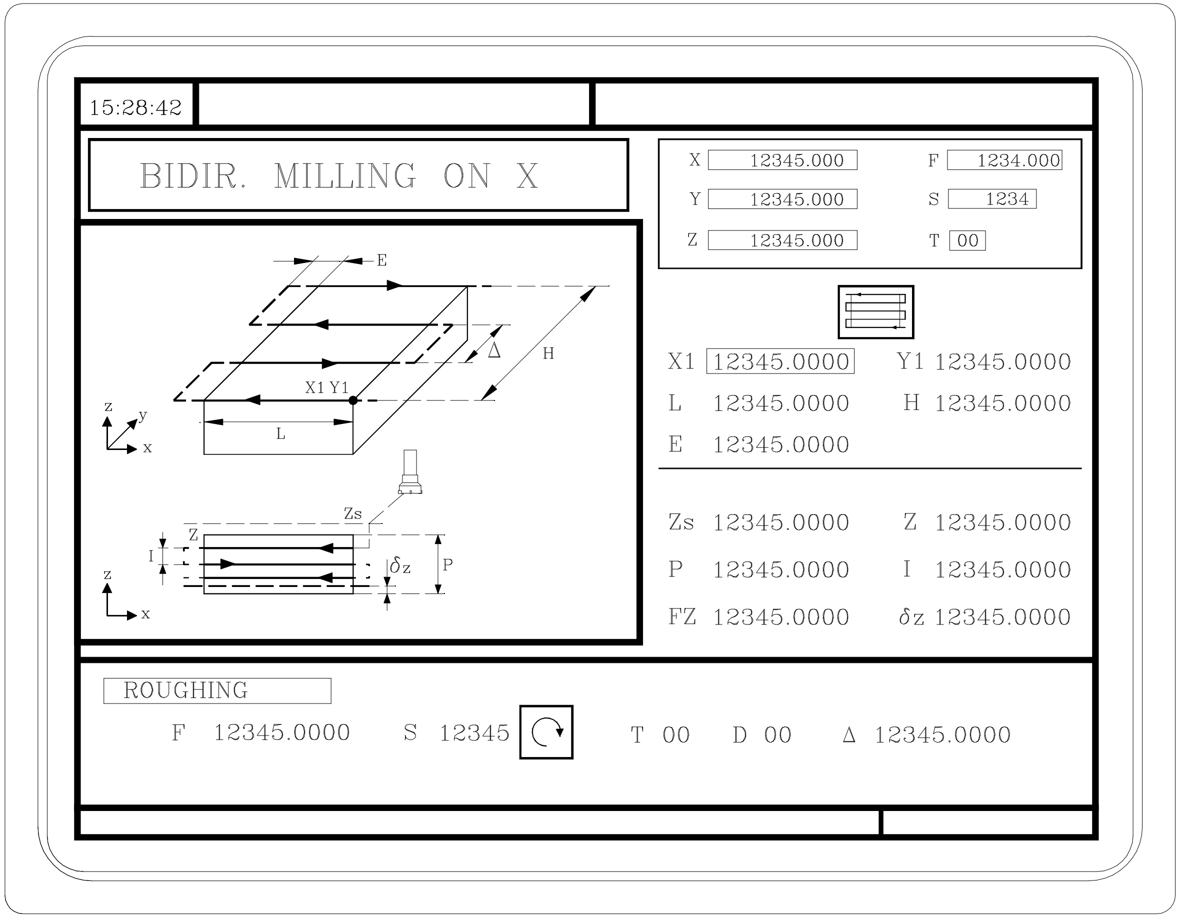




At this cycle level, the profile is defined by points. (Up to a maximum of 12 points).

At this cycle level, the profile is defined by the profile editor. (Section 5.16).

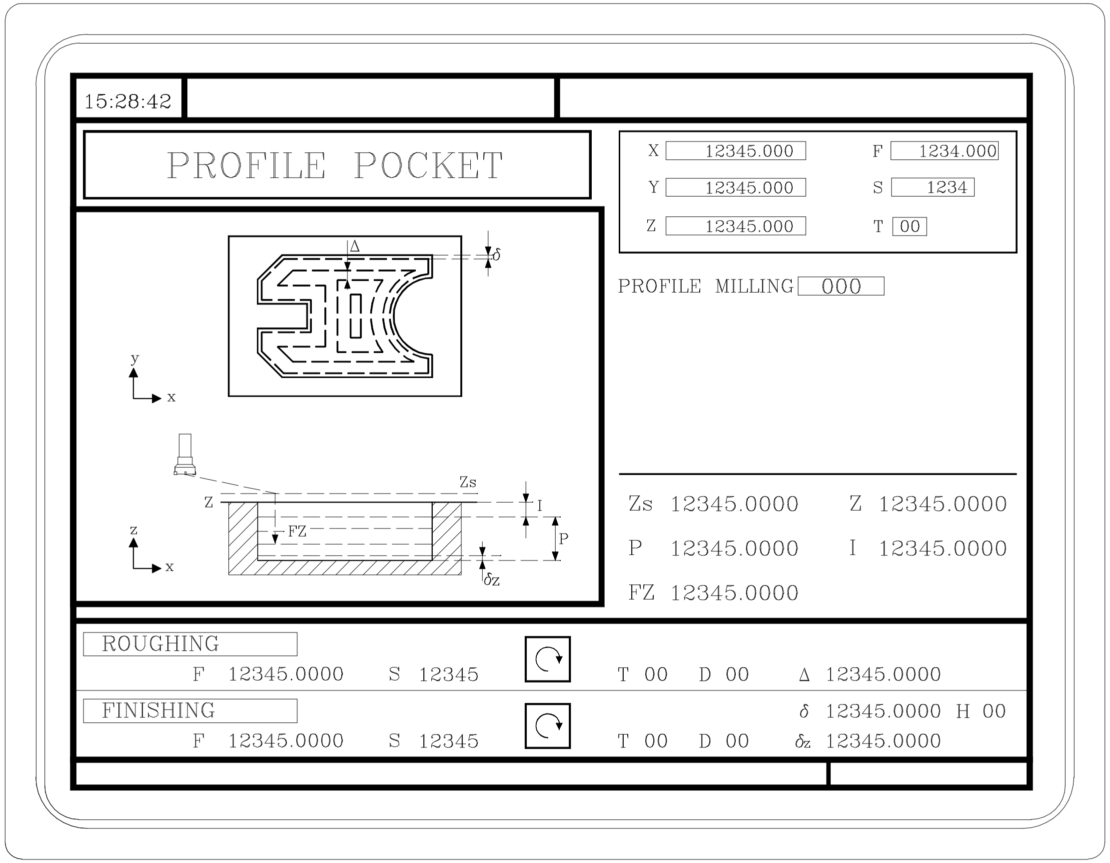
NOTE: Refer to the Operation Manual Chapter 4 Section 4.3

# 5.2 Surface milling operation.




NOTE: Refer to the Operation Manual Chapter 4 Section 4.4

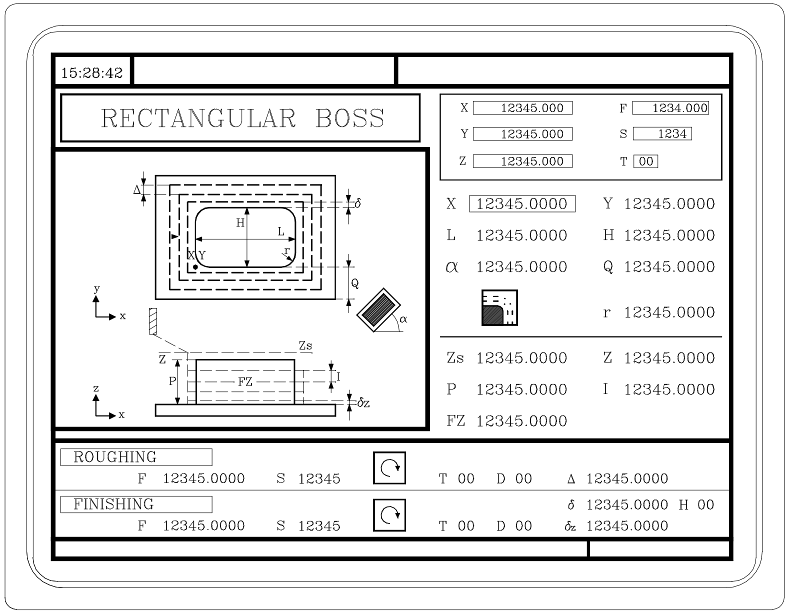
# 5.3 Pocket cycle with Profile.



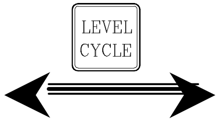

The profile is generated with the profile editor (Section 5.16).

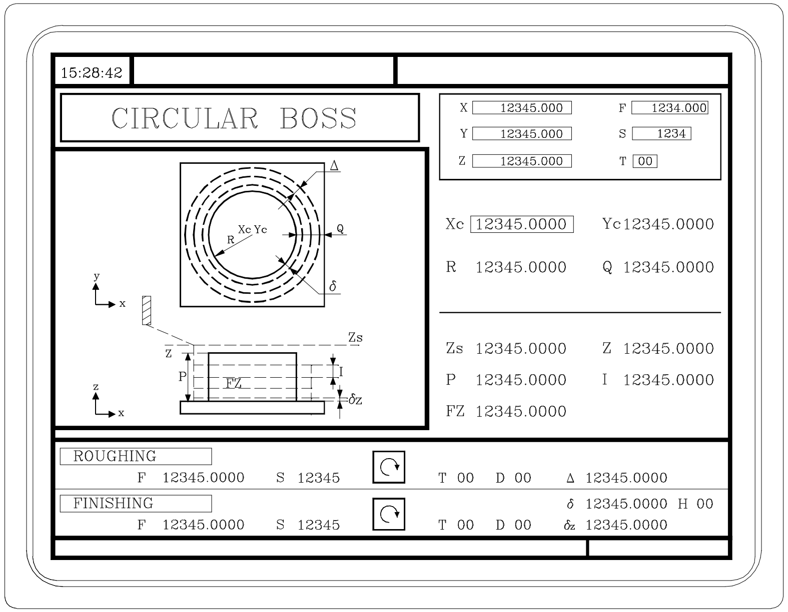
NOTE: Refer to the Operation Manual Chapter 4 Section 4.5

# 5.4 Rectangular and Circular Boss milling cycles.




Rectangular Boss

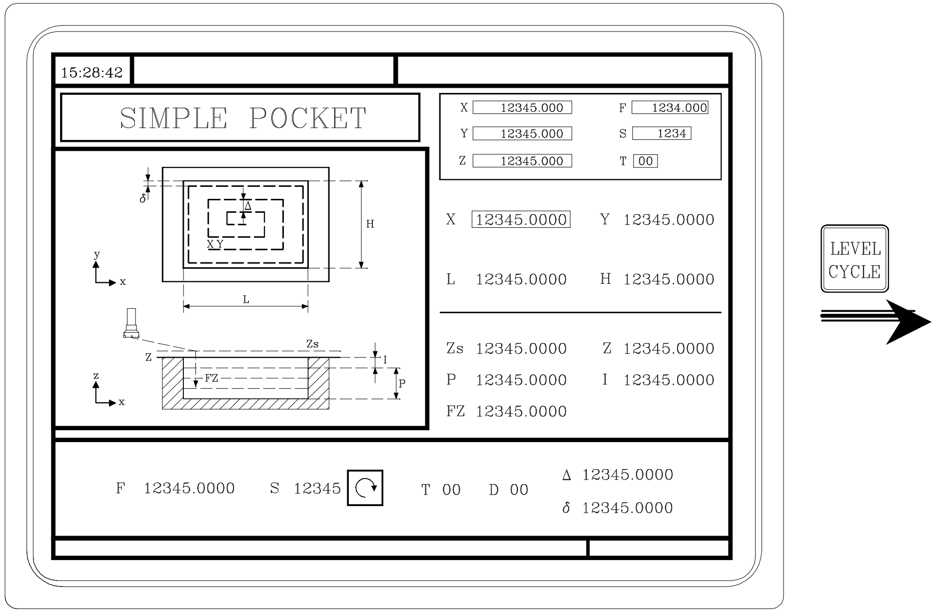


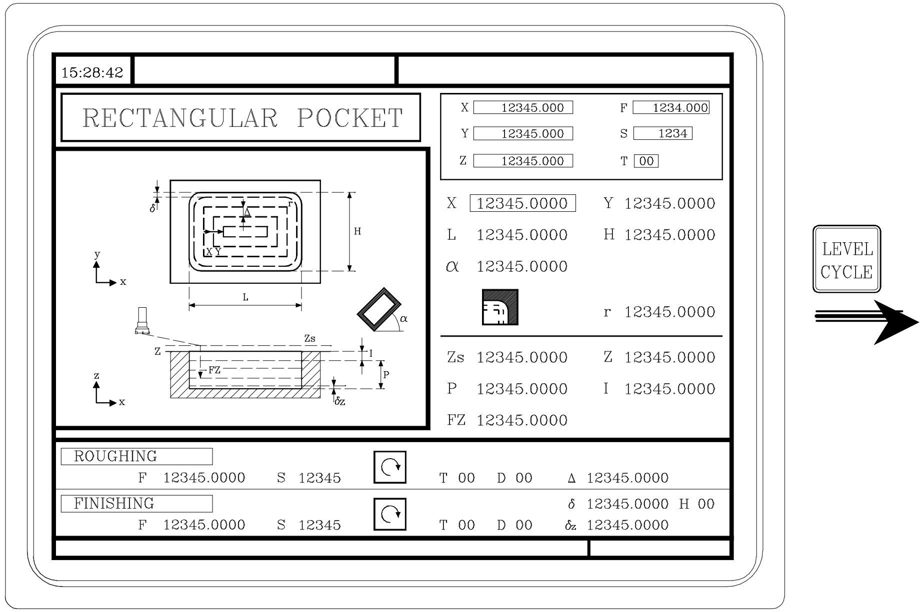

Circular Boss

NOTE: Refer to the Operation Manual Chapter 4 Section 4.6

# 5.5 Rectangular and Circular pocket milling cycles.



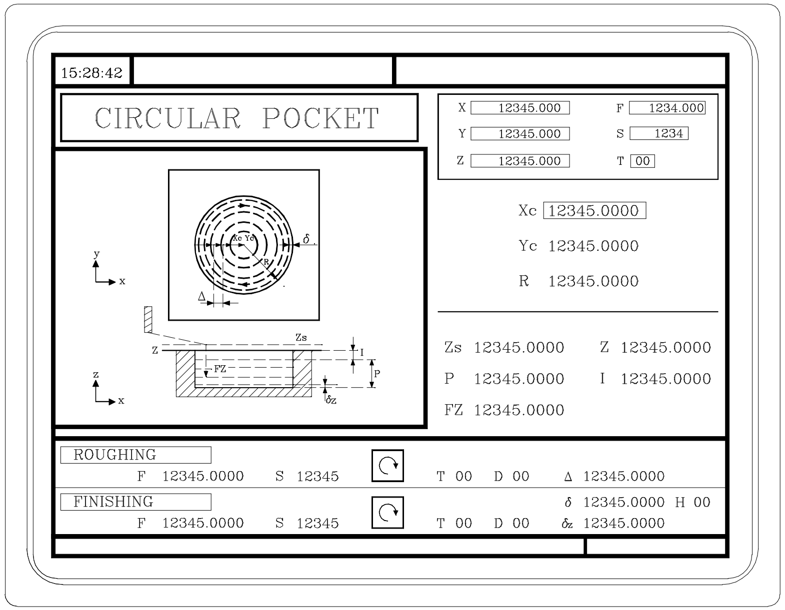
Simple pocket



Rectangular pocket

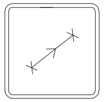
At this cycle level, the type of pocket corner may be chosen as well as the inclination angle of the pocket.

NOTE: Refer to the Operation Manual Chapter 4 Section 4.7

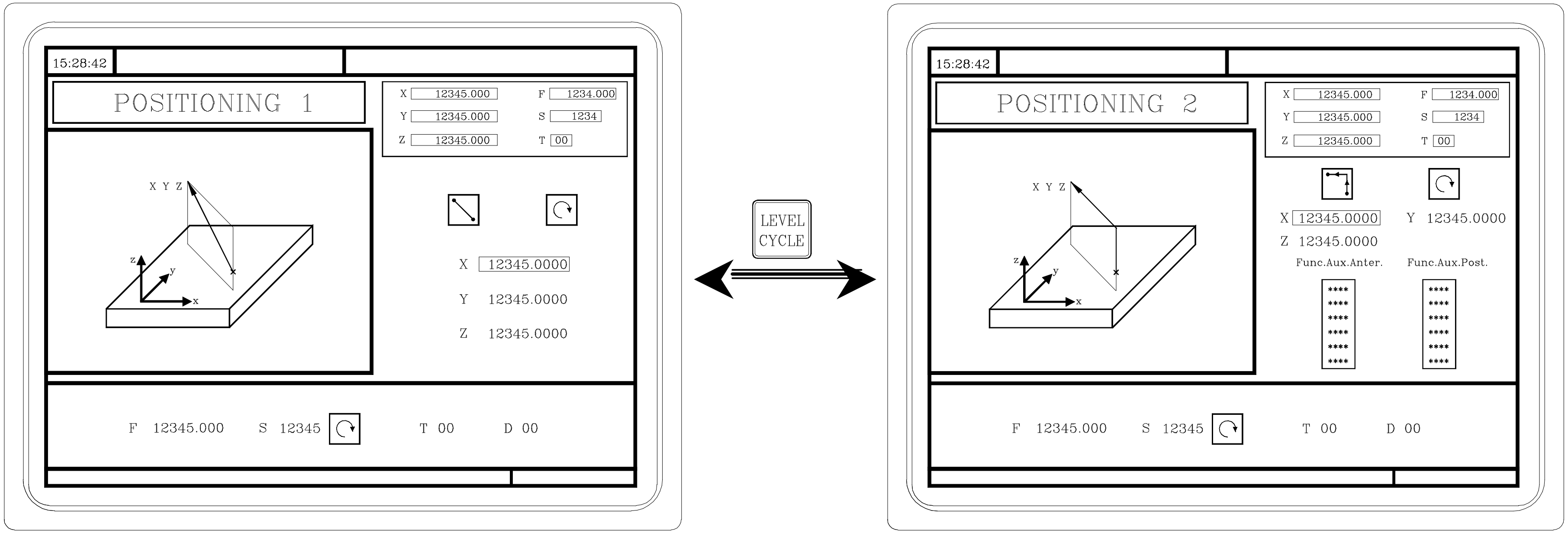



Circular pocket

NOTE: Refer to the Operation Manual Chapter 4 Section 4.7



# 5.6 Positioning.

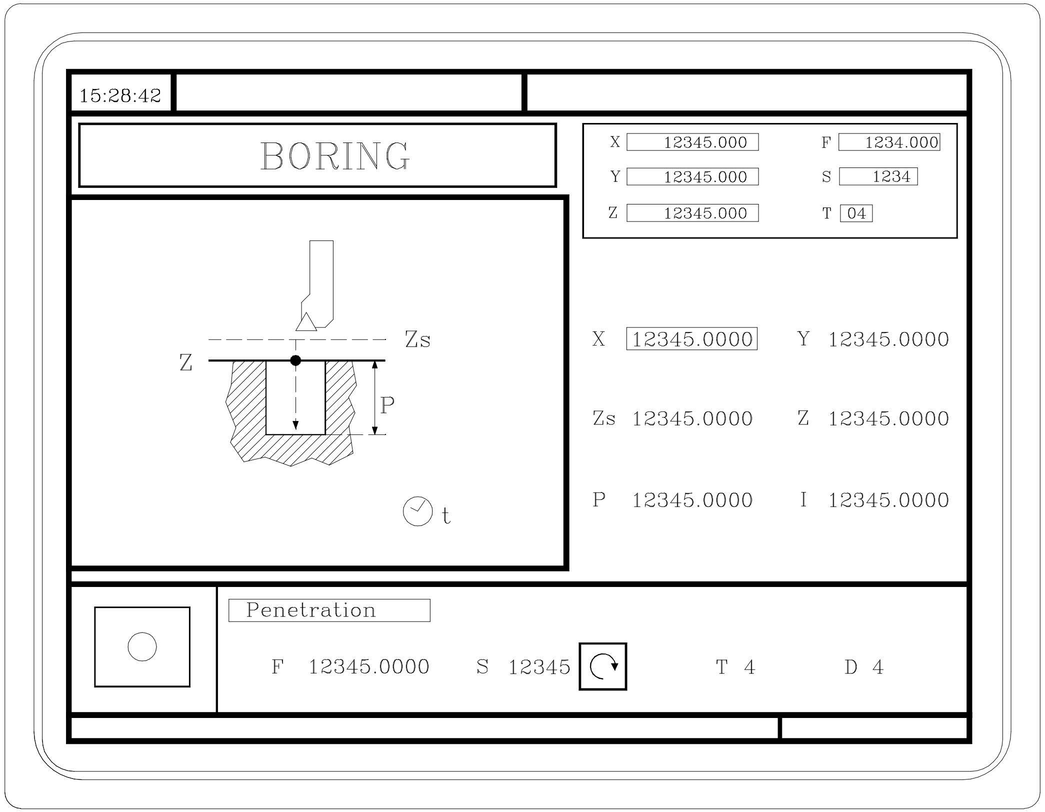



At this cycle level, auxiliary functions may be defined to be executed before or after the movement.

NOTE: Refer to the Operation Manual Chapter 4 Section 4.8

# 5.7 Boring operation.

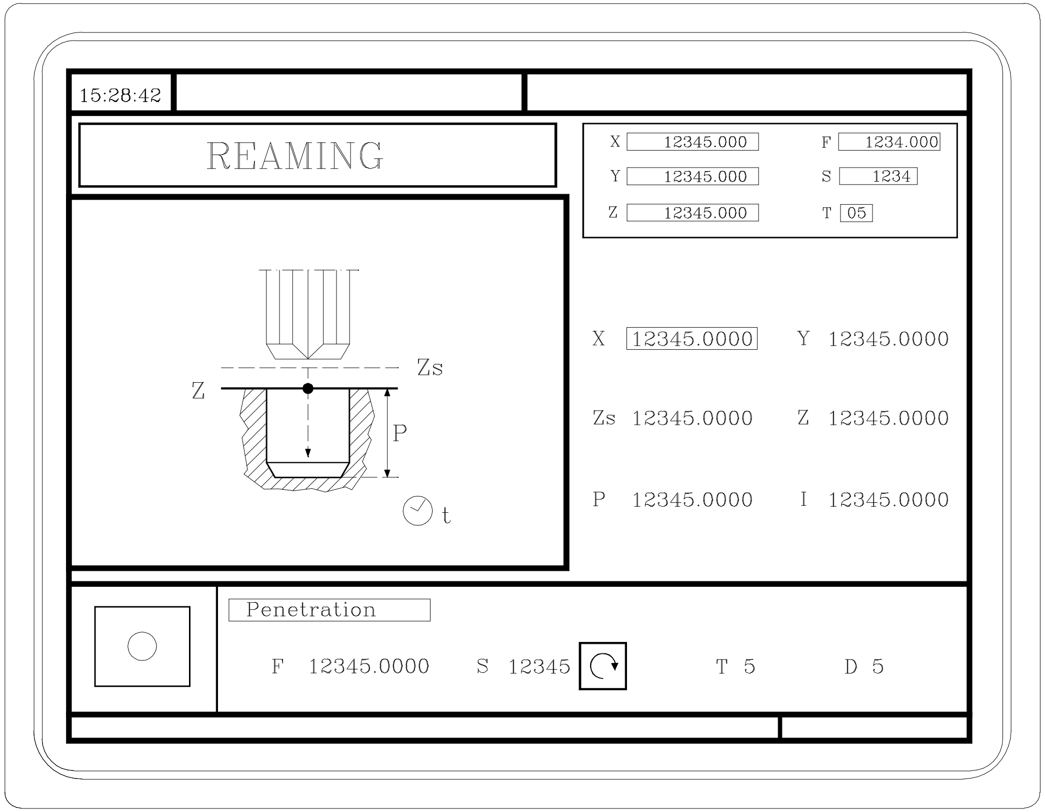
This operation may be carried out at the indicated position (X,Y) or may be repeated at different positions using the keys.




NOTE: Refer to the Operation Manual Chapter 4 Section 4.9

# 5.8 Reaming operation.

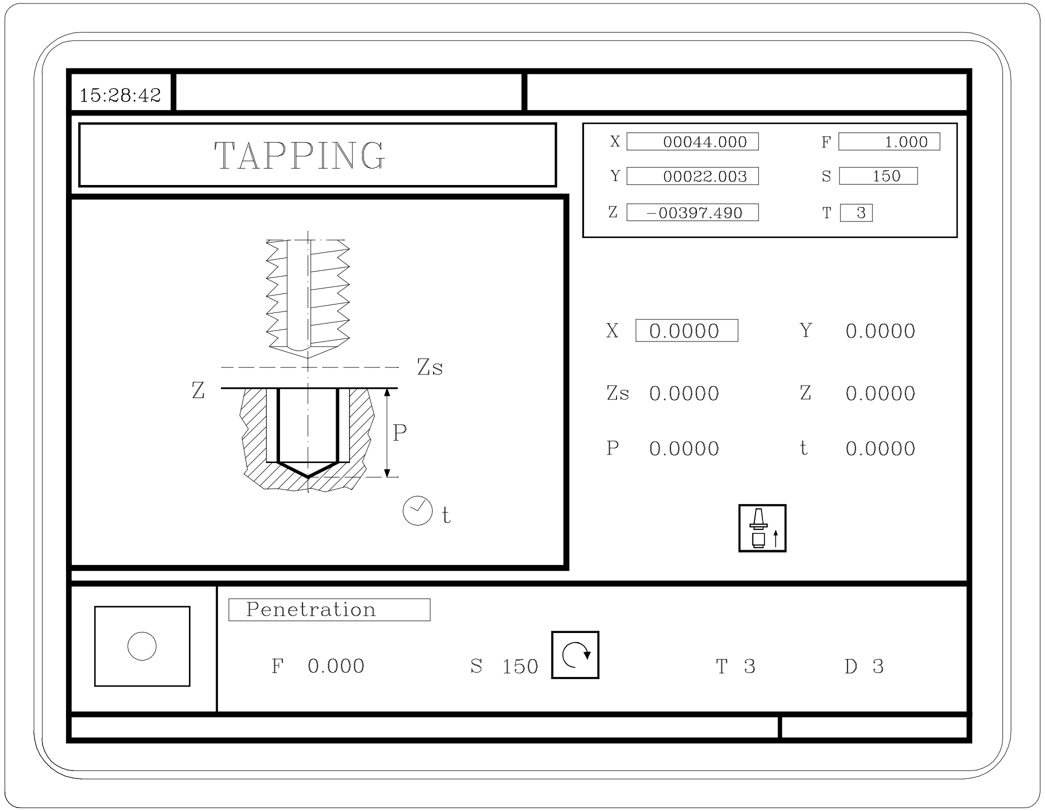
This operation may be carried out at the indicated position (X,Y) or may be repeated at different positions using the keys.




NOTE: Refer to the Operation Manual Chapter 4 Section 4.10

# 5.9 Threading operation.

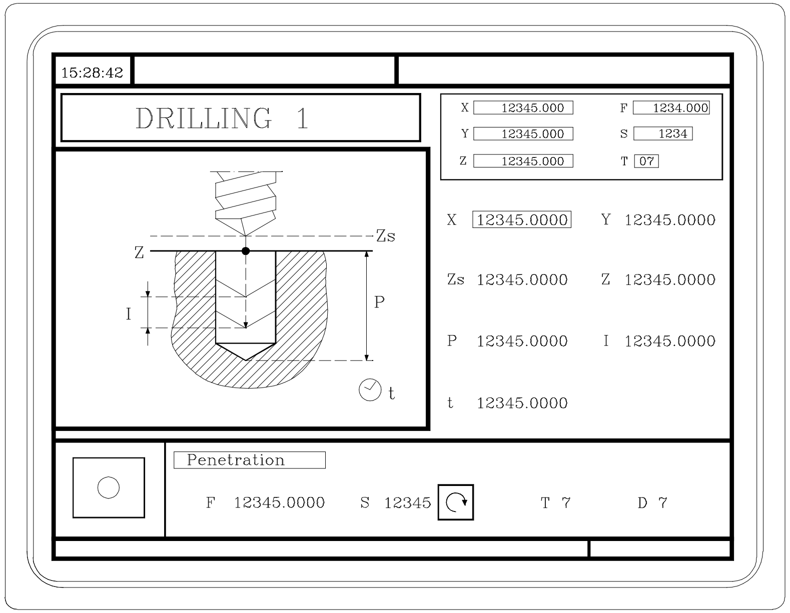
This operation may be carried out at the indicated position (X,Y) or may be repeated at different positions using the keys.




NOTE: Refer to the Operation Manual Chapter 4 Section 4.11

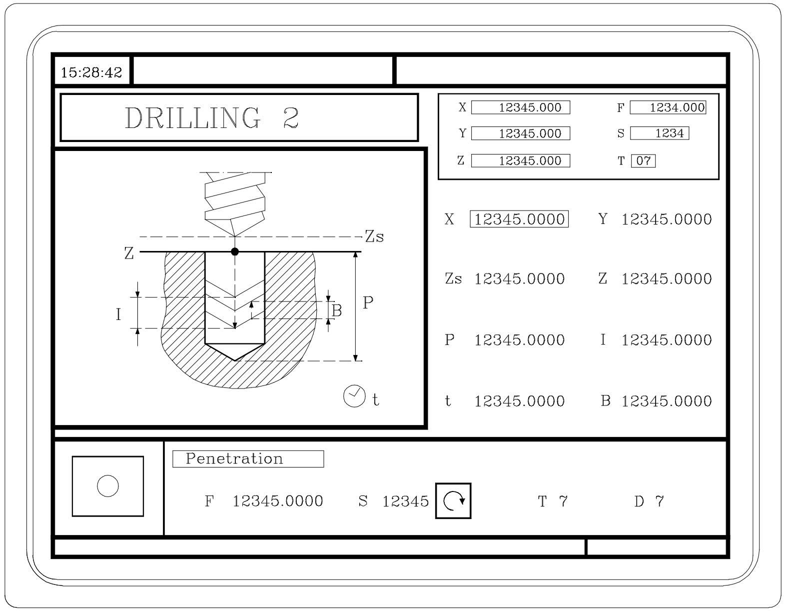
# 5.10 Drilling and Center punching operations.

These operations may be carried out at the indicated position (X,Y) or may be repeated at different positions using the keys.




LEVEL CYCLE

Drilling.

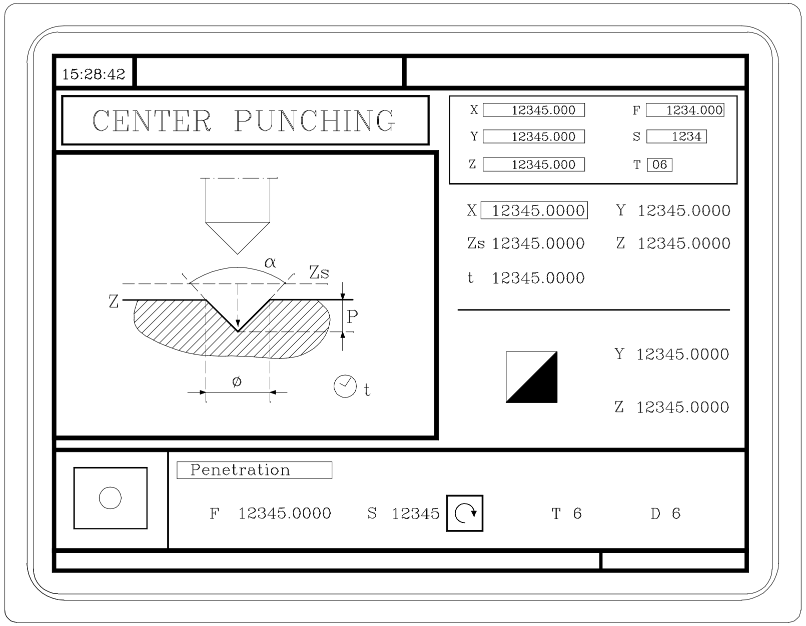



LEVEL CYCLE

Drilling.

At this cycle level, one programs the distance the tool withdraws after each penetration (drilling peck).

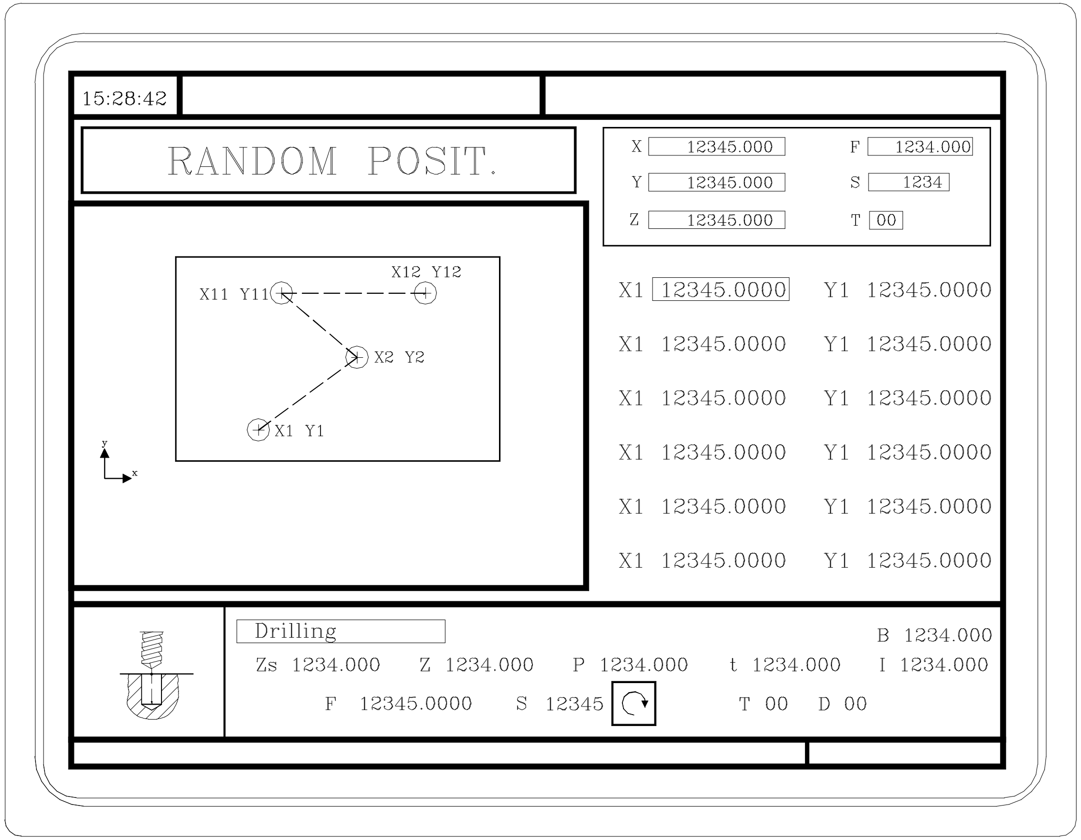
NOTE: Refer to the Operation Manual Chapter 4 Section 4.12




Center punching.

NOTE: Refer to the Operation Manual Chapter 4 Section 4.12

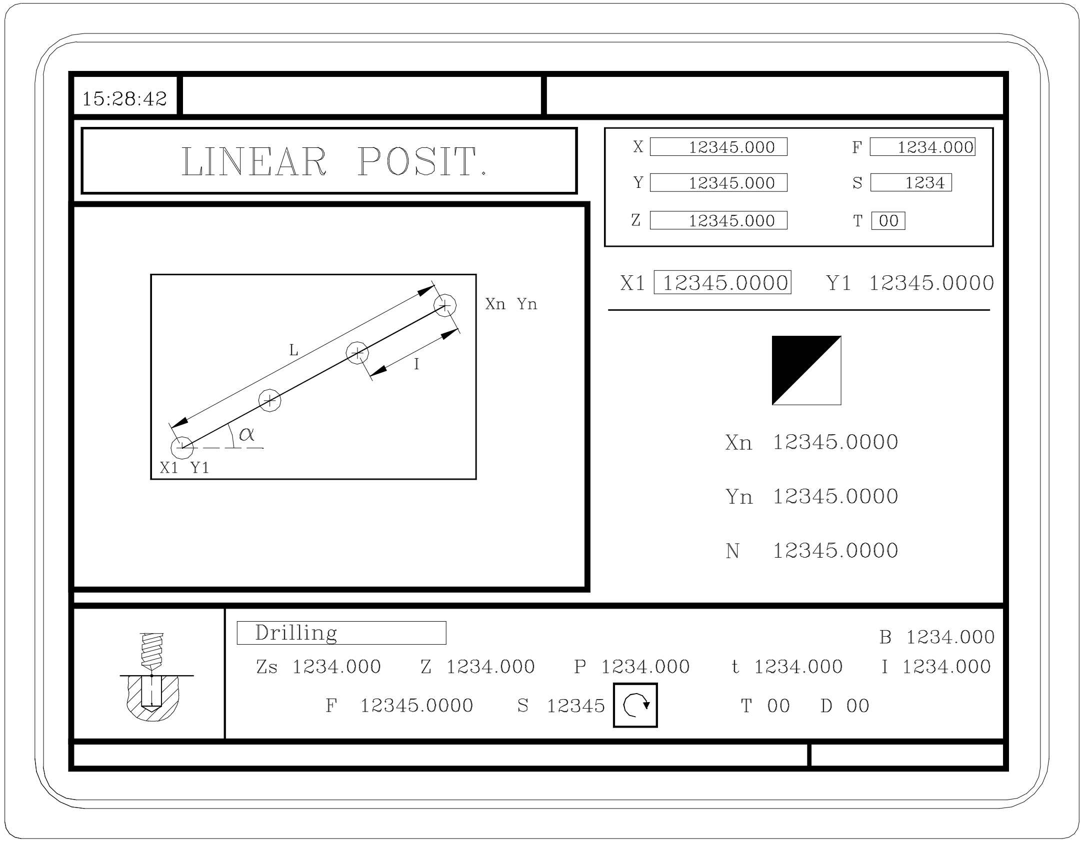
# 5.11 Multiple positioning at several points.




Only for Boring, Reaming, Drilling and Center punching operations.

NOTE: Refer to the Operation Manual Chapter 4 Section 4.13.1

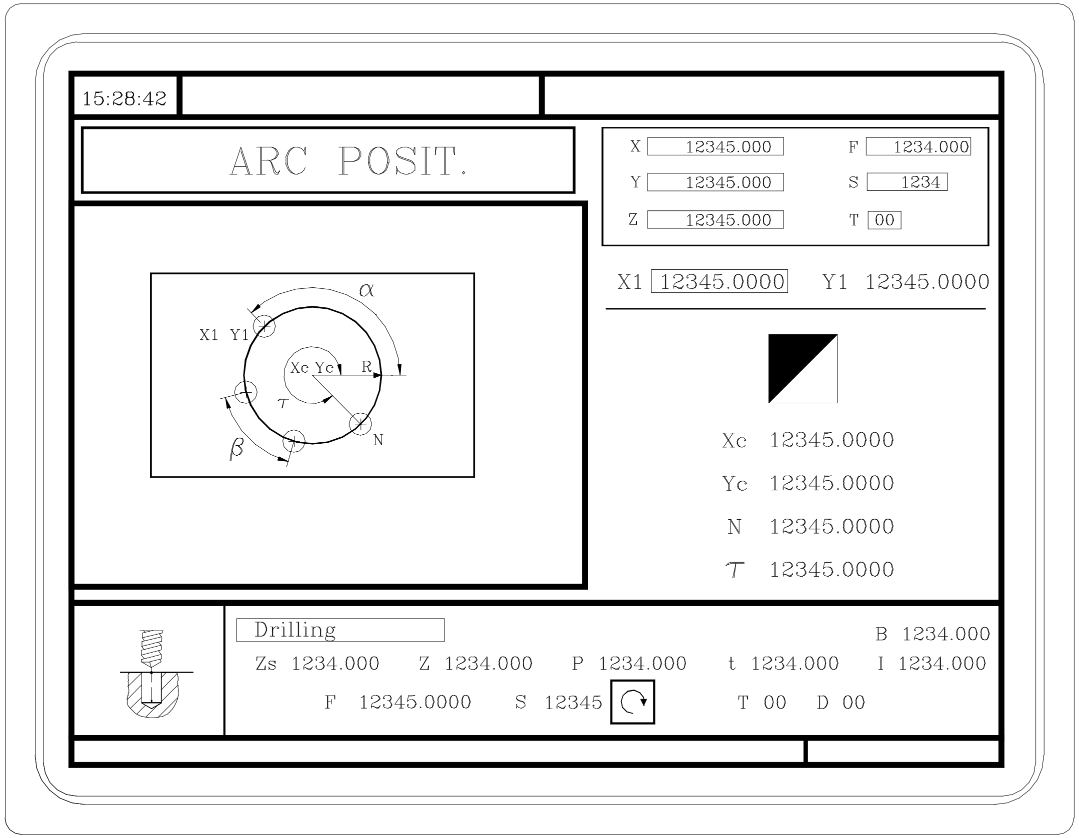
# 5.12 Multiple positioning in a straight line.




Only for Boring, Reaming, Drilling and Center punching operations.

NOTE: Refer to the Operation Manual Chapter 4 Section 4.13.2

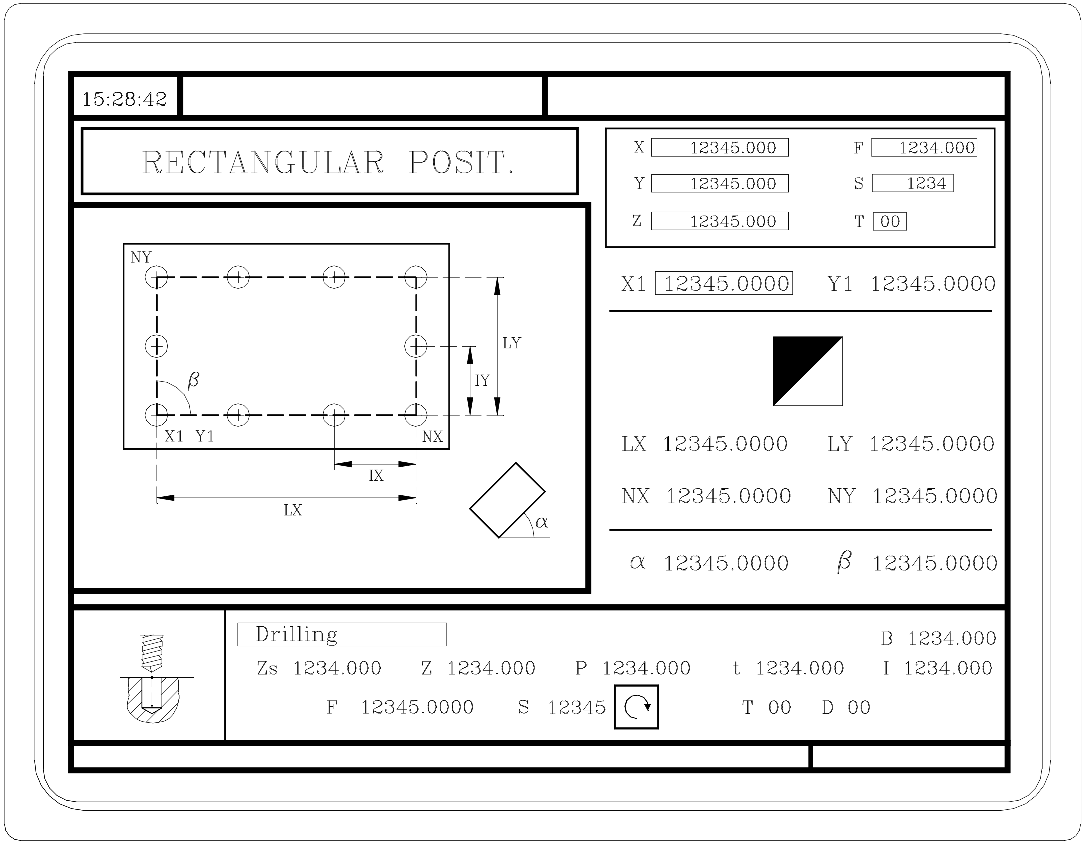
# 5.13 Multiple positioning in an arc.




Only for Boring, Reaming, Drilling and Center punching operations.

NOTE: Refer to the Operation Manual Chapter 4 Section 4.13.3

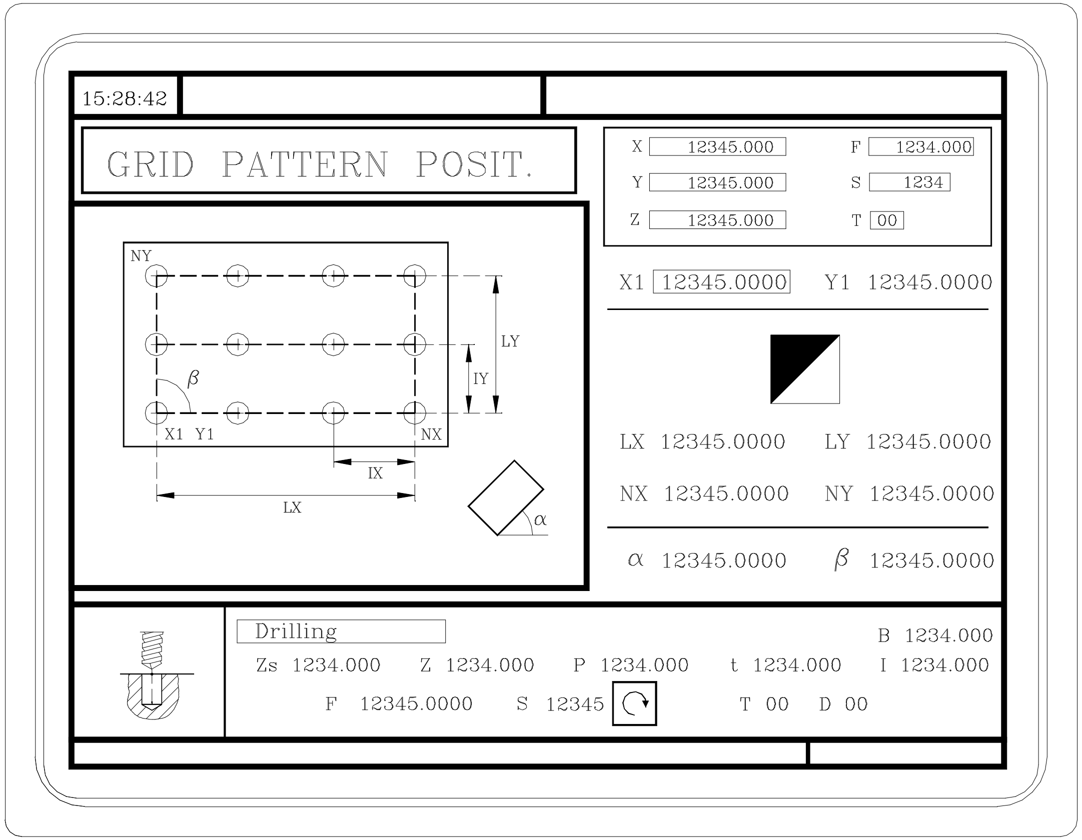
# 5.14 Multiple positioning in a parallelogram pattern.




Only for Boring, Reaming, Drilling and Center punching operations.

NOTE: Refer to the Operation Manual Chapter 4 Section 4.13.4

# 5.15 Multiple positioning in a grid pattern.

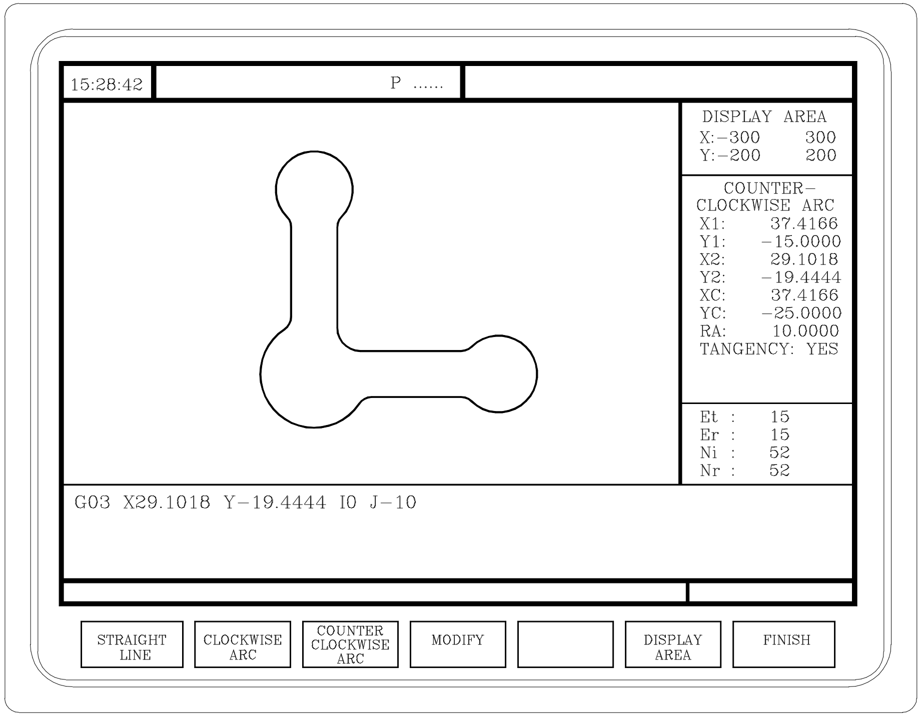



Only for Boring, Reaming, Drilling and Center punching operations.

NOTE: Refer to the Operation Manual Chapter 4 Section 4.13.5

# 5.16 Profile editor.

With the profile editor it is possible to define straight and circular sections of the profile (the editor solves the intersection and tangency problems) and then modify those sections by adding rounded corners, chamfers as well as tangential entries and exits.




It is used to define the “Profile milling” cycle and the “Pocket with profile” cycle.

Chapter 6

# Conversational part-programs

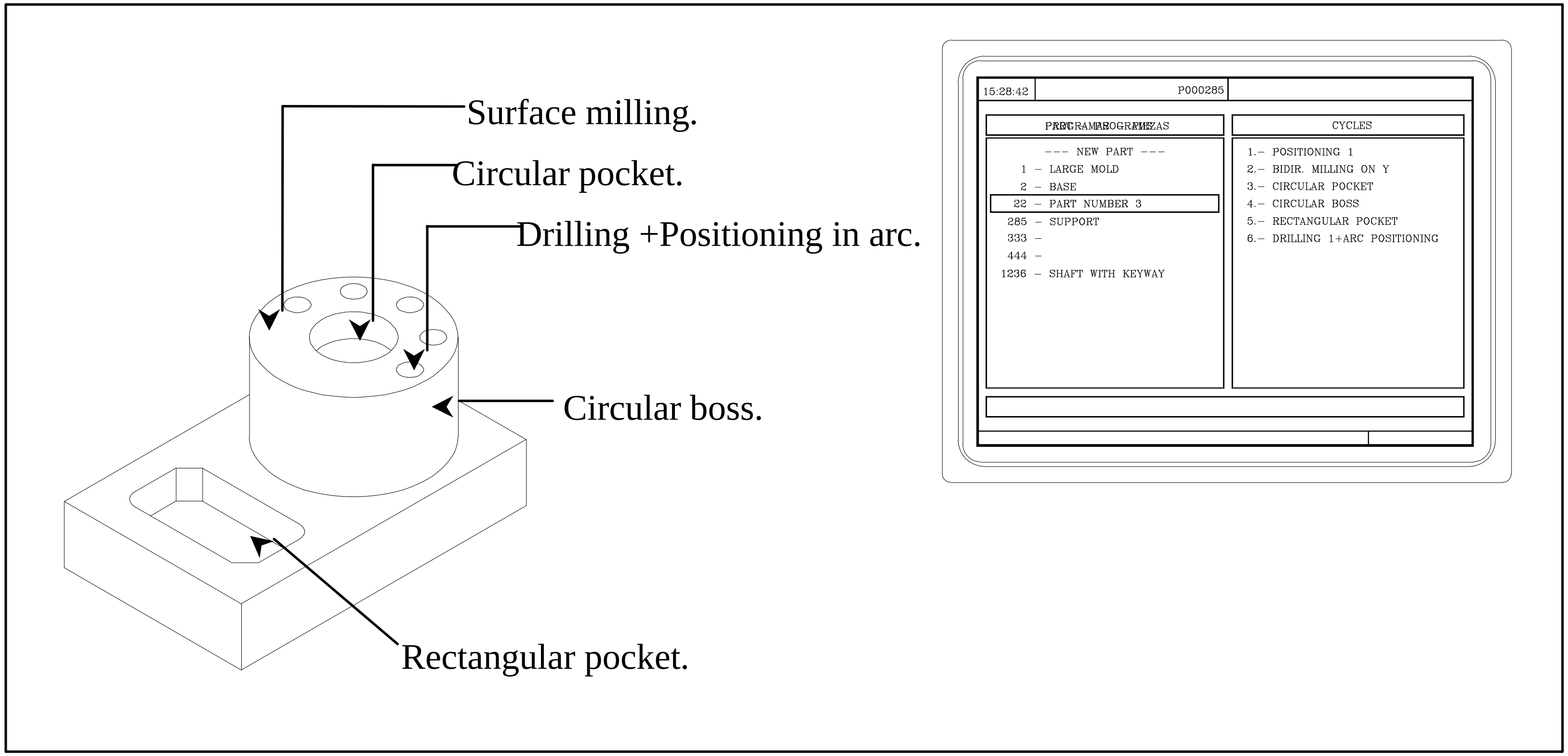
This chapter describes:

- 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?

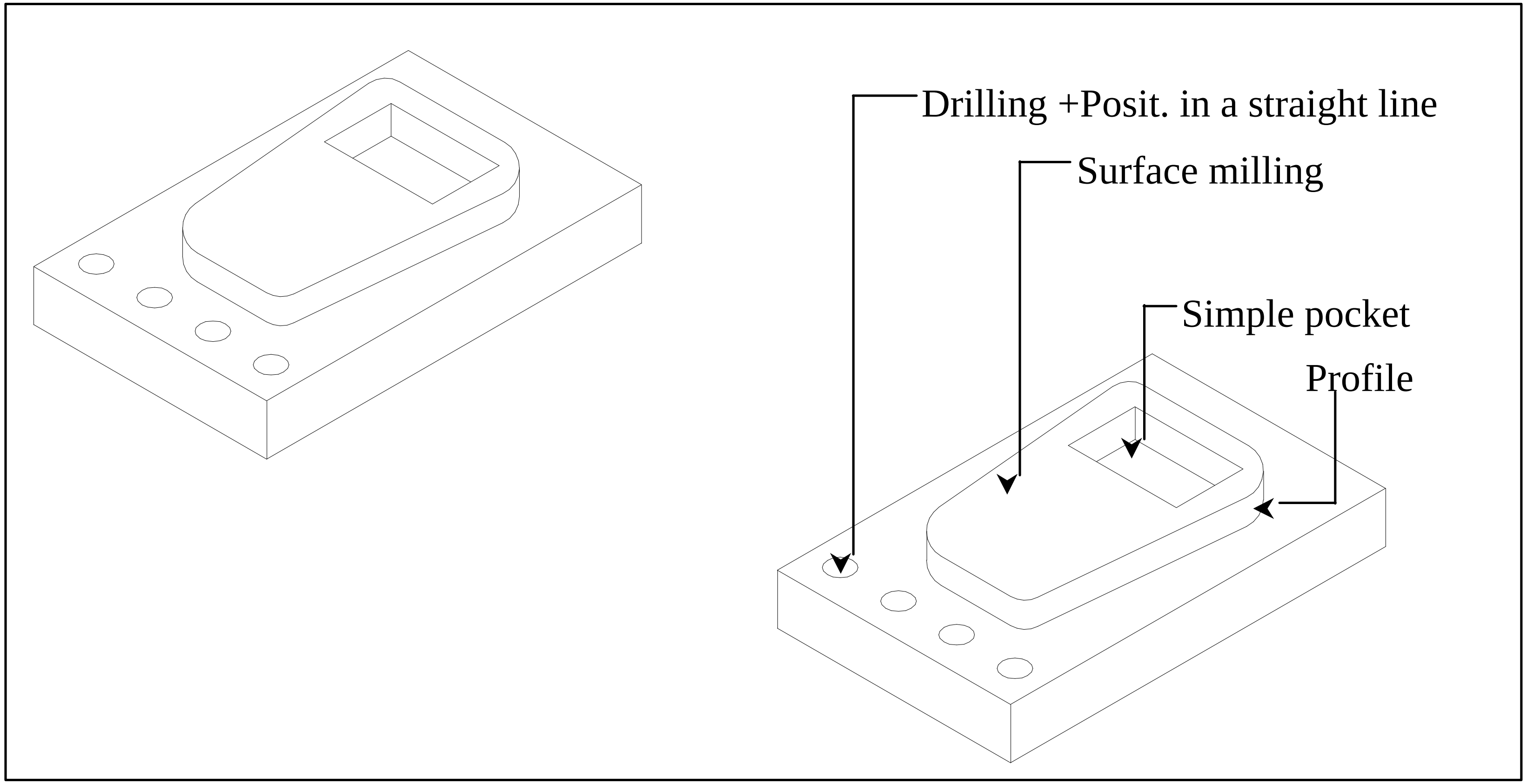
It is a set of operations ordered secuentially. Each operation is defined separately and they are then stored one after the other in a program.

The name of the part-program can be any integer between 1 - 899999.

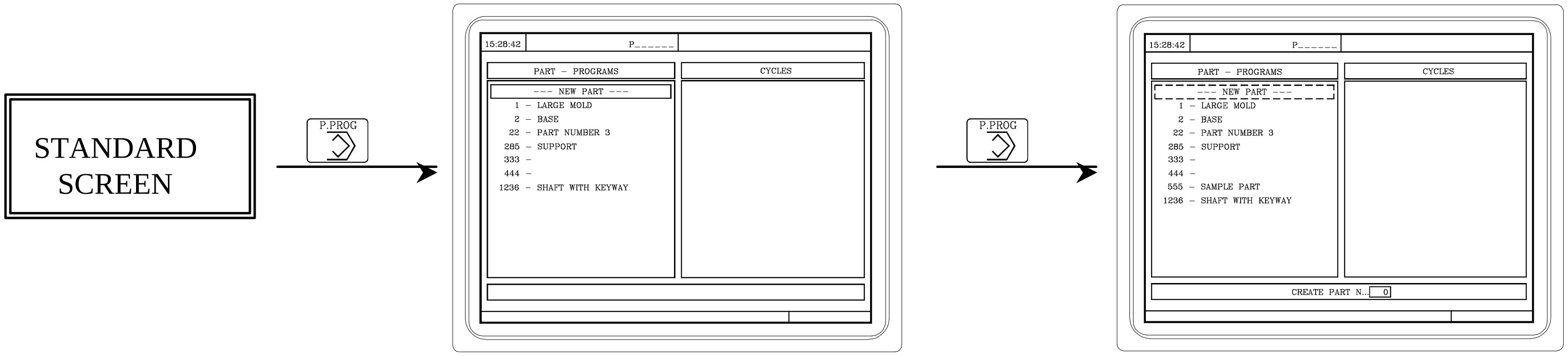


# 6.2 Edit a part-program.

To edit a part-program, we first choose the operations needed to execute the part. A part may be executed in various ways.

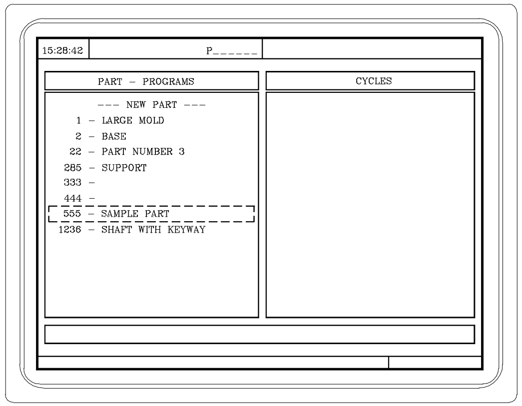


Once the sequence of operations has been chosen, the part-program is built by editing the operations one by one.




(Enter number) + ENTER + (Comment) + ENTER

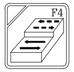
e.g. : <555> + ENTER + <SAMPLE PART> + ENTER



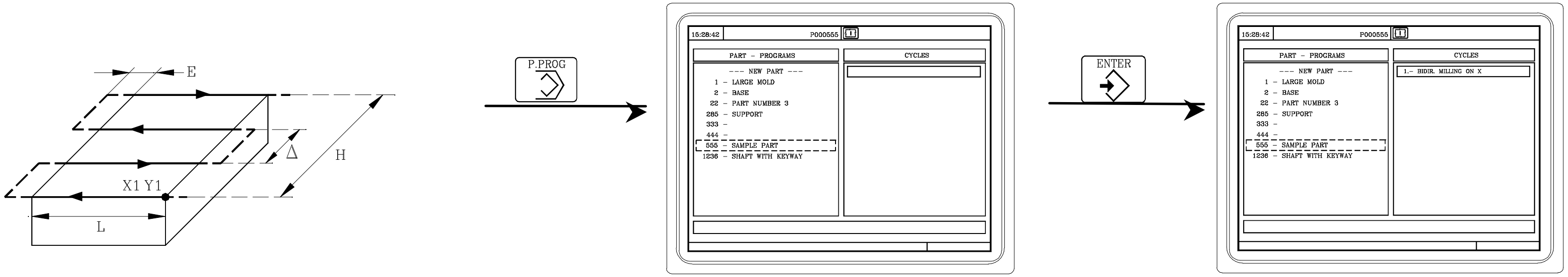

NOTE: The following keys are used:

: To move up and down on each column.

: To change columns.

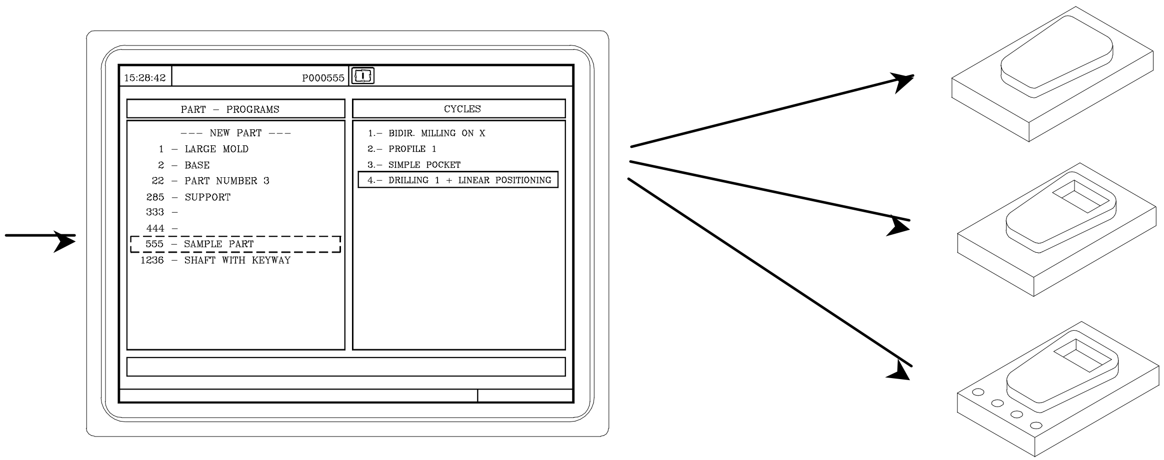


Choose the operation and define the parameters.




Repeat these steps with the other operations. In our case, the finished part-program will be:

Program number

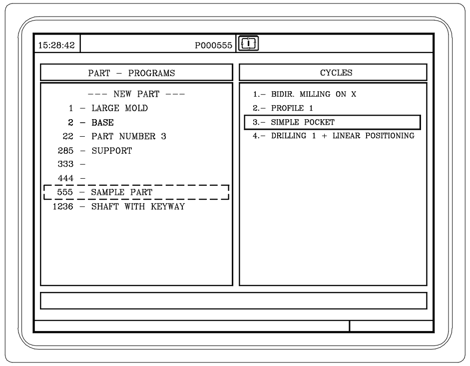


# 6.3 Modify a part-program.

The operations making up a part-program can be modified.

**REPLACE AN OPERATION**

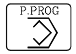
Choose operation

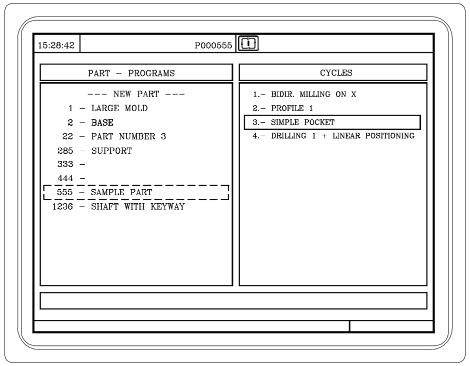


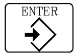


The CNC shows the cycle with all its data.

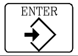
Modify the operation parameters like in the editing mode.




The CNC requests an option. Choose REPLACE.




The new operation replaces the previous one.

NOTE: Refer to the Operation Manual Chapter 5 Section 5.6.4

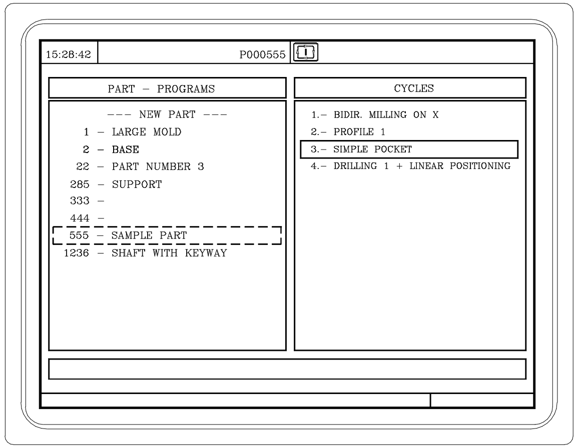
New operations can also be inserted into a part-program.

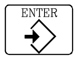
## INSERT AN OPERATION

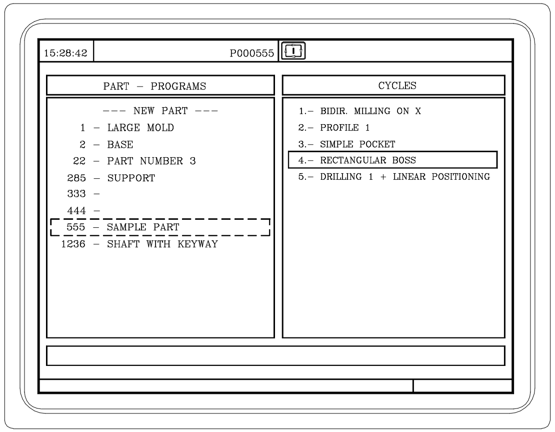
Choose operation

Define the parameters and cutting conditions of the operation to be inserted. Press P.PROG

Choose position



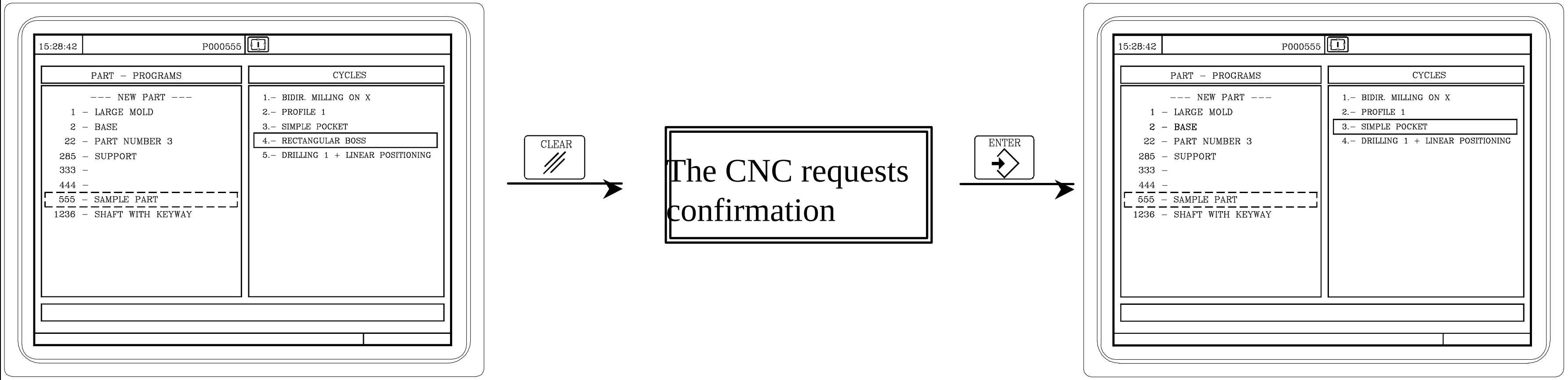

The new operation is inserted after the chosen position.

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.

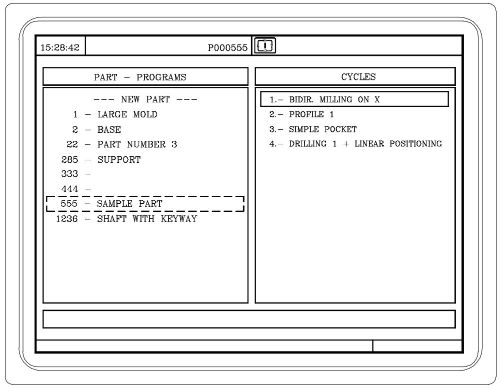



NOTE: Refer to the Operation Manual Chapter 5 Section 5.6.1

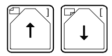
The position of an operation can also be changed.

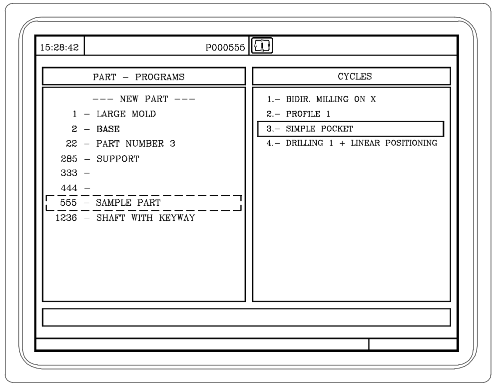
## CHANGE THE POSITION OF AN OPERATION

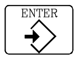
Select, on the right column, the operation to be moved.

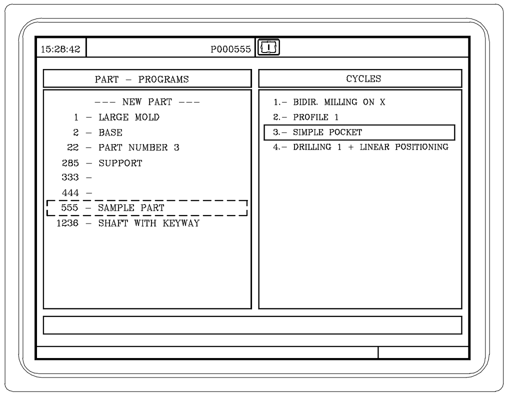



Select the new position.





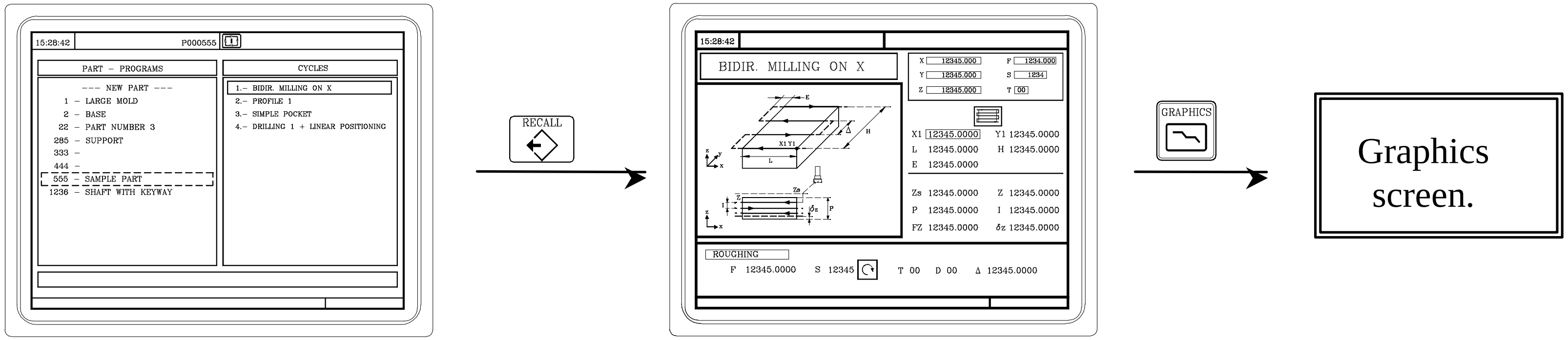




The operation is inserted behind the operation occupying that position.

NOTE: Refer to the Operation Manual Chapter 5 Section 5.6.2

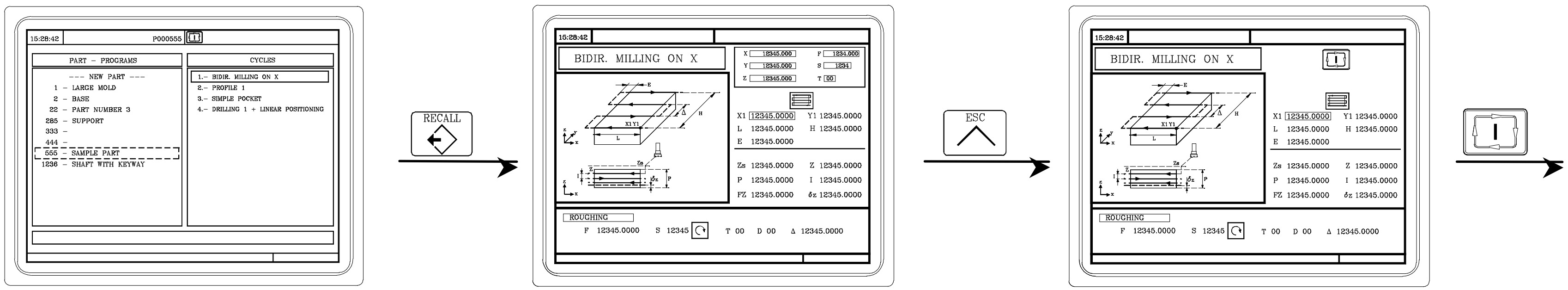
# 6.4 Simulate/execute an operation.

Select, on the right column, the operation to be SIMULATED:




More information about the graphics screen in chapter 4.3.2 of this manual.

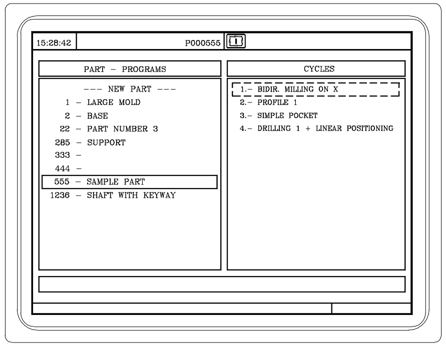
Select, on the right column, the operation to be EXECUTED:

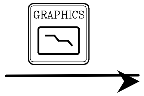



NOTE: Refer to the Operation Manual  Chapter 6   Section 6.3

# 6.5 Simulate/execute a part-program.

Select, on the left column, the part-program to be SIMULATED:

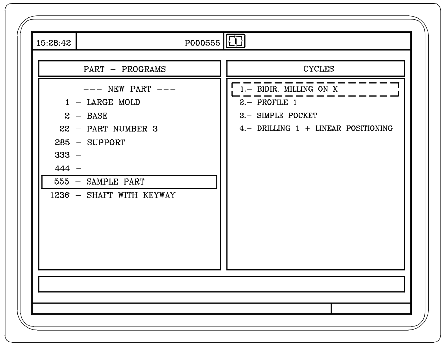


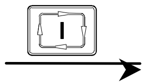




Graphics screen

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:

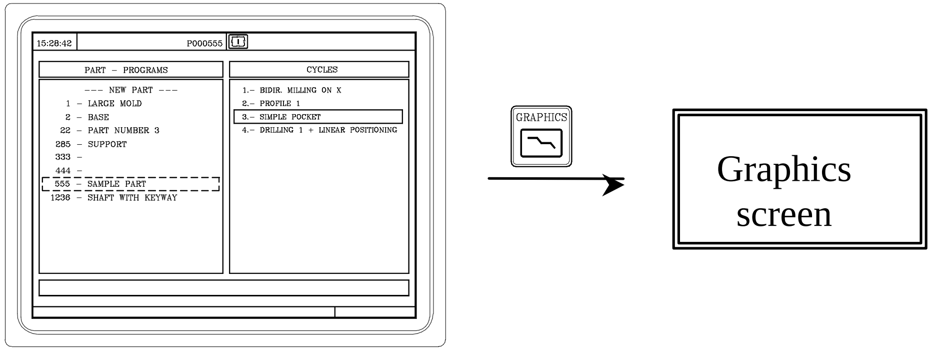


NOTE: Refer to the Operation Manual Chapter 6 Section 6.2

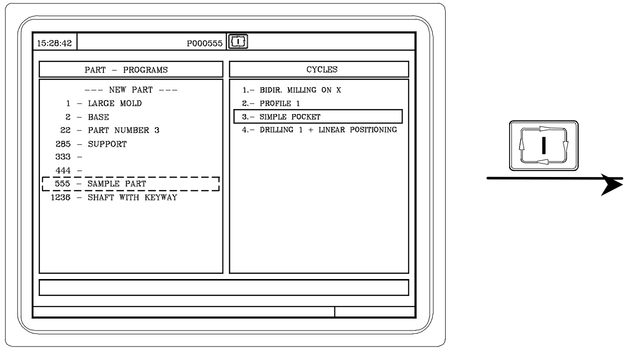
# 6.6 Simulate/execute starting at a particular operation.

Select, on the right column, the operation where the SIMULATION is to be started:




More information about the graphics screen in chapter 4.3.2 of this manual.

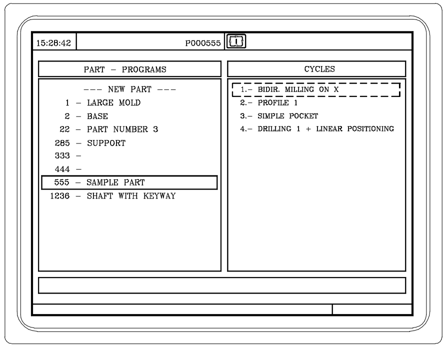
Select, on the right column, the operation where the EXECUTION is to be started:

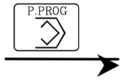


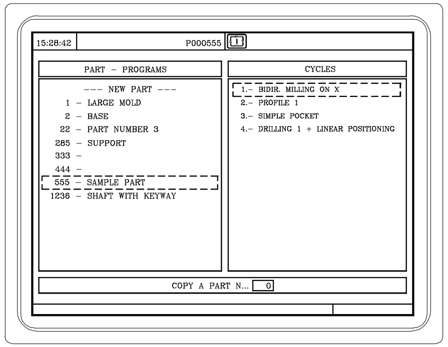

NOTE: Refer to the Operation Manual  Chapter 6   Section 6.2.1

# 6.7 Copy a part-program into another one.

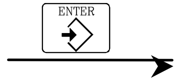
Select, on the left column, the part-program to be COPIED:



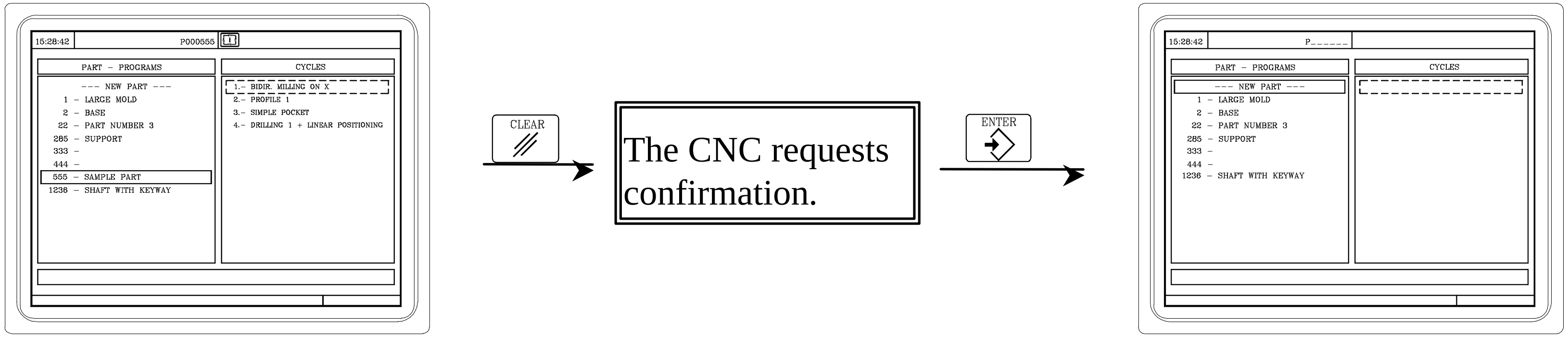

Key in the number and comment of the new program.




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:

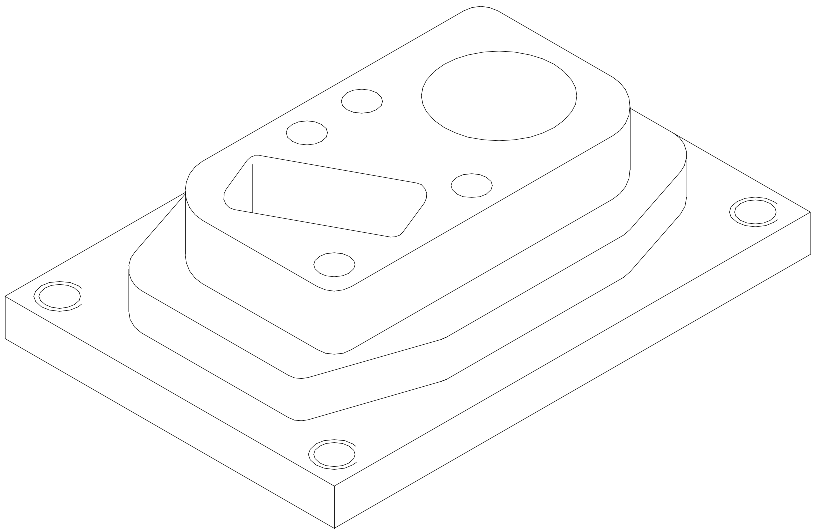


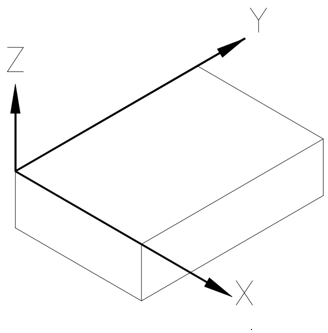

NOTE: Refer to the Operation Manual Chapter 5 Section 5.4

Appendix I

# Programming example

## Step 0: Part to be machined.






### INITIAL CONSIDERATIONS

This chapter shows an example of how to create a part-program.

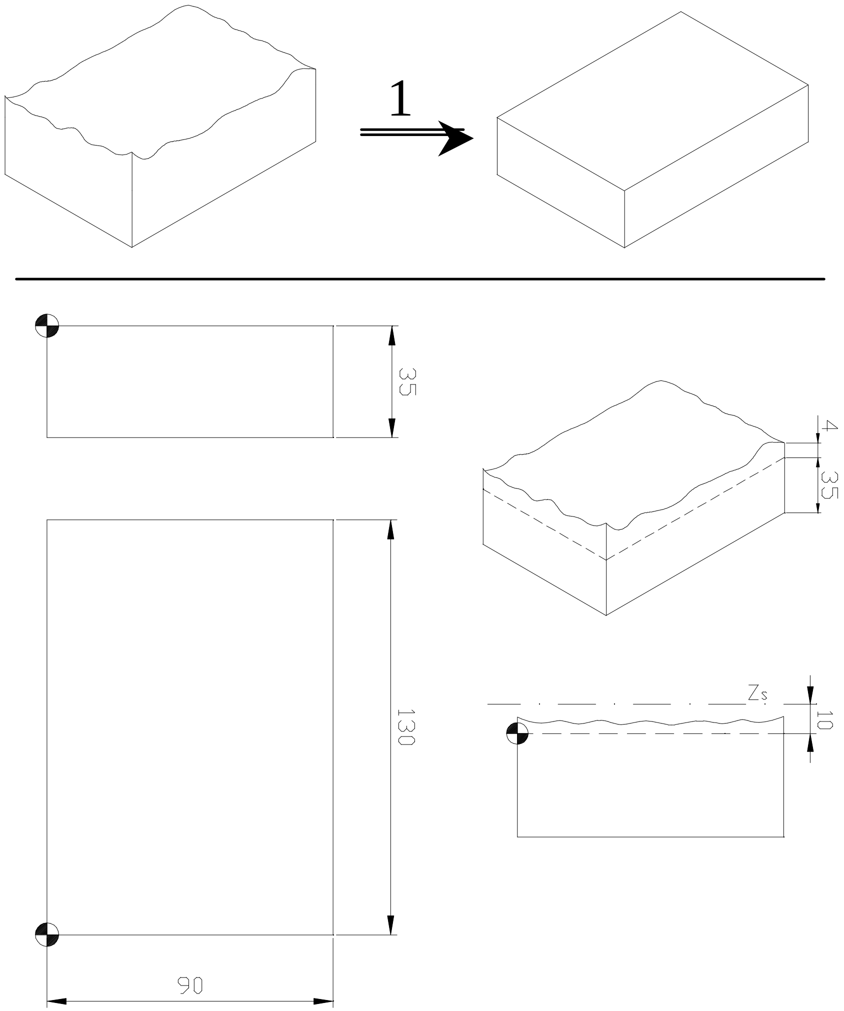
Remember that the tool number may be different depending on the machine. The tool used in this example are:

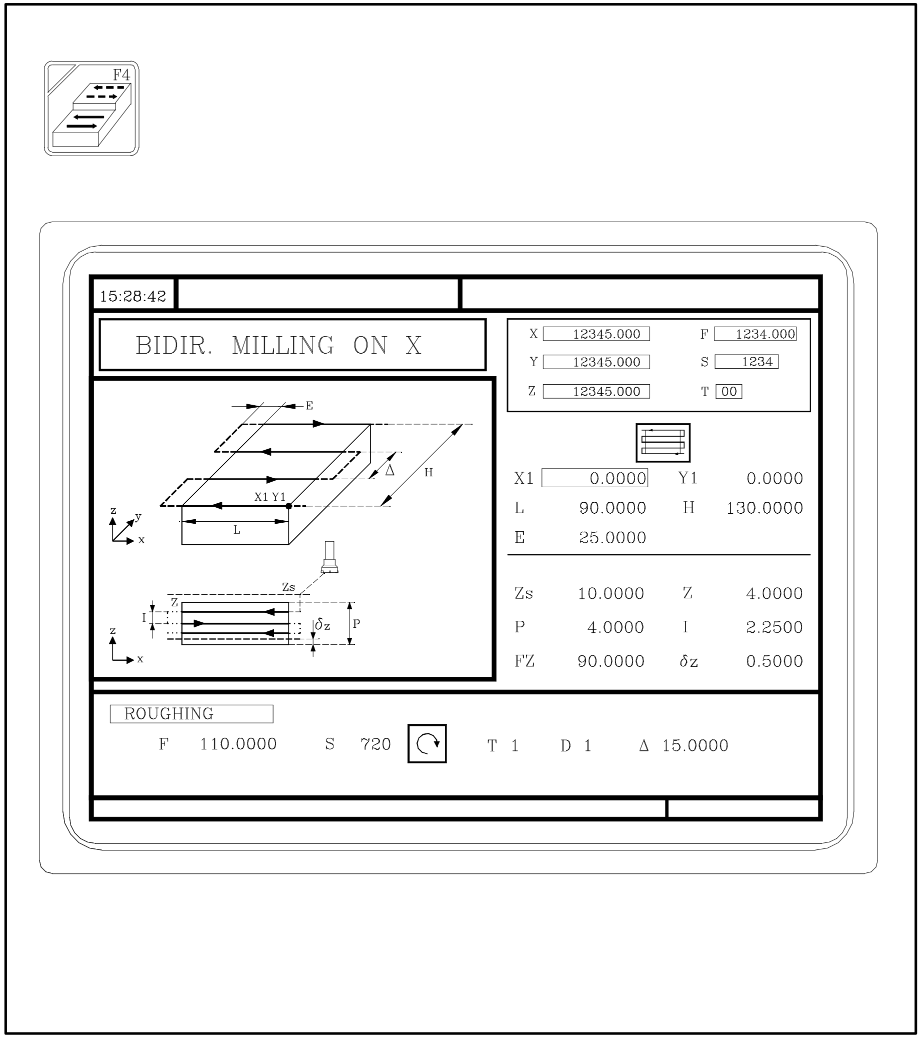
| | |
|---|---|
| T1: ? 40 endmill. | T5: ? ??drill |
| T2: ? 25 endmill. | T6: ? 5 drill. |
| T3: ? 10 endmill. | T7: M-6 tap. |
| T4: Center punch. | |

The spindle speed and axis feedrates are orientative and they may be other than the ones shown here.

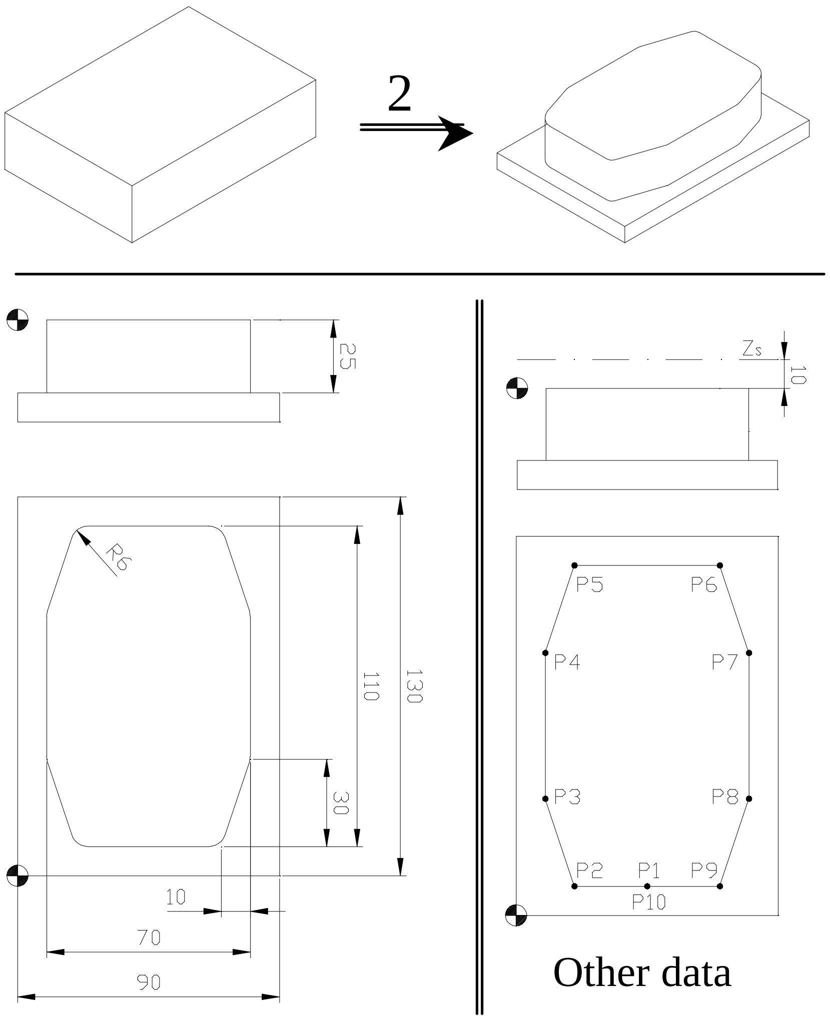
The “Part zero” position is represented here by the symbol.

## Step 1: Surface milling.

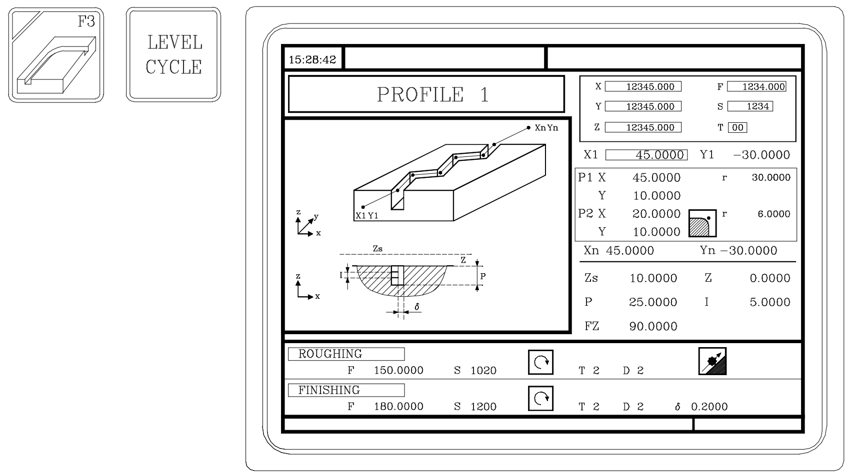


## Step 2: Machining the profile.

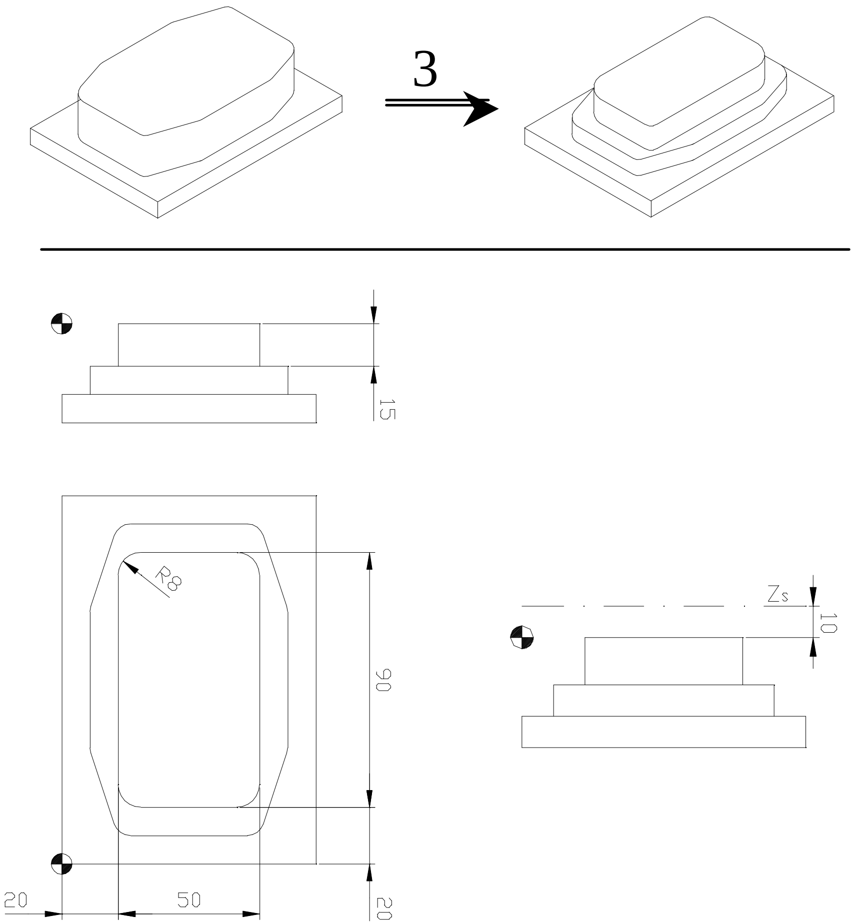


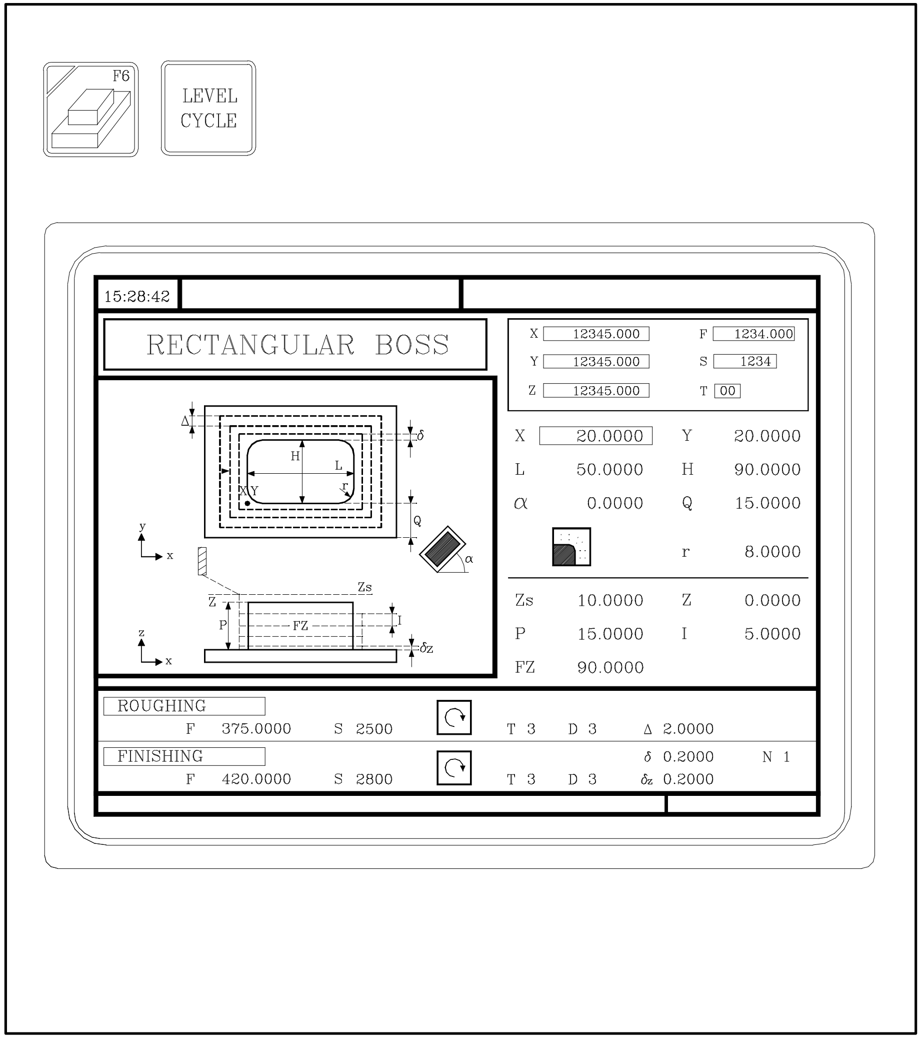

Other data



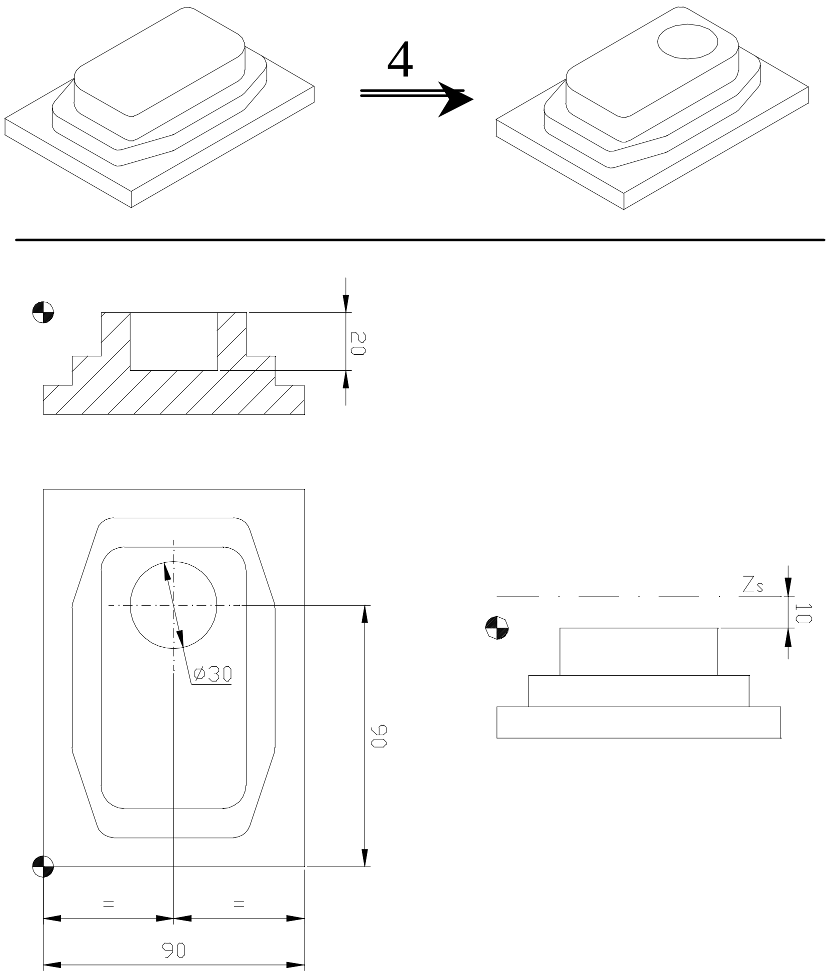

| | | | |
|---|---|---|---|
| P1 | X 45.0000 | Y 10.0000 | r 30.0000 |
| P2 | X 20.0000 | Y 10.0000 | r 6.0000 |
| P3 | X 10.0000 | Y 40.0000 | r 6.0000 |
| P4 | X 10.0000 | Y 90.0000 | r 6.0000 |
| P5 | X 20.0000 | Y 120.0000 | r 6.0000 |
| P6 | X 70.0000 | Y 120.0000 | r 6.0000 |
| P7 | X 80.0000 | Y 90.0000 | r 6.0000 |
| P8 | X 80.0000 | Y 40.0000 | r 6.0000 |
| P9 | X 70.0000 | Y 10.0000 | r 6.0000 |
| P10 | X 45.0000 | Y 10.0000 | r 6.0000 |
| P11 | X 45.0000 | Y 10.0000 | r 6.0000 |
| P12 | X 0.0000 | Y 0.0000 | r 30.0000 |

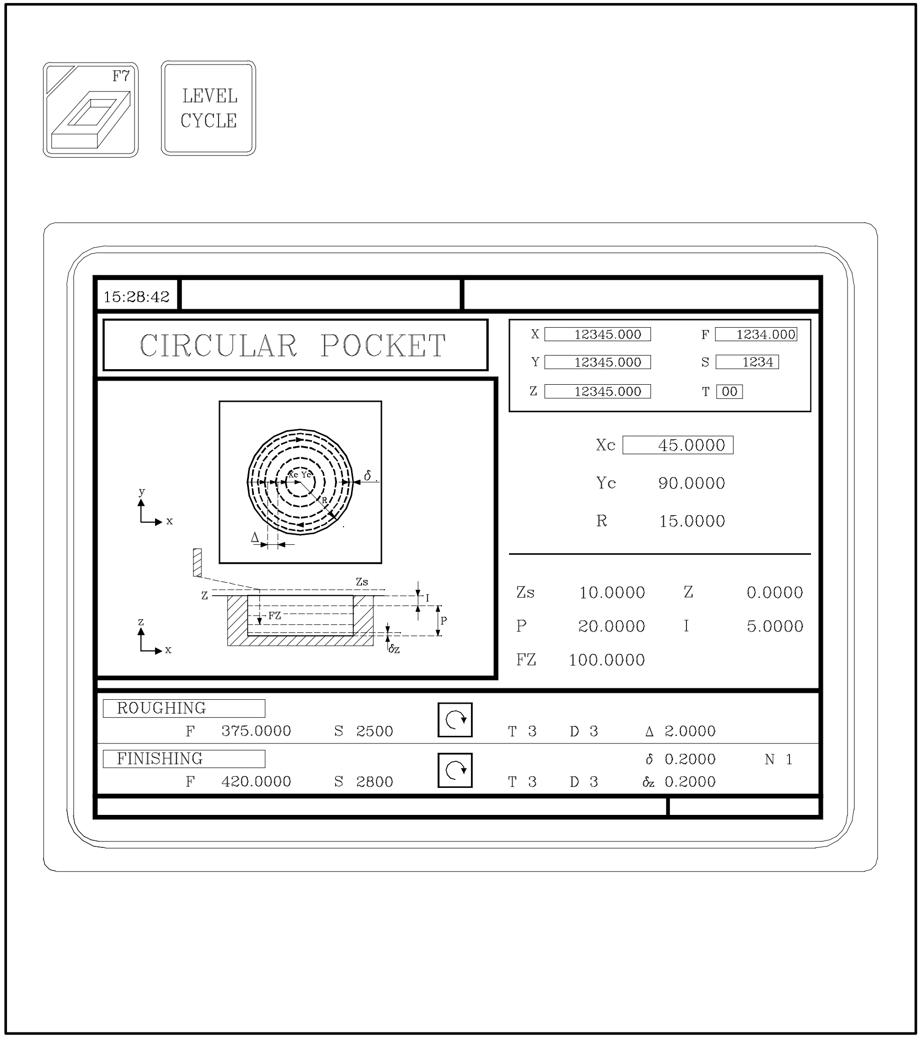
# Step 3: Rectangular boss.



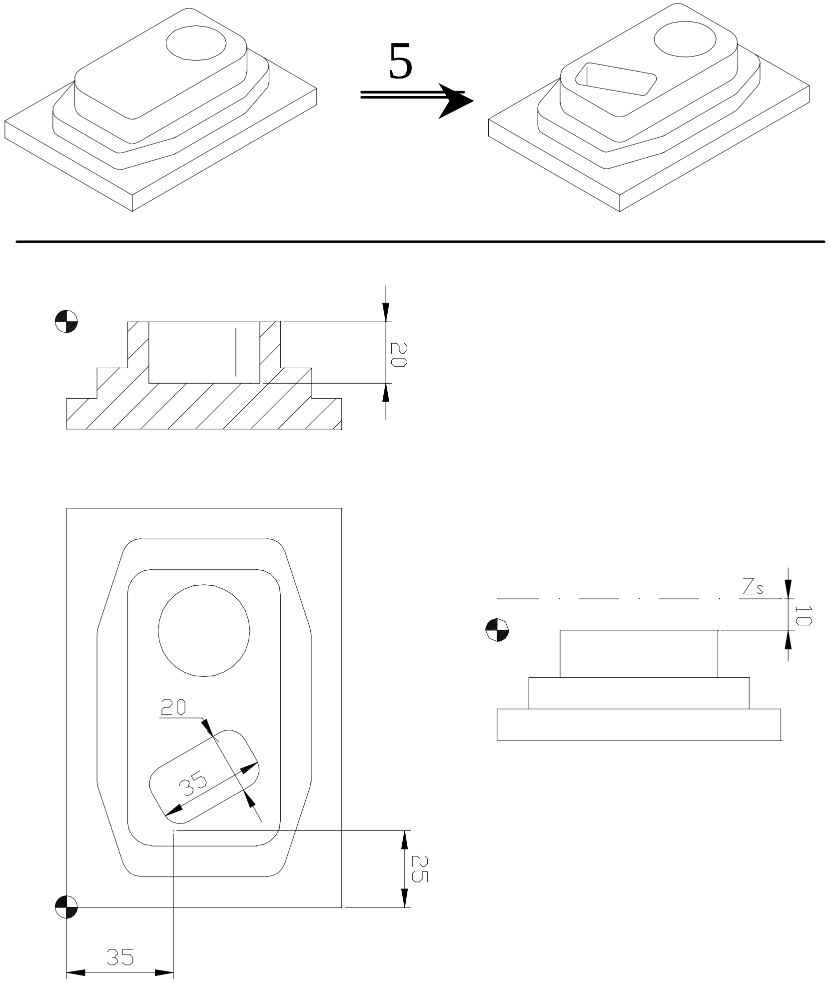



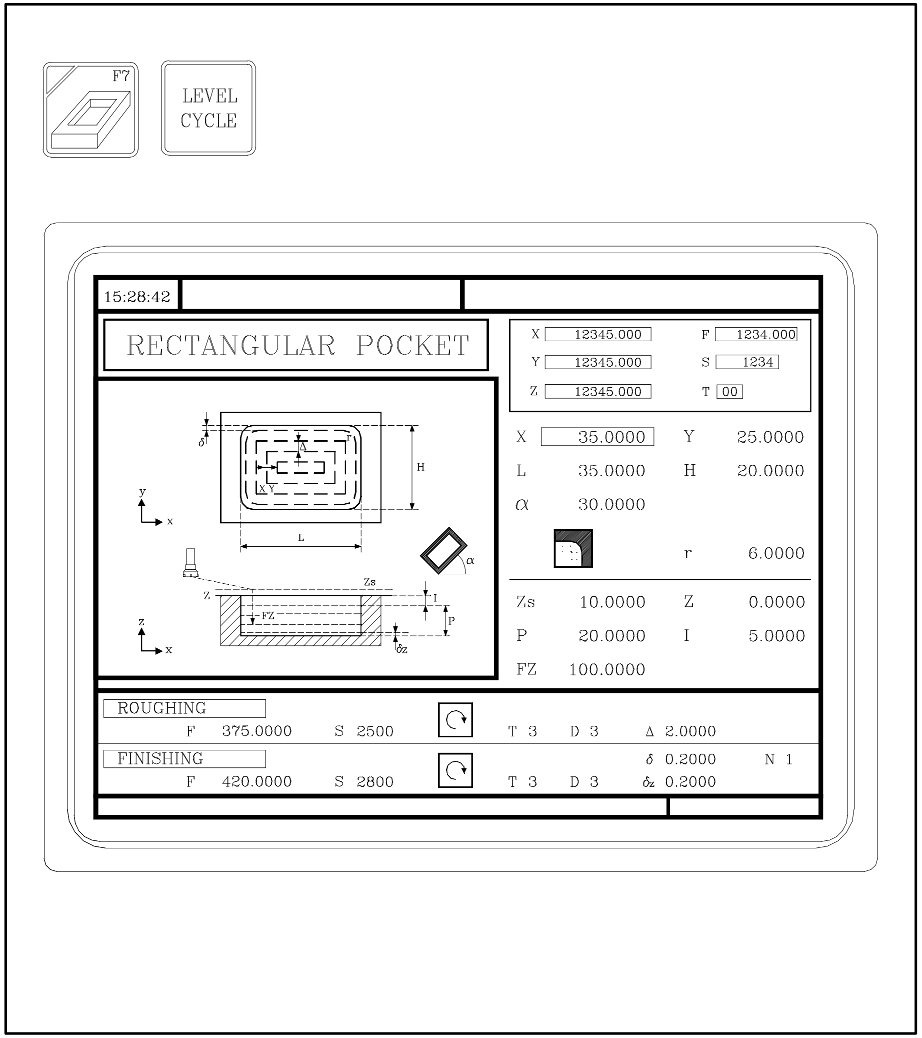
## Step 4: Circular pocket.



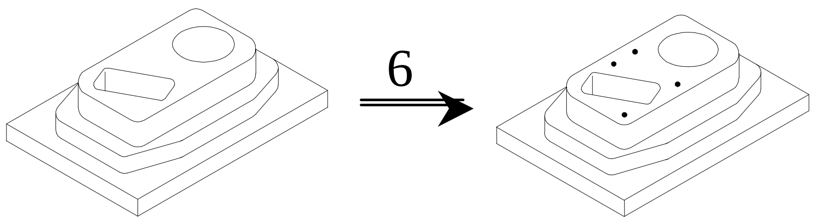



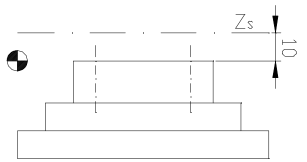
# Step 5: Rectangular pocket.

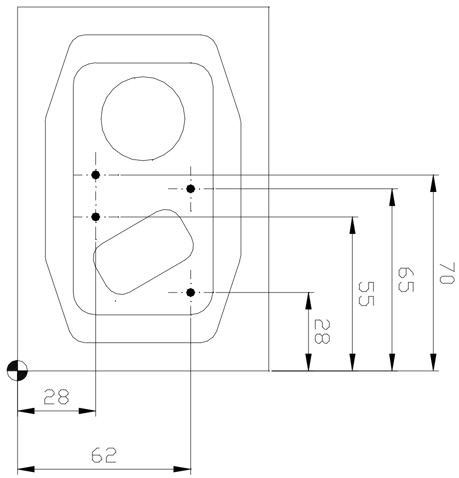


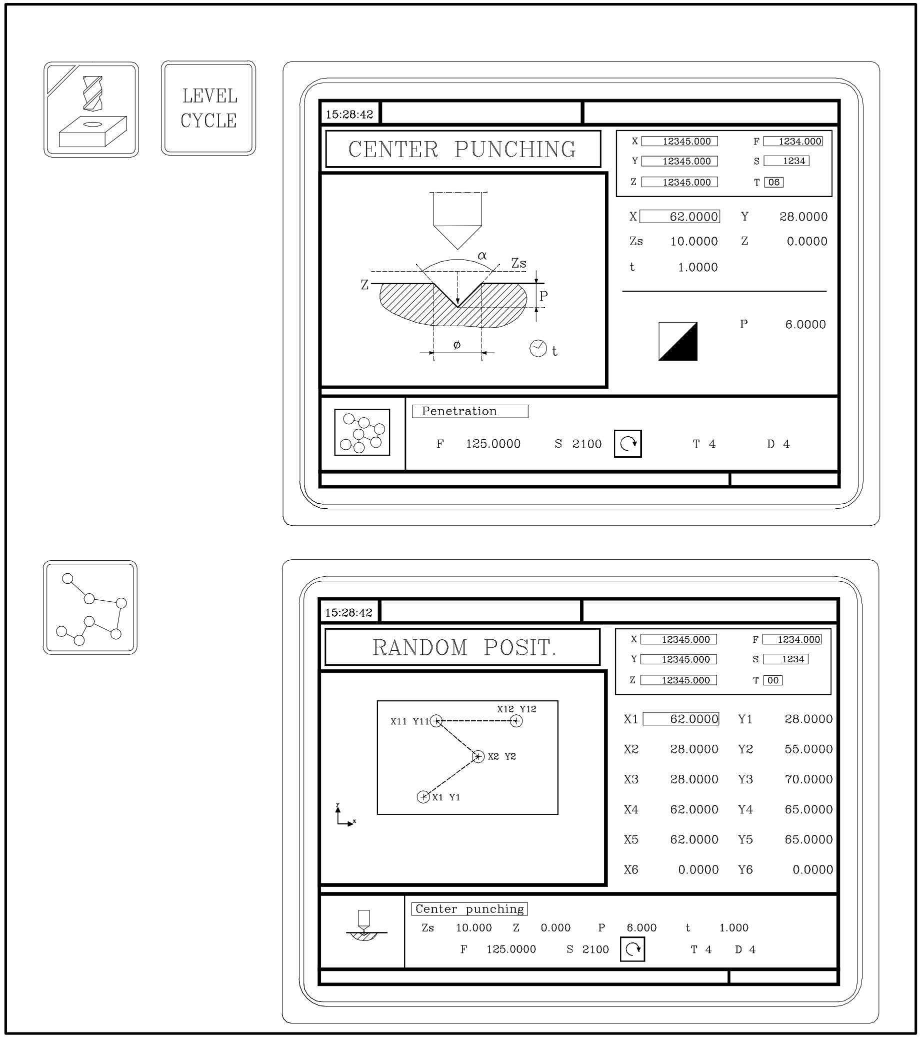



# Step 6: Center punching + Multiple positioning at several points.

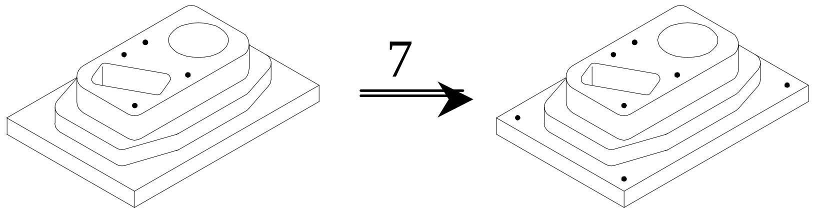


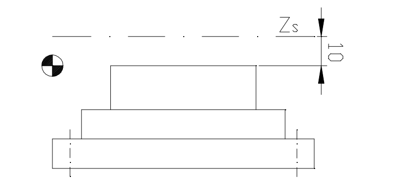



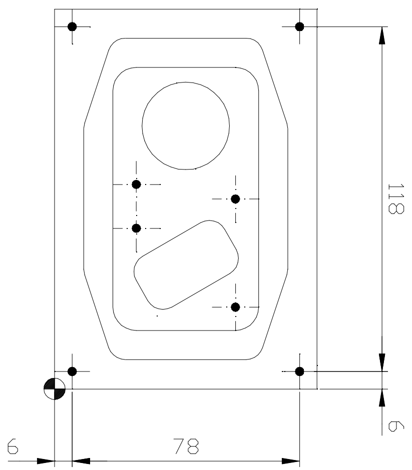



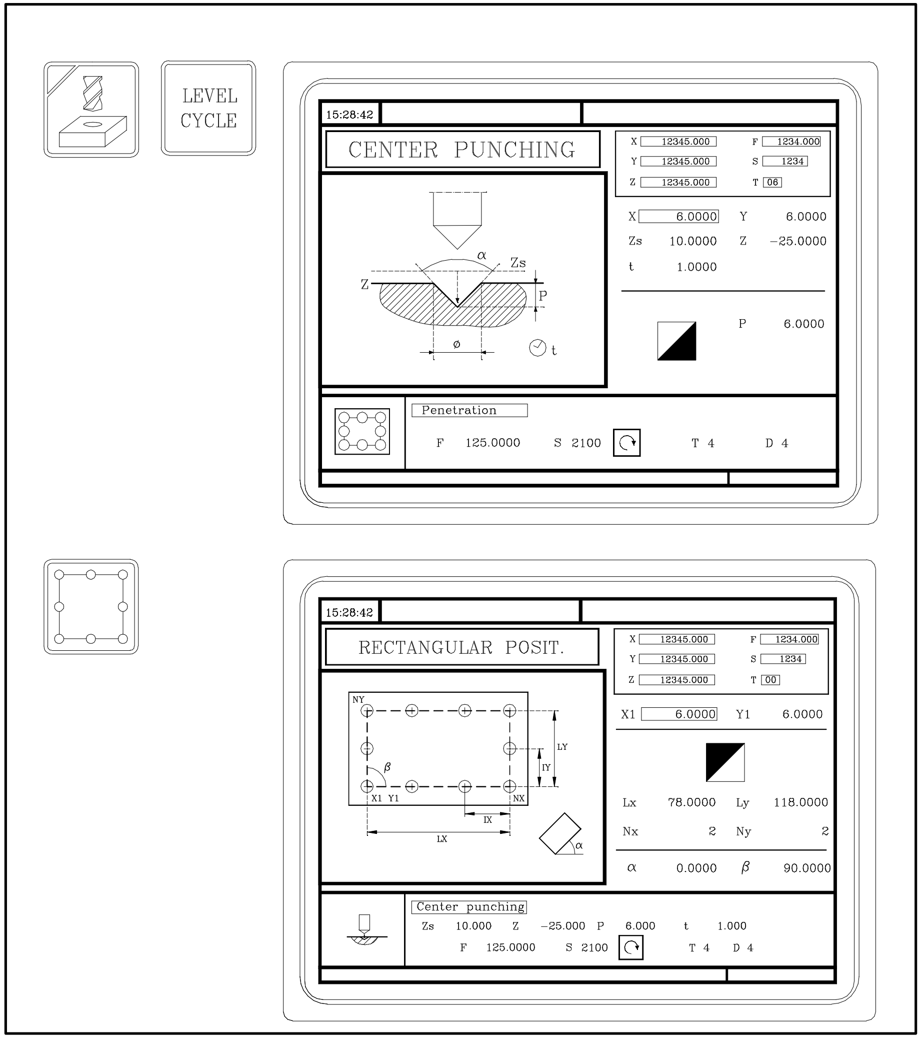



# Step 7: Center punching + Multiple positioning in parallelogram pattern.

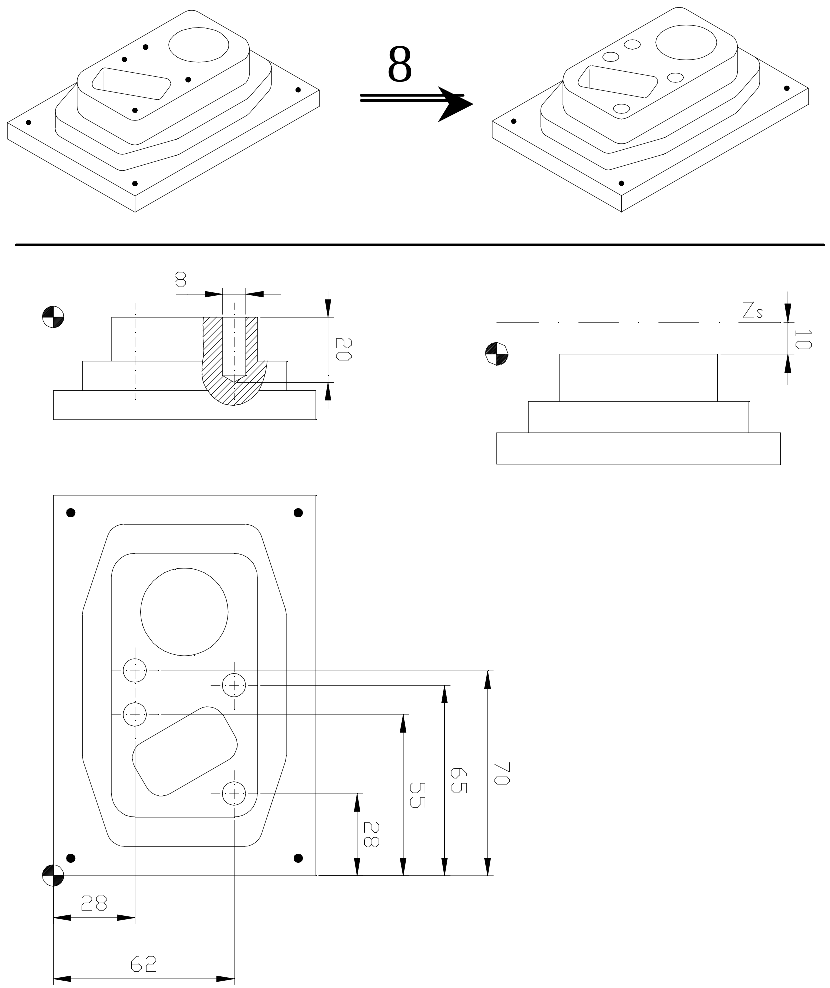


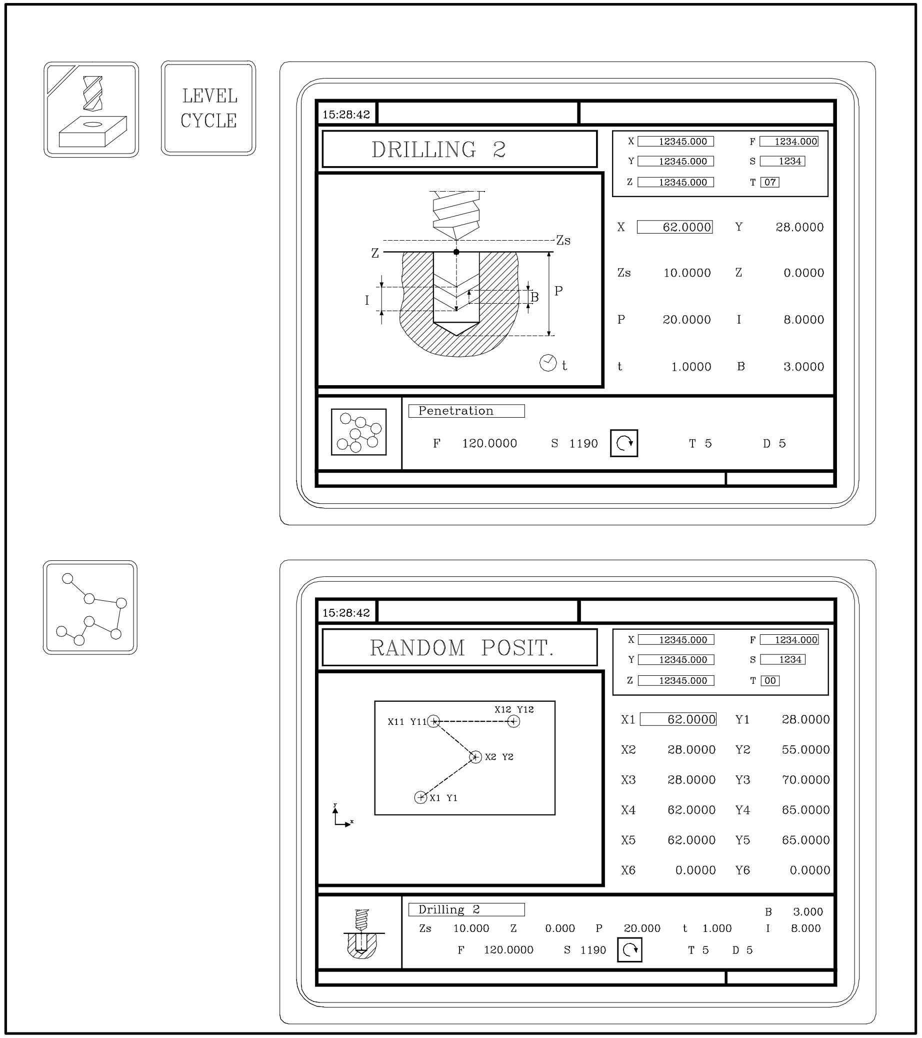



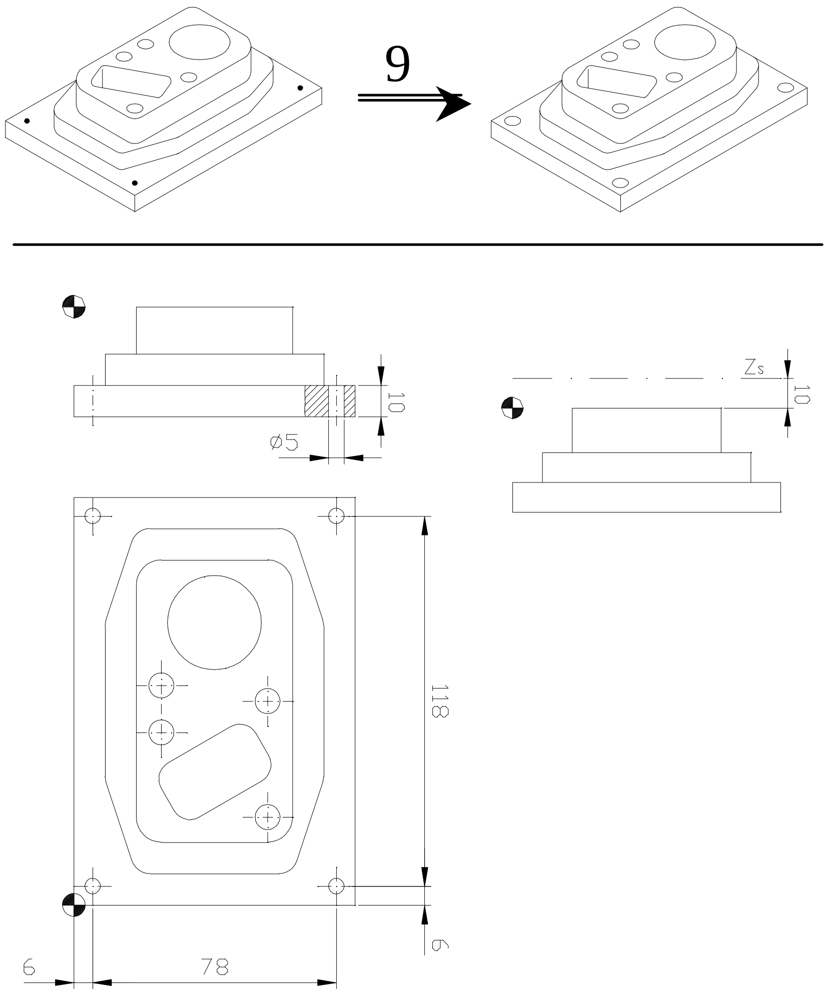



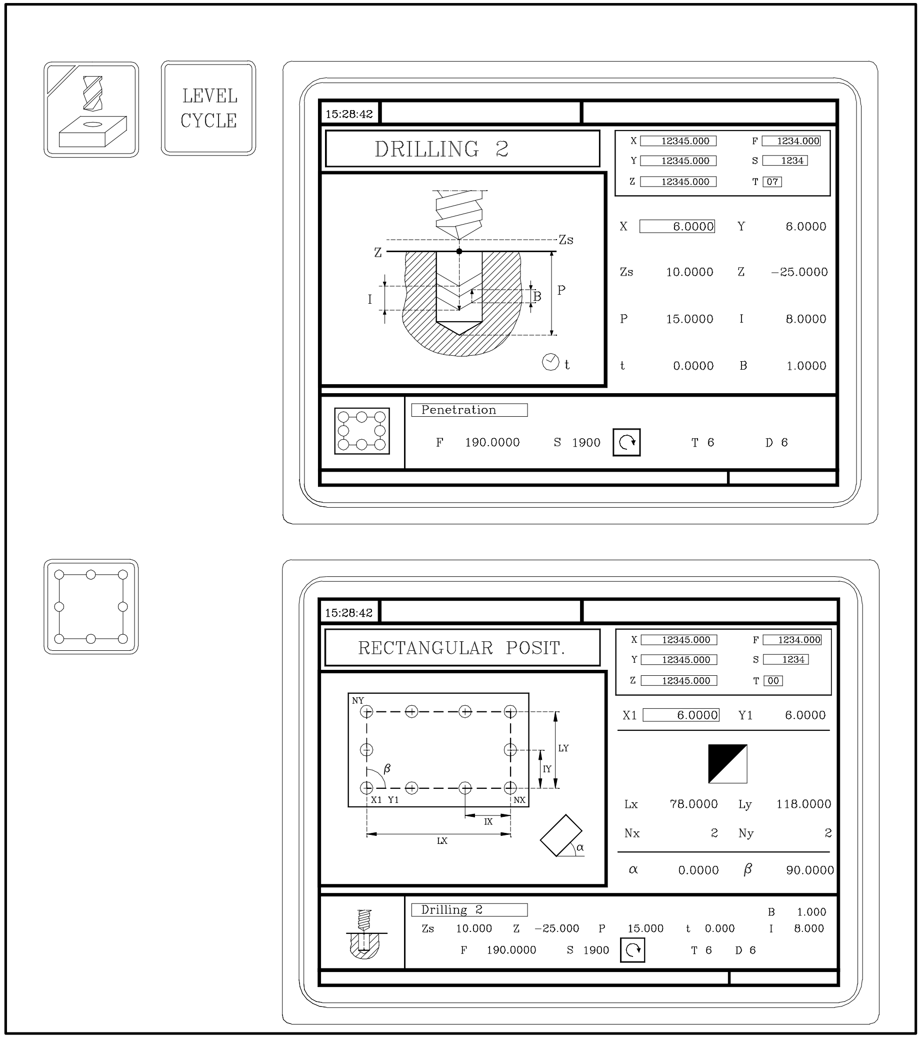
# Step 8: Drilling + multiple positioning at several points.



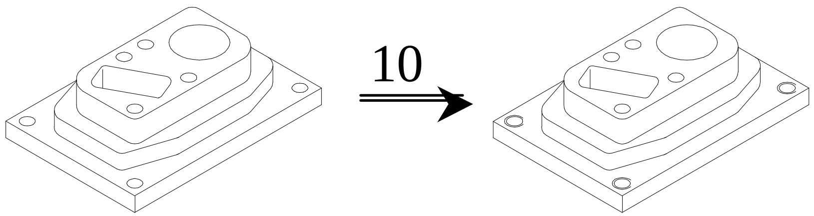



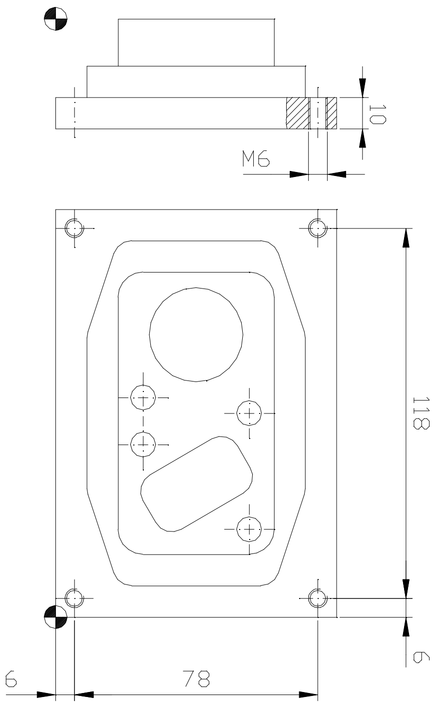
# Step 9: Drilling + multiple positioning in parallelogram pattern.

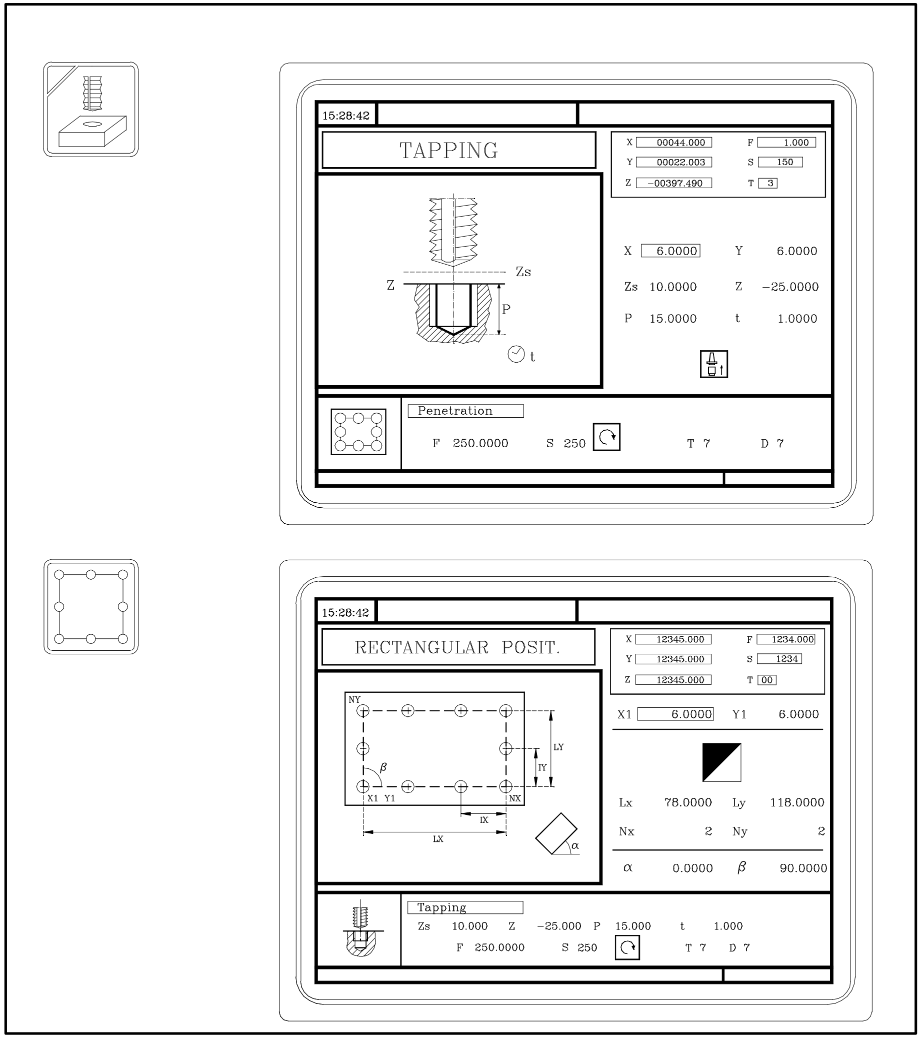


## Step 10: Tapping + multiple positioning in parallelogram pattern.



# Step 11: Part-program.

Once the operations have been entered, the part program will be like this: