

**CNC**

**8055 MC**

**Self-teaching manual**

REF. 1010



**FAGOR AUTOMATION**



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

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

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

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

The content of this manual and its validity for the product described here has been verified. Even so, involuntary errors are possible, thus no absolute match is guaranteed. Anyway, the contents of the manual is periodically checked making and including the necessary corrections in a future edition. We appreciate your suggestions for improvement.

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

---

This product uses the following source code, subject to the terms of the GPL license. The applications *busybox* V0.60.2; *dosfstools* V2.9; *linux-ftpd* V0.17; *ppp* V2.4.0; *utelnnet* V0.1.1. The library *grx* V2.4.4. The linux kernel V2.4.4. The linux boot *ppcboot* V1.1.3. If you would like to have a CD copy of this source code sent to you, send 10 Euros to Fagor Automation for shipping and handling.

# INDEX

<b>CHAPTER 1</b>	<b>THEORY ON CNC MACHINES.</b>	
	1.1 Machine axes.....	7
	1.2 Machine reference zero and part zero.....	9
	1.3 Home search.....	10
	1.4 Travel limits.....	11
	1.5 Setting the part zero.....	12
	1.6 Work units.....	13
<b>CHAPTER 2</b>	<b>THEORY ON TOOLS.</b>	
	2.1 The tool change.....	17
	2.2 The tool table.....	18
	2.3 Tool calibration.....	20
<b>CHAPTER 3</b>	<b>MANUAL PRACTICE.</b>	
	3.1 Description of the screen and the keyboard.....	25
	3.1.1 Keyboard description.....	26
	3.1.2 Description of the standard screen.....	28
	3.1.3 Description of the auxiliary screen.....	29
	3.2 Home search.....	30
	3.3 Spindle.....	31
	3.3.1 Speed ranges (gears).....	32
	3.4 Jogging the axes.....	33
	3.4.1 Handwheels.....	34
	3.4.2 Jogging.....	35
	3.4.3 Moving an axis to a coordinate.....	36
	3.5 Tool change.....	37
	3.5.1 Tool change position.....	38
	3.5.2 Tool calibration.....	39
	3.5.3 How to change any data of the tool table.....	42
	3.5.4 Checking for proper calibration.....	43
<b>CHAPTER 4</b>	<b>OPERATIONS OR CYCLES.</b>	
	4.1 Description of the screen and the keyboard.....	47
	4.2 Work modes.....	49
	4.3 Operations or cycles.....	50
	4.3.1 Editing an operation or cycle.....	51
	4.3.2 Simulating an operation or cycle.....	54
	4.3.3 Executing an operation or cycle.....	60
<b>CHAPTER 5</b>	<b>SUMMARY OF WORK CYCLES.</b>	
	5.1 Profile milling operation.....	65
	5.2 Surface milling and slot milling operation.....	66
	5.3 Pocket cycle with a profile.....	67
	5.4 Rectangular and circular boss cycles.....	68
	5.5 Rectangular and circular pocket cycles.....	69
	5.6 Positioning.....	71
	5.7 Boring operation.....	72
	5.8 Reaming operation.....	73
	5.9 Tapping operation.....	74
	5.10 Drilling and center punching operation.....	75
	5.11 Multiple positioning in several locations.....	77
	5.12 Multiple positioning in a straight line.....	78
	5.13 Multiple positioning in an arc.....	79
	5.14 Multiple positioning in a rectangular pattern.....	80
	5.15 Multiple positioning in a grid pattern.....	81
	5.16 Profile editor.....	82



·MC· Option

REF. 1010

CHAPTER 6

CONVERSATIONAL PART PROGRAMS.

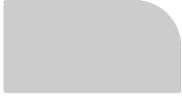
---

6.1	What is a conversational part-program?.....	85
6.2	Editing a part-program.....	86
6.3	Modifying a part-program. ....	89
6.4	Simulating/executing an operation. ....	93
6.5	Simulating/executing a part-program.....	94
6.6	Simulating/executing a part-program starting with an operation. ....	95
6.7	Copying a part-program into another one.....	96
6.8	Deleting a new part program. ....	97

APPENDIX

PROGRAMMING EXAMPLE.

---



·MC· Option

REF. 1010

# 1

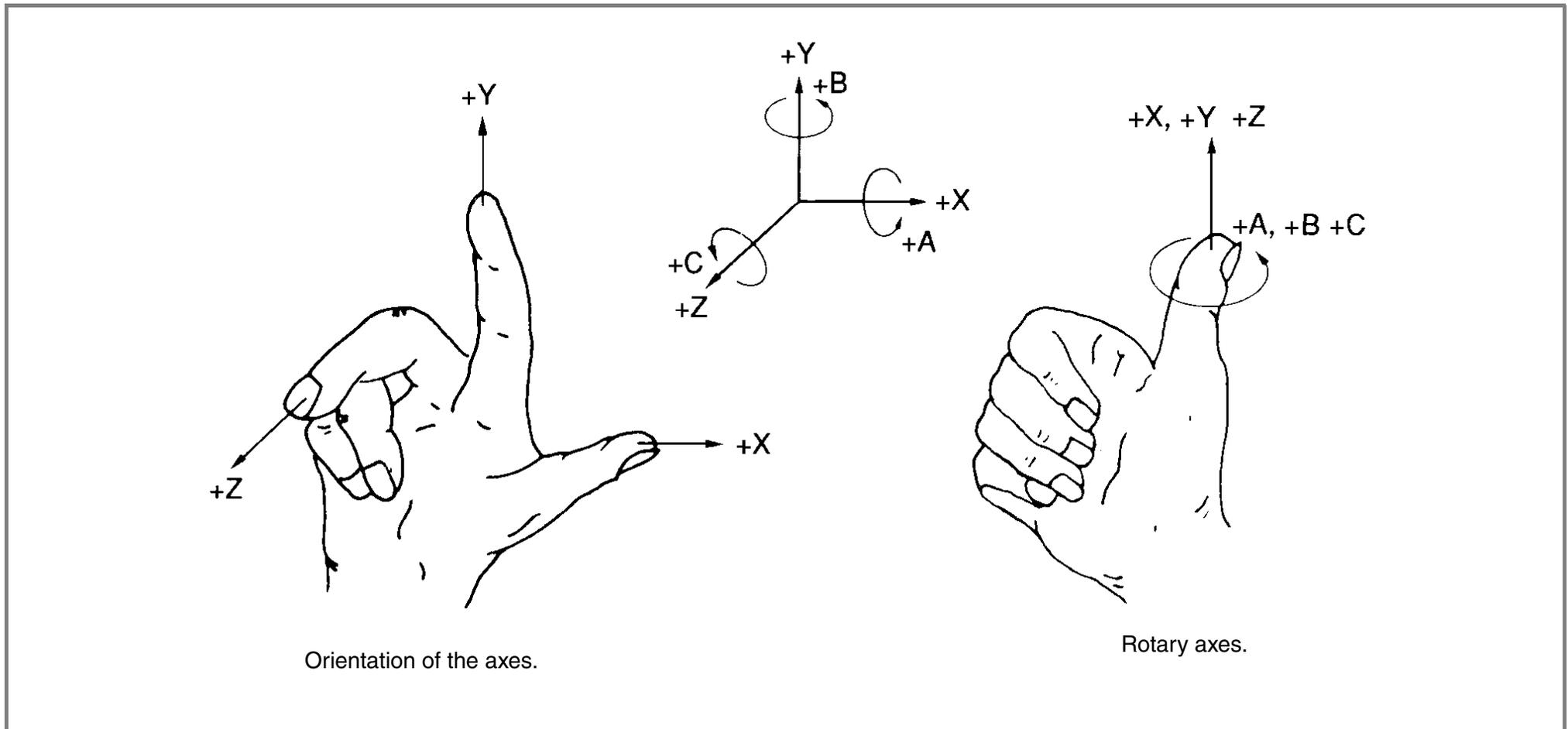
## THEORY ON CNC MACHINES.

- How are the machine axes named?.
- What do machine reference zero and part zero mean?.
- What is "home search"?.
- What are travel limits?.
- How is a part zero preset?.
- Which are the work units?.

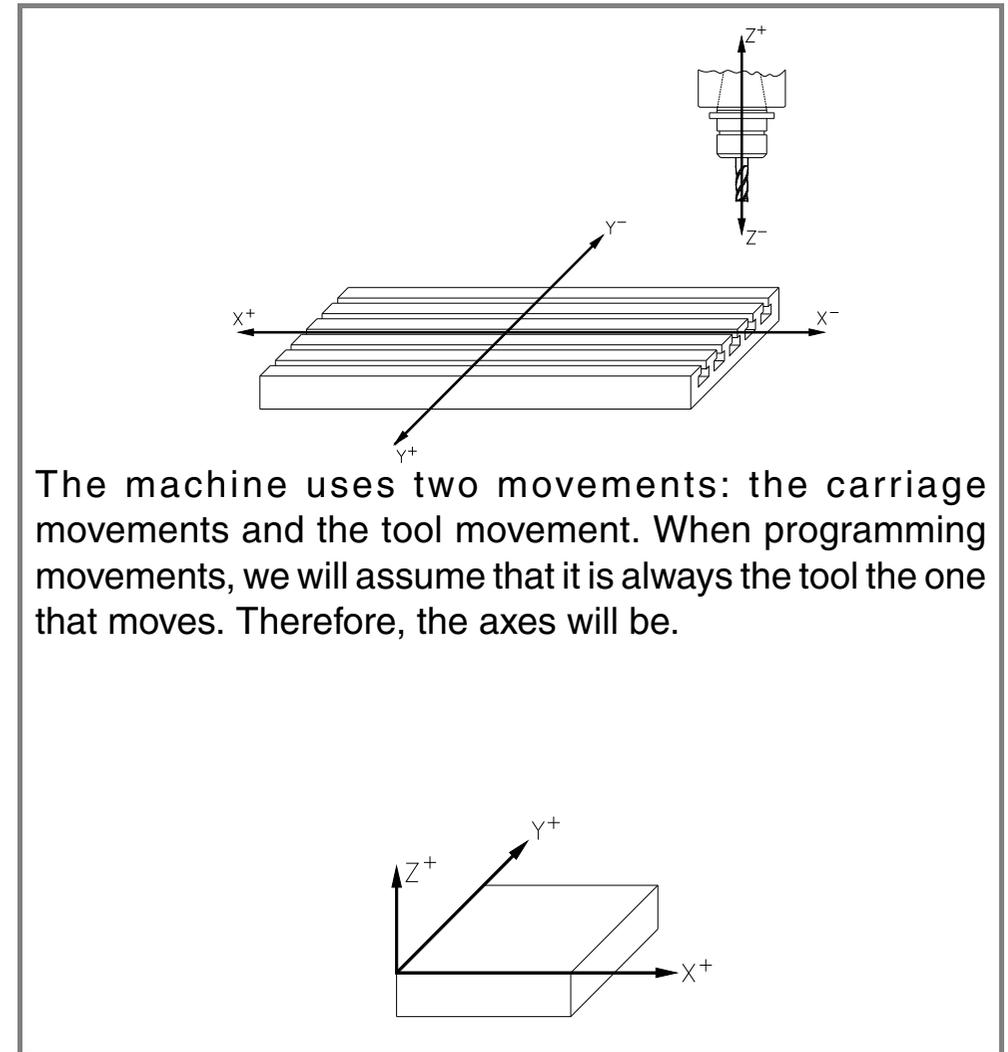
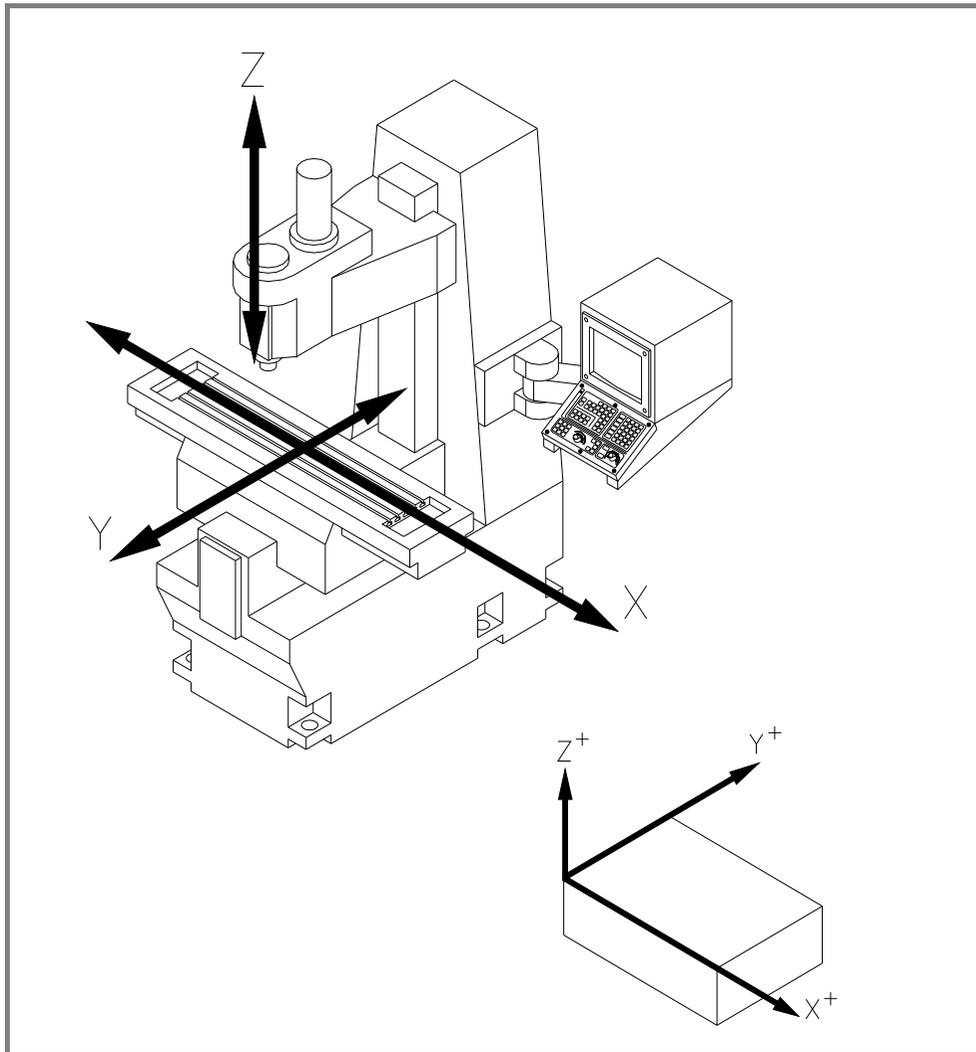


# 1.1 Machine axes.

The orientation of the axes depends on the type of machine according to the "right hand" rule.



The manual uses the following axis configuration.



## 1.2 Machine reference zero and part zero.

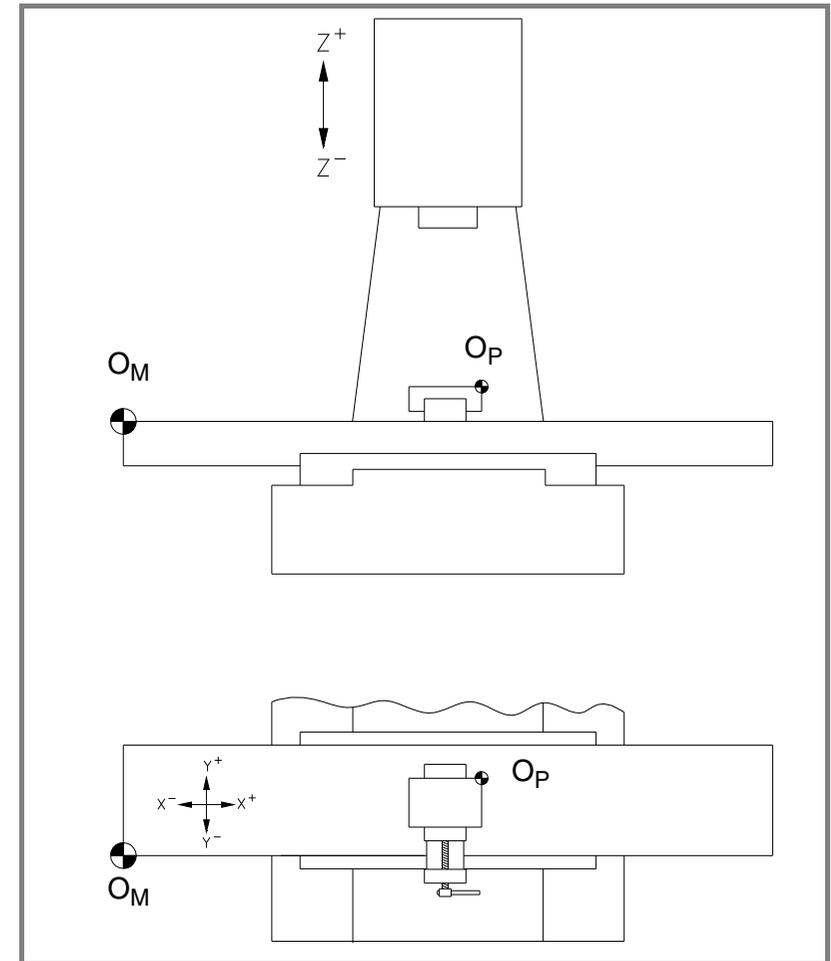
They are the references that the machine needs to work.

### Machine reference zero ( $O_M$ )

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

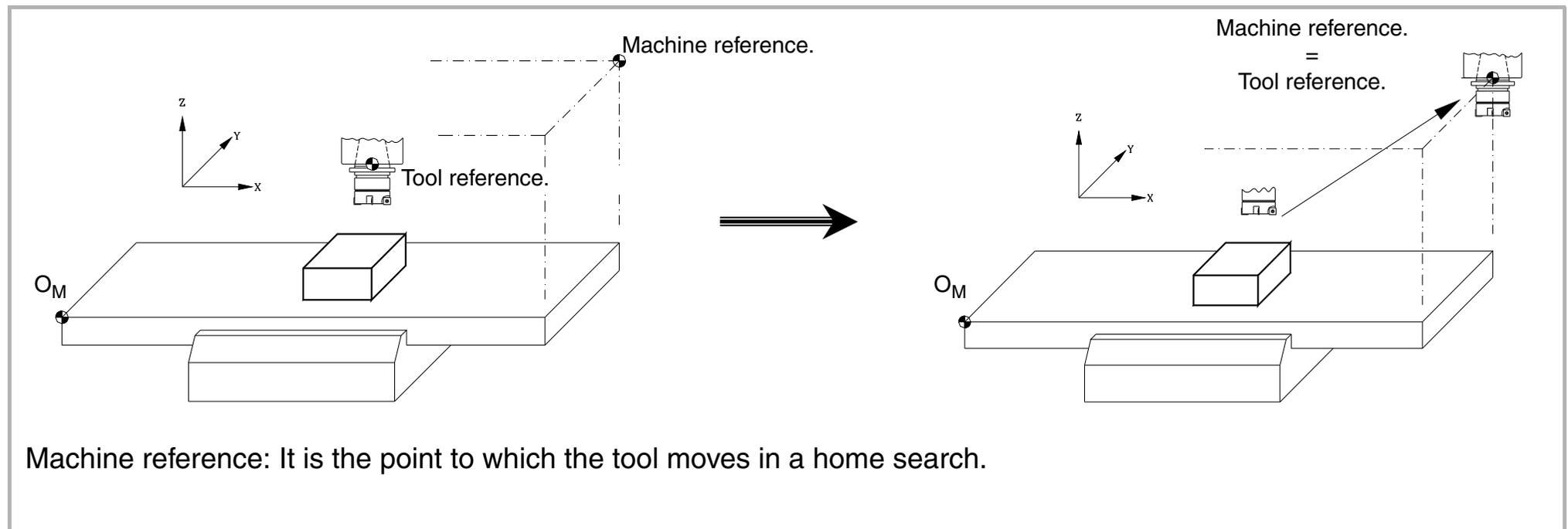
### Part zero ( $O_P$ )

It is set by the operator. It is the part's origin point, used as reference to program the movements. It may be preset anywhere on the part.



## 1.3 Home search.

When the CNC is off, the axes may be moved manually or unintentionally. When that happens, the CNC loses the actual position of the axes, that's why it is recommended (although not necessary) to do a "home search" when the turning the CNC on. With this operation, the tool moves to a point set by the manufacturer and the CNC synchronizes its position assuming the coordinates set by the manufacturer for that point, referred to machine reference zero.

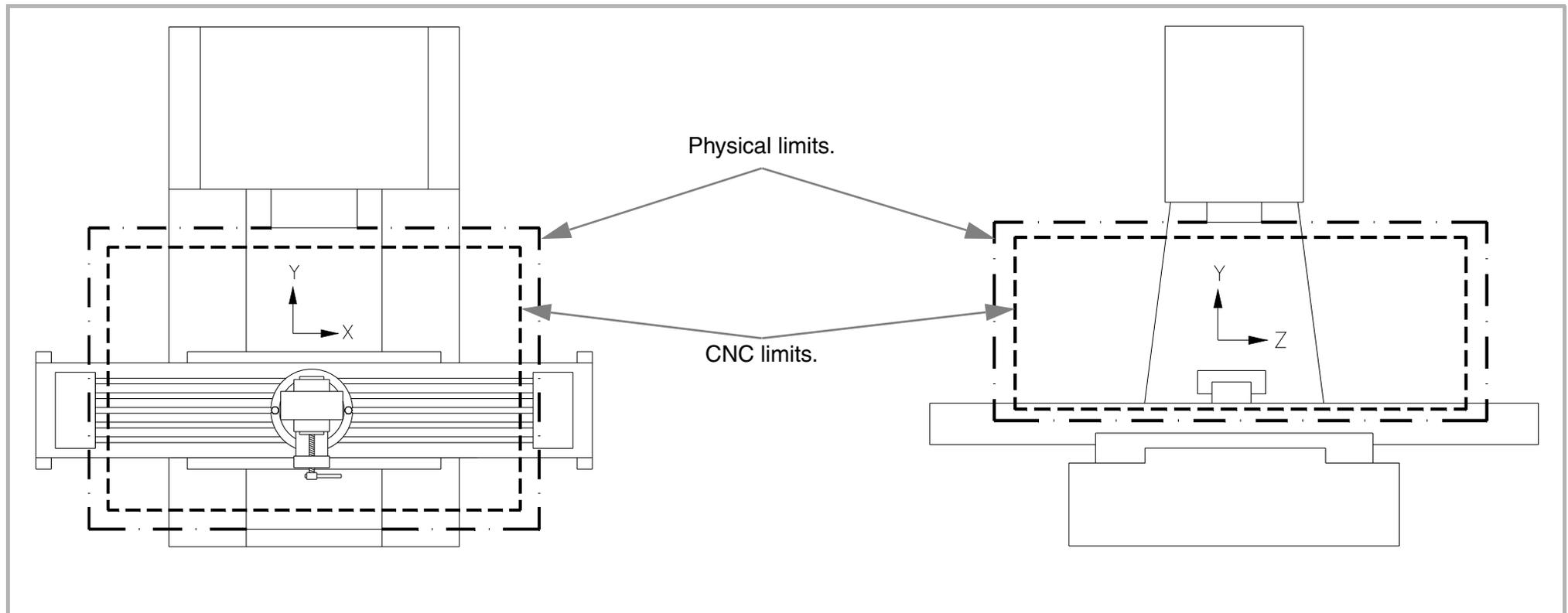


NOTE: With the new feedback devices (distance-coded reference marks - lo -), it is possible to know the position of the axes by just moving them a very short distance. This does away with the concept of machine reference.

## 1.4 Travel limits.

This type of machines have two types of travel limits:

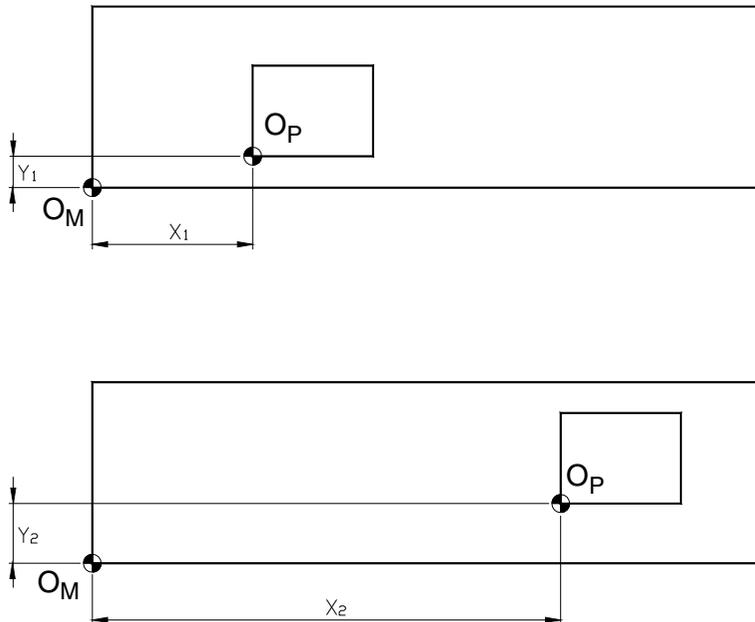
- Physical limits. They are set by the machine to prevent the carriages from going off its ways (cams and mechanical stops).
- CNC limits. They are set by the manufacturer at the CNC to prevent the carriages from running into the physical travel limits.



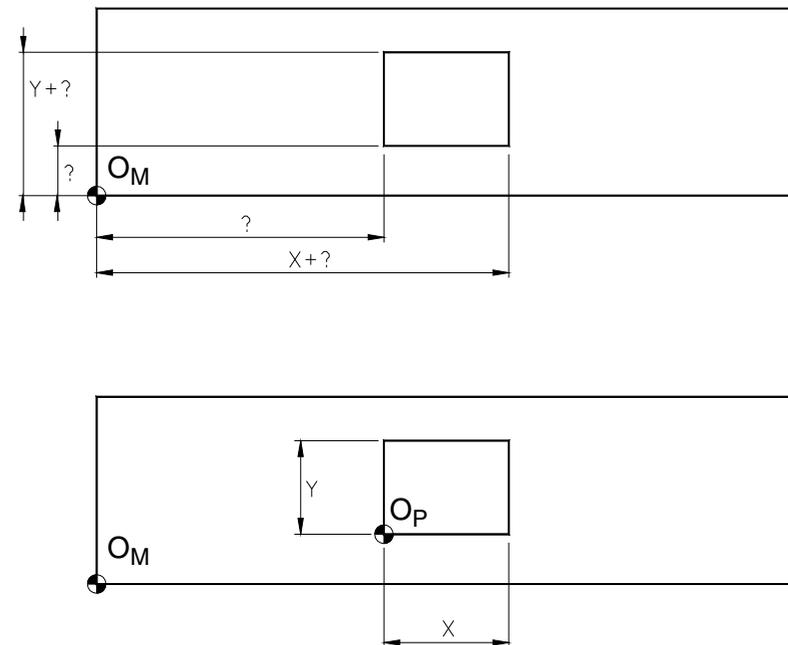
## 1.5 Setting the part zero.

The part zero is set on all three axes.

When machining several parts, the distance from machine reference zero ( $O_M$ ) to the part is different for each of them. It would require running a program for each part. When programming from a part zero ( $O_P$ ), it is irrelevant where the part is fixed.



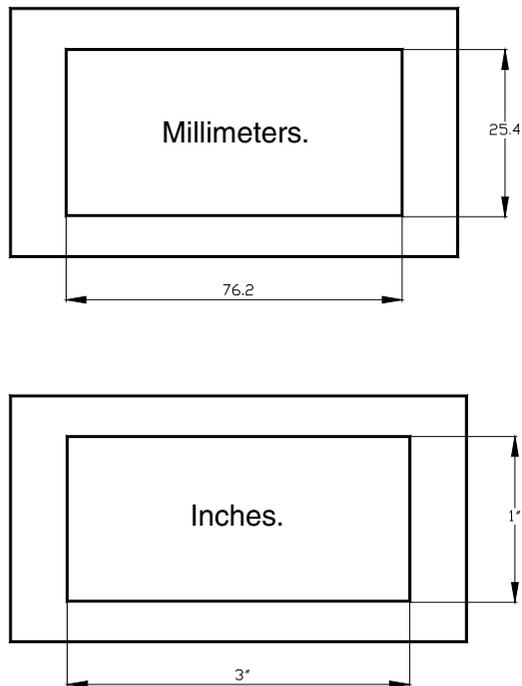
Programming gets complicated when done from the machine reference zero ( $O_M$ ) and it is only good for that part in that position. When programming from a part zero ( $O_P$ ), the part dimensions may be obtained from the blueprint.



## 1.6 Work units.

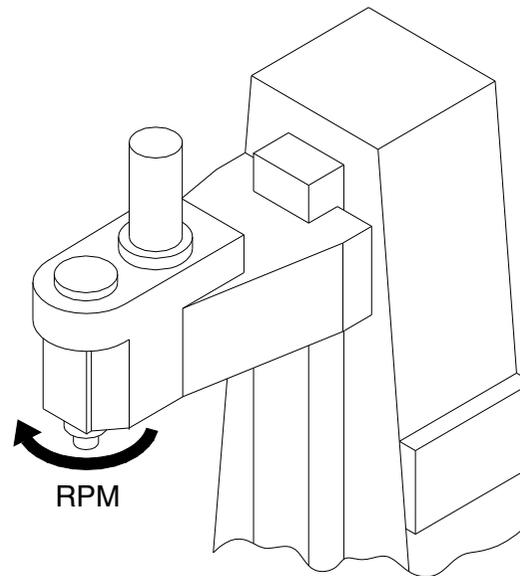
### Programming units.

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



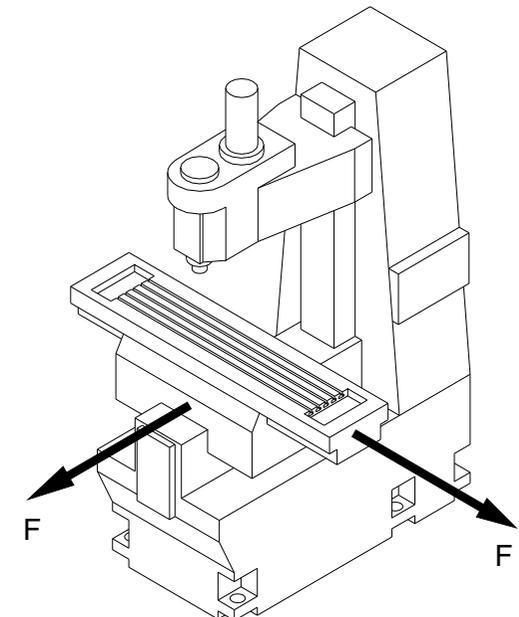
### Spindle speed.

The spindle speed is programmed in rpm.



### Feedrate of the axes.

The axis feedrate (F) is programmed in mm/min (inches/min).





# 2

## THEORY ON TOOLS.

---

- How is the tool change managed?.
- What is the tool table and what data does it contain?
- What is tool calibration?.
- Defects due to errors in the tool table.



## 2.1 The tool change.

The tools to be used by the CNC may be located in a magazine of the machine. Depending on whether the machine has a magazine or not, the tool change will be done differently:

- If the machine does not have a tool magazine, the tools must be changed manually (line on a conventional machine).
- If the machine has a tool magazine, the CNC will change the tools automatically.

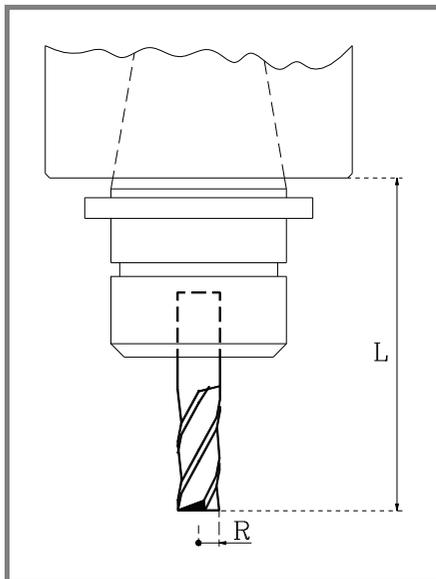
## 2.2 The tool table.

This table keeps the data related to the tools. When changing a tool, the CNC assumes the data defined in the table for that tool.

The data included in the table is:

- .T. Tool number.
- .D. Offset associated with the tool.

The offset defines the tool dimensions.



- .L. Tool length.
- .R. Tool radius.
- .J. Tool radius wear.
- .K. Tool length wear.

- Nominal life.

Machining time or the number of operations that the tool can carry out.

- Real life.

Machining time or number of operations carried out.

- Family.

Tools with the same characteristics.

- Status.

The CNC updates this data. The operator cannot modify them.

Tool type.

- N. Normal tool.

- S. Special tool.

Tool status.

- A. Available tool.

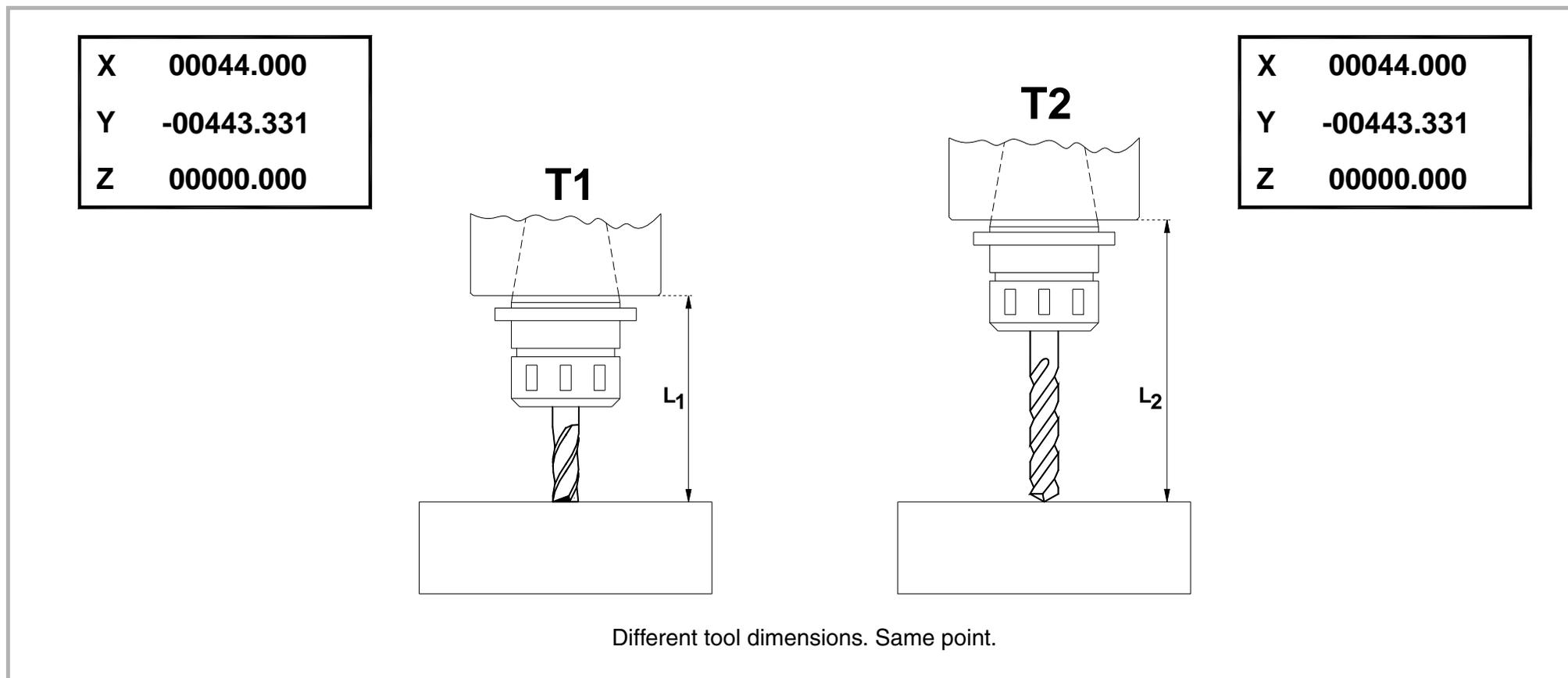
- E. Expired tool (worn out, "real life" > "nominal life").

- R. Tool rejected by the PLC.

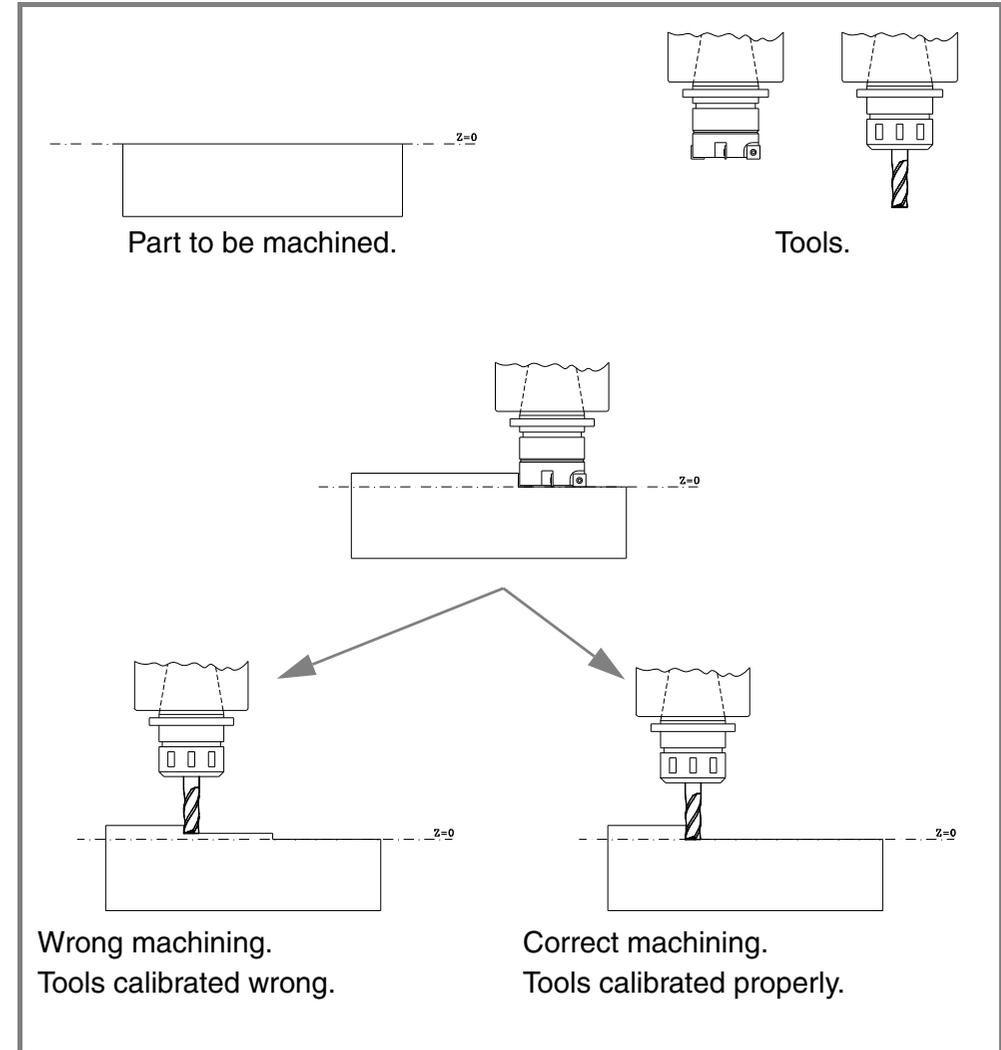
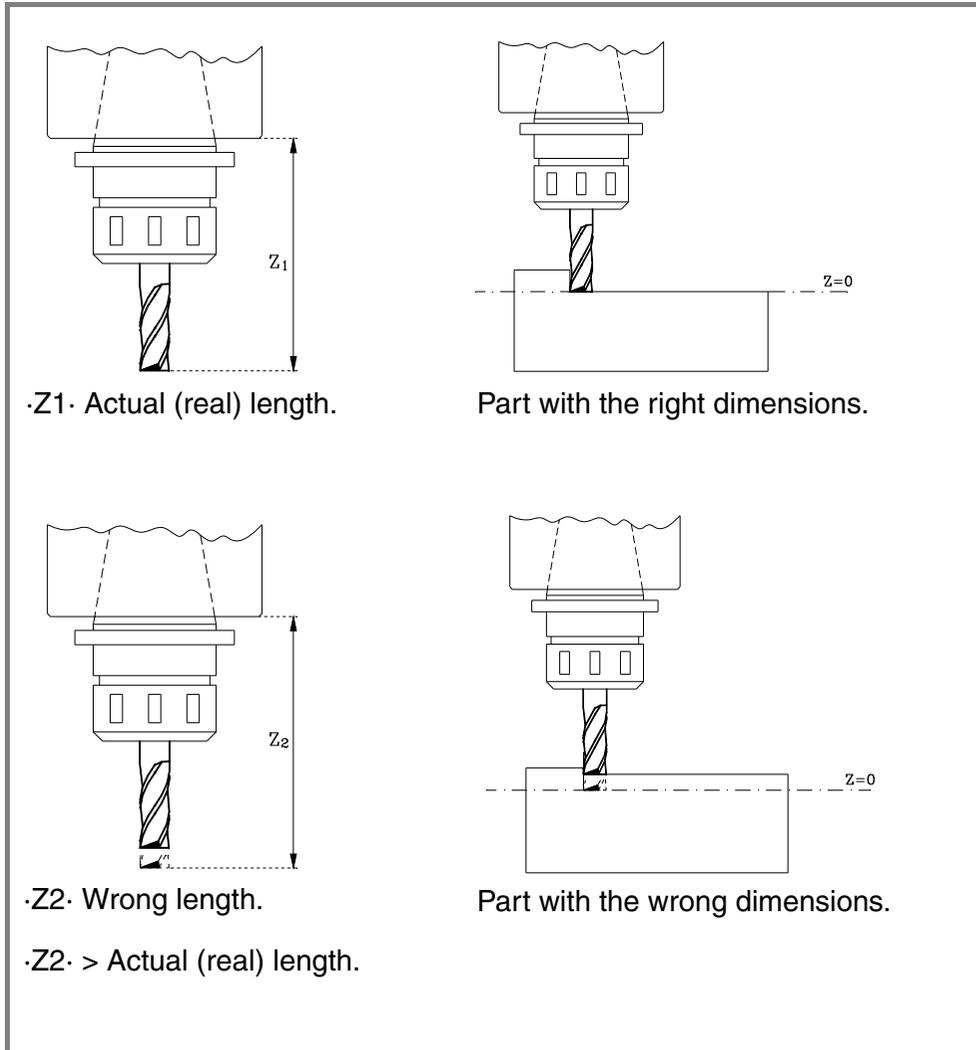
When requesting an expired (worn out) or rejected tool, the CNC looks for a tool that belongs to the same family. If there is one, it will select it and if not, it will show the corresponding error message.

## 2.3 Tool calibration.

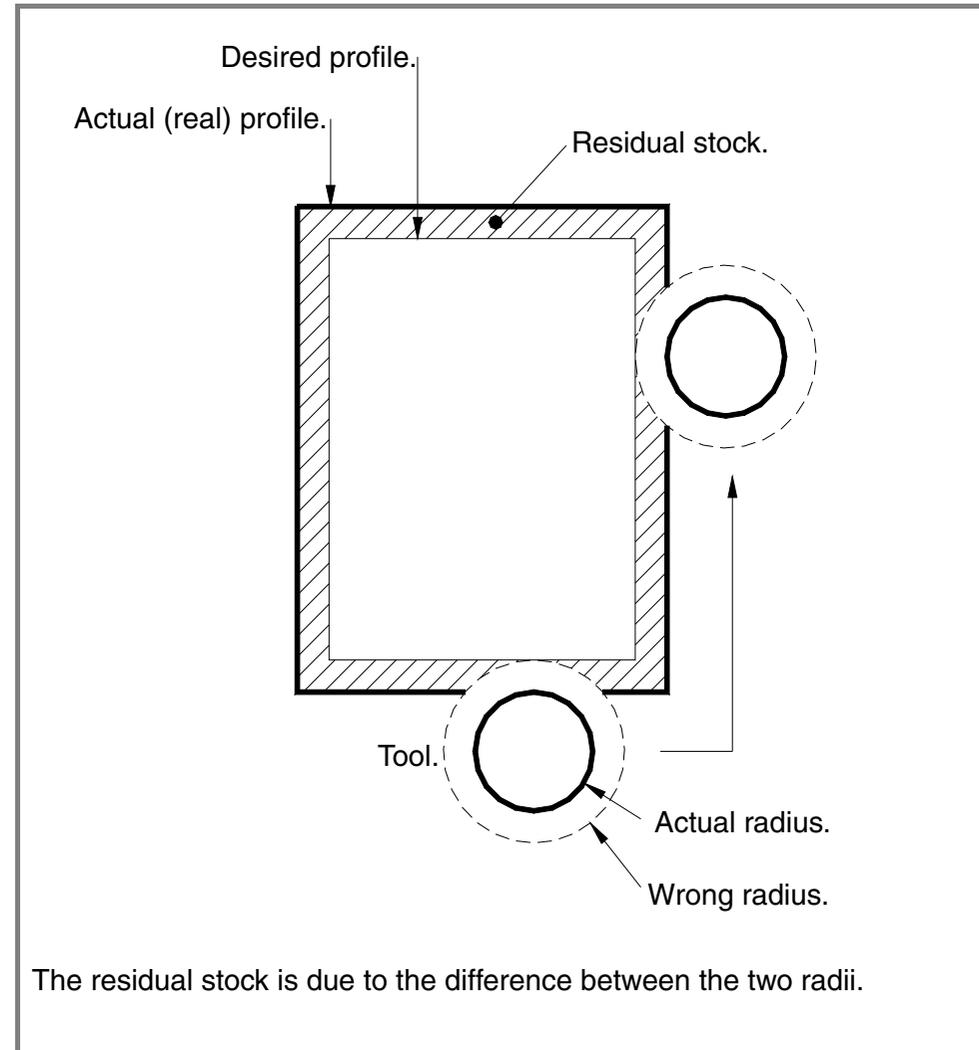
Tool calibration is the operation the CNC uses to define the dimensions of the tool. It is very important to carry out this operation properly so the parts are made with the right dimensions and so when changing the tool the CNC keeps controlling the same point.



## DEFECTS DUE TO WRONG TOOL LENGTH CALIBRATION.



## DEFECTS DUE TO WRONG TOOL RADIUS CALIBRATION.



# 3

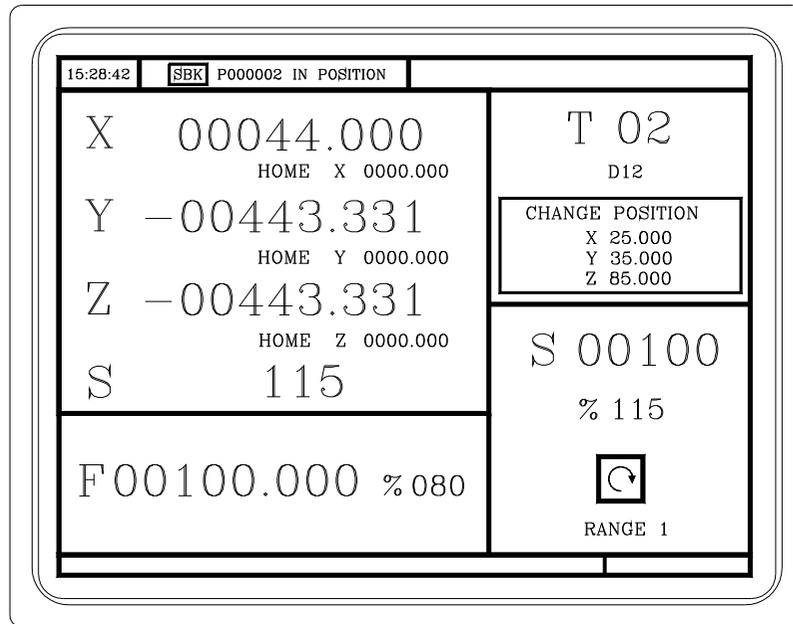
## MANUAL PRACTICE.

- Description of the screen and the keyboard.
- How to do a home search.
- Working with the spindle.
- Ways to jog the axes (handwheels, continuous jog, incremental jog).
- Tools.

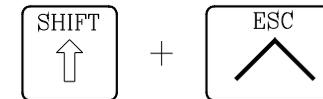


### 3.1 Description of the screen and the keyboard.

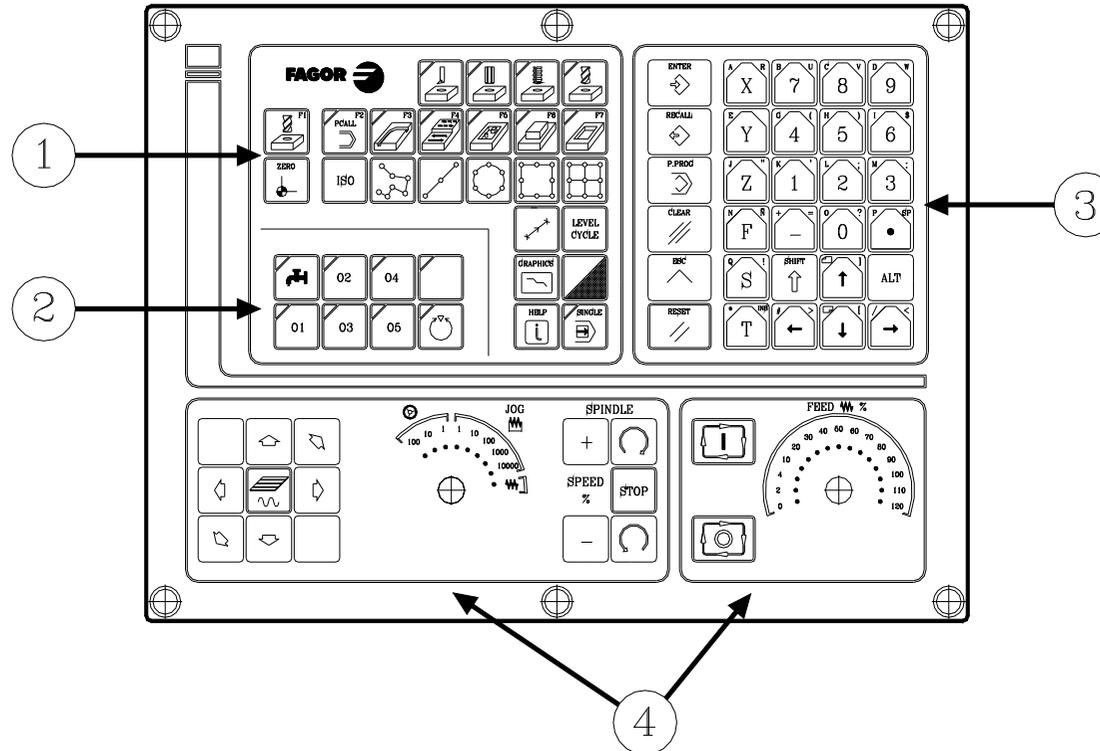
On power-up, the CNC shows the following screen.



If the CNC does not show this screen, the CNC may be in ·M· mode. To go into ·MC· mode, press the following keys.

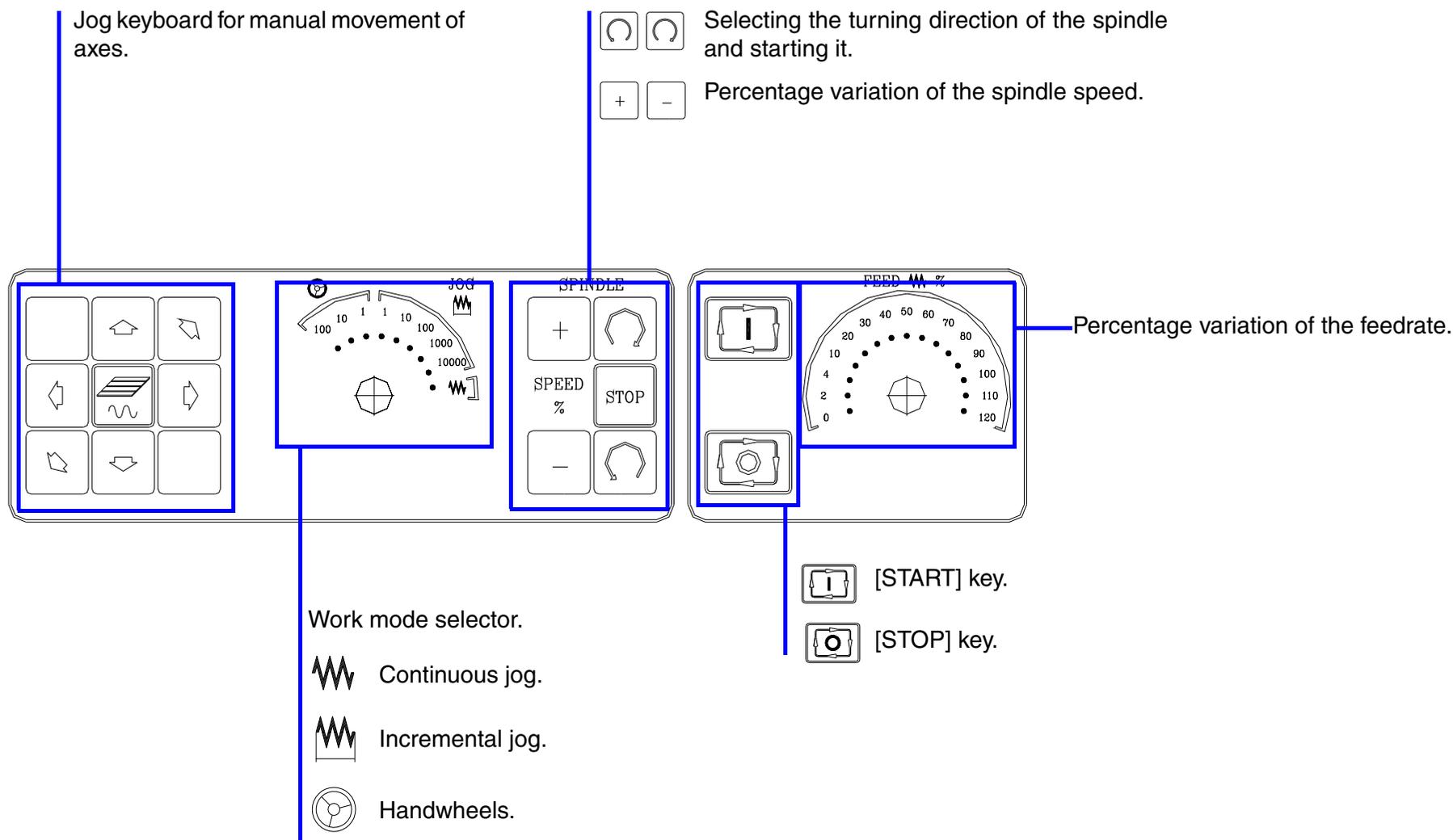


## 3.1.1 Keyboard description.

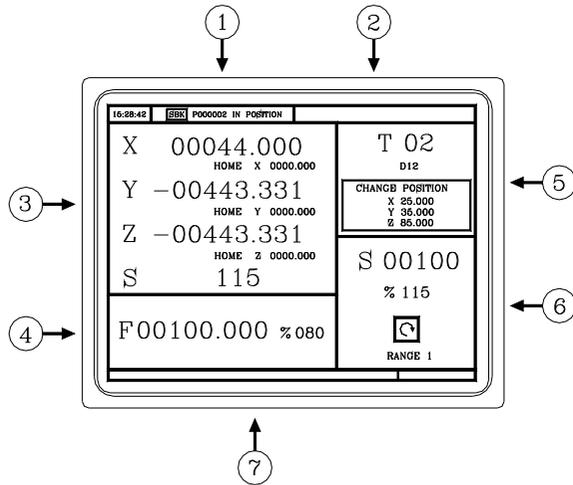


- 1• Keys to define the machining operations.
- 2• Keys for external devices.
- 3• Alphanumeric keys and command keys.
- 4• Operator panel.

Operator panel description.



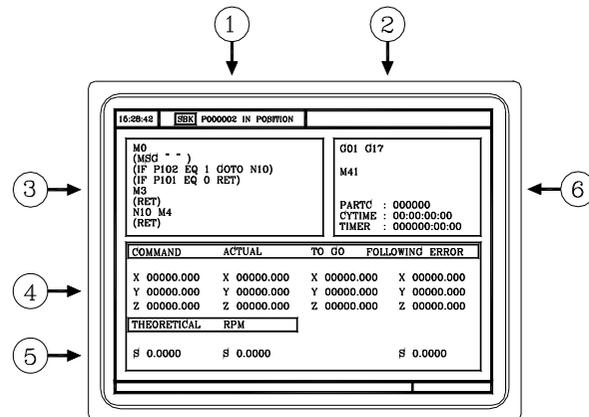
## 3.1.2 Description of the standard screen.



1. Time, type of execution ("single block"/continuous), program number, execution status (in position, execution, interrupted or reset) and PLC messages.
2. CNC messages.
3. Tool position referred to the part zero and to home. Real spindle rpm.
4. Axis feedrate and percentage applied.
5. Tool information.
6. Spindle information. Selected work speed, applied percentage, spindle status (turning clockwise, counterclockwise or stopped) and active range (gear).
7. Help messages.

## 3.1.3 Description of the auxiliary screen.

 Changing the screen.



1. Time, type of execution ("single block"/continuous), program number, execution status (in position, execution, interrupted or reset) and PLC messages.
2. CNC messages.
3. Blocks of the program currently selected.
4. Information of the axes. Target position of the movement (COMMAND), current axis position (ACTUAL), remaining distance (TO GO) and the difference between the theoretical position and the real position (FOLLOWING ERROR).
5. Spindle information. Programmed theoretical feedrate, actual (real) feedrate and following error (axis lag).
6. Active G and M functions.  
 Number of consecutive parts executed with the program (PARTC), part execution time (CYTIME) and PLC timer.

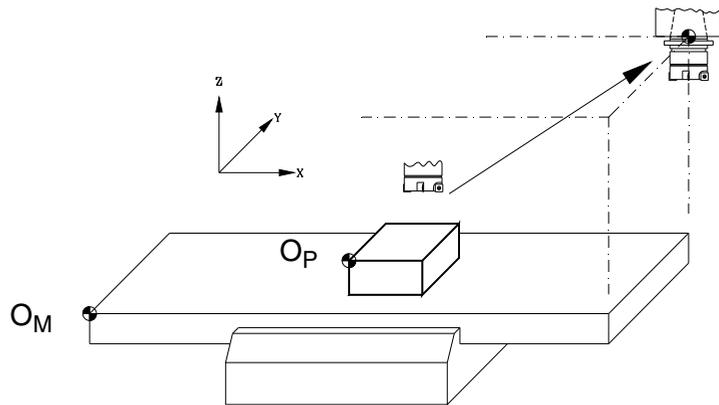
### 3.2 Home search.

After powering the machine up, it is recommended to home the axes just in case they have been moved while the machine was turned off. There are two ways to do a home search.

#### Automatic home search.

The home search is carried out on all the axes at the same time, according to a subroutine pre-established by the manufacturer. The CNC keeps the part zero.

Press:  

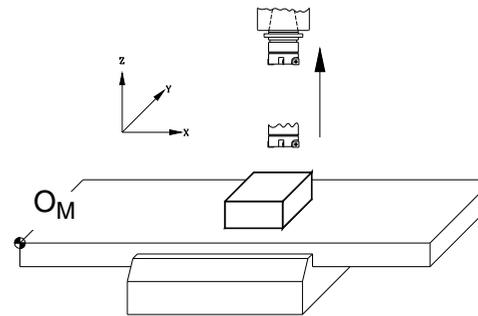


The CNC shows the coordinates referred to part zero ( $O_P$ ) considering the tool dimensions.

#### Manual home search.

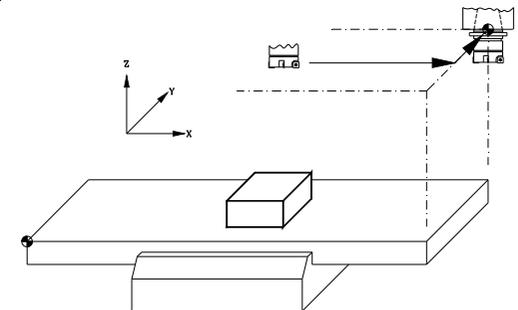
Each axis is homed separately. The CNC does not keep the part zero.

Press:   



Press:   

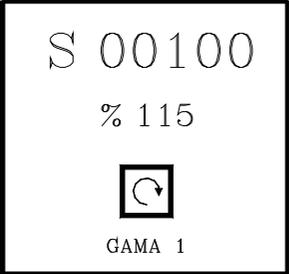
Press:   



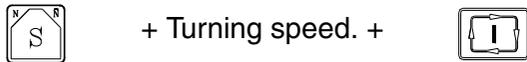
The CNC shows the coordinates referred to machine zero ( $O_M$ ) considering the tool dimensions.

## 3.3 Spindle.

The CNC shows the following spindle-related information.

	Selected speed.
	Percentage applied.
	Turning direction.
	Active spindle range (gear).

To select the work speed (in rpm), press the following keystroke sequence.

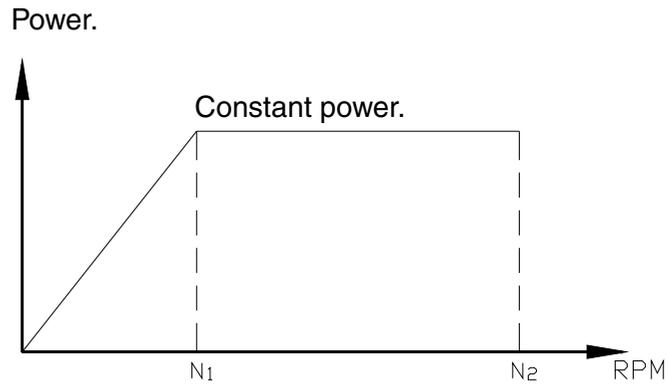


Use the keys of the operator panel to start the spindle.

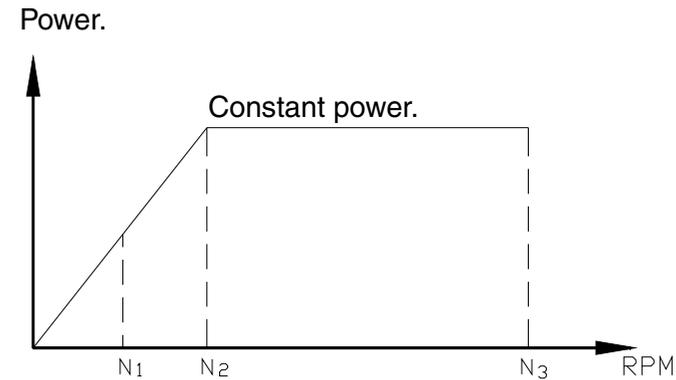
-  Spindle clockwise. Clockwise.
-  Stops the spindle.
-  Spindle counterclockwise. Counterclockwise.
-  Increase or decrease the percentage of spindle speed.

## 3.3.1 Speed ranges (gears).

The machine may use a gear box. Use these gears to select the best motor torque for the programmed speed. It is recommended to always work at constant power to extend tool life.



SPEED RANGE (GEAR) ·1·



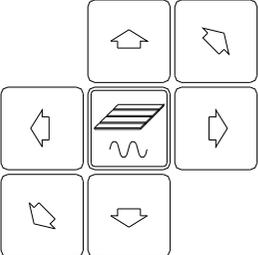
SPEED RANGE (GEAR) ·2·

For a work speed between N1 and N2, use GEAR 1.

For a work speed between N2 and N3, use GEAR 2.

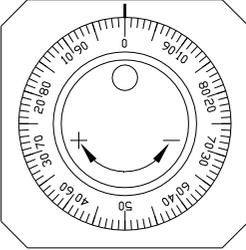
### 3.4 Jogging the axes.

The axes may be jogged using the jog keyboard or handwheels.



Each key can move an axis in one direction, according to the axes of the machine.

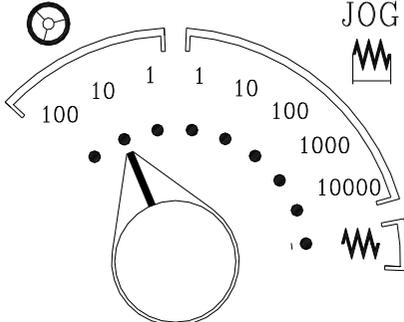
Jog keyboard.



It may have one, two or three handwheels. It follows the turning direction of the handwheel.

Handwheels.

Use the selector switch to select the type of feed.



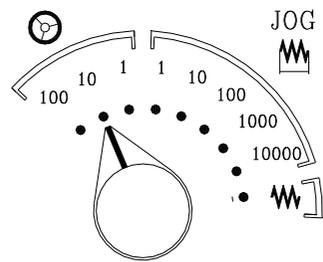
Movement with handwheels.

JOG Movement in incremental jog.

Movement in continuous jog.

## 3.4.1 Handwheels.

- Use the selector switch to select the type of feed.



- (1) Movement: 1 micron.
- (10) Movement: 10 microns.
- (100) Movement: 100 microns.

- If the machine has 1 handwheel.

Select an axis using the JOG keys. The machine moves the axis as the handwheel is being turned.

- If the machine has 2 or more handwheels.

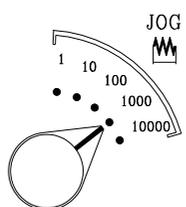
The machine moves one axis with each handwheel.

### 3.4.2 Jogging.

#### Incremental jog.

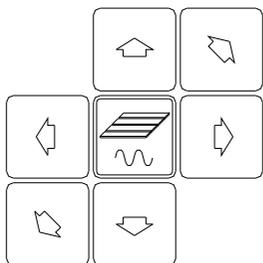
Every time a jog key is pressed, the axes move the selected increment and at the programmed feedrate (in rapid if F=0).

- Use the selector switch to select the type of feed.



(1)	Movement: 0.001 mm.
(10)	Movement: 0.010 mm.
(100)	Movement: 0.100 mm.
(1000)	Movement: 1.000 mm.
(10000)	Movement: 10.000 mm.

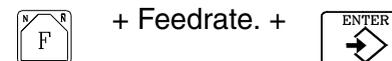
- Jogging the axes with the jog keyboard.



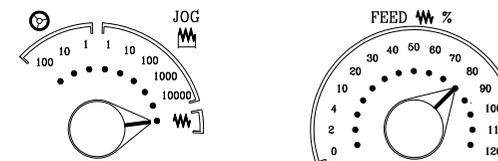
#### Continuous jog.

While a jog key is kept pressed, the axes move at the selected feedrate, considering the selected percentage (0% to 120%).

- Enter the desired feedrate.



- Use the selector switch to select the type of feedrate and the percentage to be applied.



- Jogging the axes with the jog keyboard.
- If the rapid key is pressed while jogging the axes, they will move at the maximum feedrate possible (set by the manufacturer).

 "Rapid" key.

## 3.4.3 Moving an axis to a coordinate.

The [START] key may be used to move an axis to a particular coordinate. Proceed as follows:

   On the standard screen, select the axis to be moved.

Enter the target coordinate for the axis.

 Enter the target coordinate for the axis.

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

## 3.5 Tool change.

### Machine with manual tool changer.

The change is done like on a conventional machine.

1. Change the tool on the machine.



2. Press the [T] key.

3. Enter the tool number for the CNC to assume the data of the new tool.



4. Press [START].

### Machine with automatic tool changer.

The CNC manages the tool change.



1. Press the [T] key.

2. Enter the tool number.

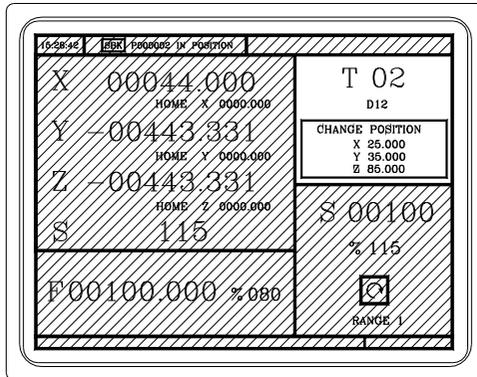


3. Press [START].

4. The CNC changes the tool and assumes the data of the new tool.

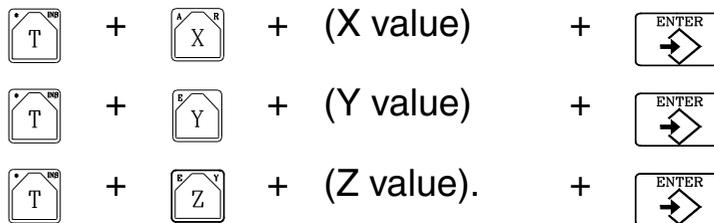
## 3.5.1 Tool change position.

The manufacturer may let the operator select a point where the tool change will take place.



Tool change position referred to machine reference zero.

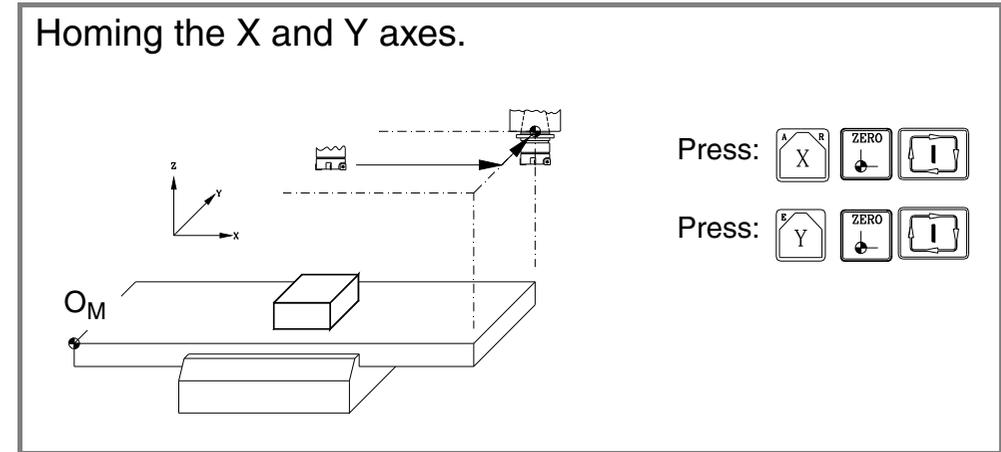
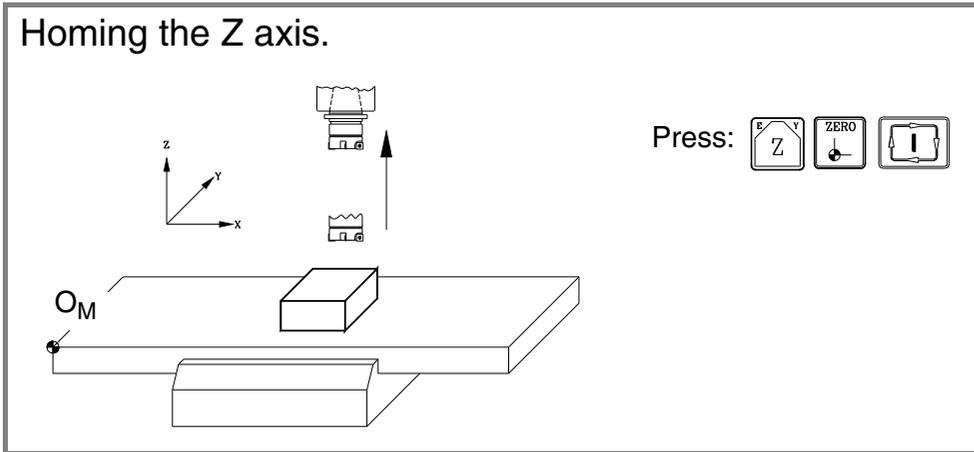
Enter in X, Y and Z, the values of the point where the tool change will take place.



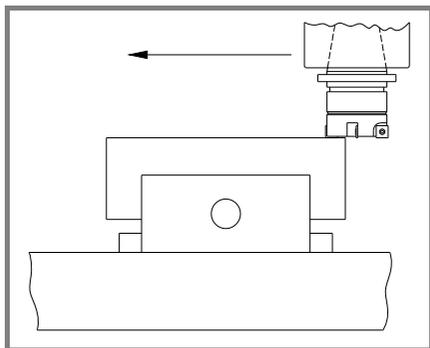
When requiring a tool change, if the manufacturer has set that way, the CNC will move the axes to that point for the tool change to take place.

## 3.5.2 Tool calibration.

- The machine must be homed before calibrating the tools.



- The calibration requires a smooth surface. Use continuous JOG or handwheels to mill the surface.



- Go into tool calibration mode. The CNC shows the screen for tool calibration.

The screenshot shows the 'TOOL CALIBRATION' screen on a CNC control. At the top left, the time is 15:28:42. The screen is divided into several sections:

- Top Right:** Real position of the axes and cutting conditions. Fields include X (00044.000), F (1.000), Y (00022.003), S (150), Z (-00397.490), and T (2).
- Center:** A graphic showing a tool with length L and radius R, positioned above a workpiece. A 'Z-ENTER' button is indicated.
- Right Side:** Tool number and tool dimensions. Fields include T (2), D (02), R (0.0000), L (0.0000) (Z-ENTER), I (0.0000), and K (0.0000).
- Bottom:** Information on the current tool status. Fields include Z (0.0000), NOMINAL LIFE (0), ACTUAL LIFE (0.00), FAMILY (0), and STATUS (N, A).

Callouts on the left side of the screen:

- Work mode.
- Help graphics.
- Height of the part used to calibrate the tool.
- Use the keys to move the cursor.

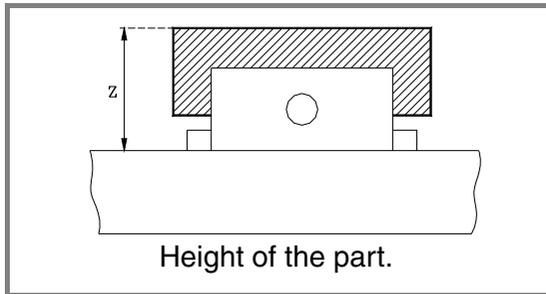
Callouts on the right side of the screen:

- Real position of the axes and cutting conditions.
- Tool number and tool dimensions.
- Information on the current tool status.

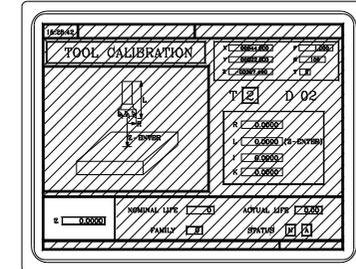
At the bottom left, there are four directional arrow keys: up, down, left, and right.

- Calibrate the tool as follows.

### 1. Measure the part.

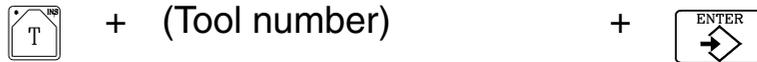


- Go to the window for tool calibration.
- Enter the Z value.

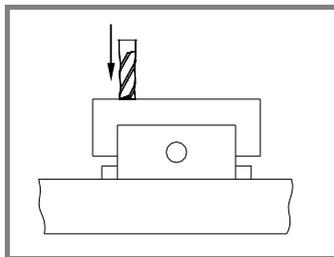


### 2. Start the spindle.

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



### 4. Jog the axes until touching the part along the Z axis.



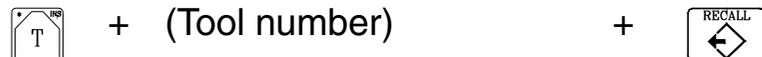
- Press:  
- The CNC calculates the new length and assigns it.
- The offset value (K) is reset to zero.

### 5. Enter the rest of the data (radius, nominal life, real life and family code) by hand.

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

## 3.5.3 How to change any data of the tool table.

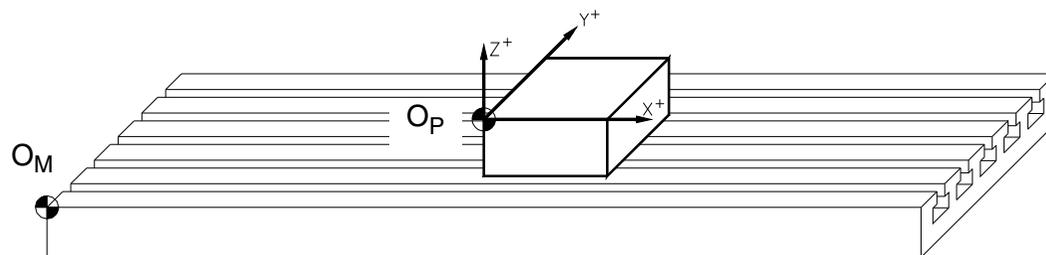
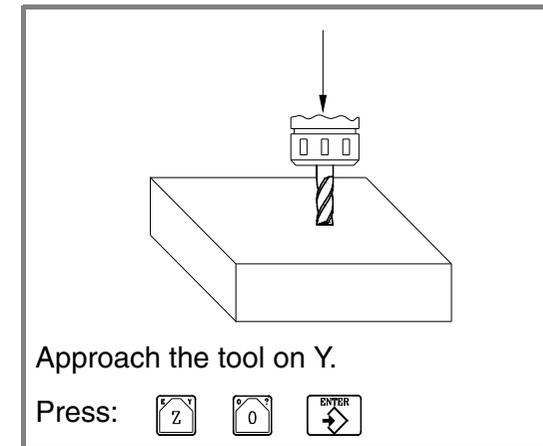
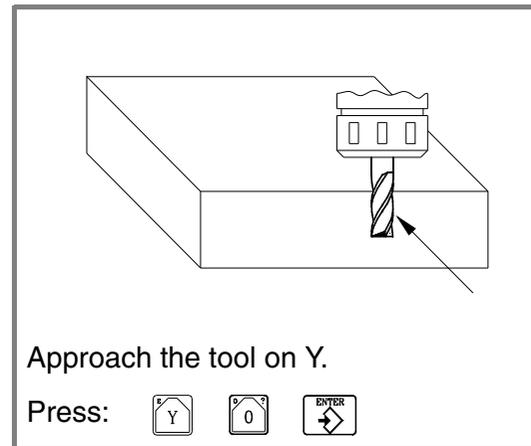
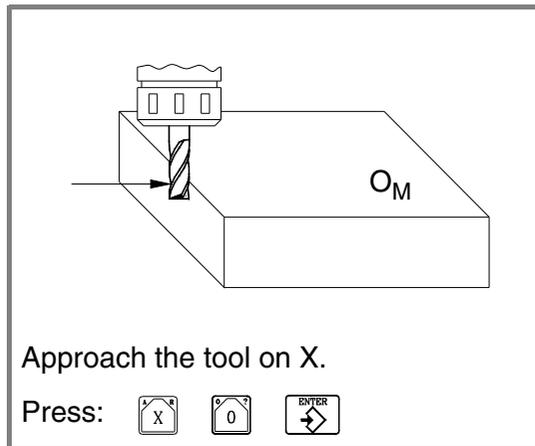
To change the tool data (T, D, R, L, I, K, nominal life, real life or family) get into tool calibration mode and display the tool data.



The CNC shows the data for that tool. To change them, place the cursor on the value to be modified, key in the new value and press [ENTER]. To exit the calibration option, press [ESC].

### 3.5.4 Checking for proper calibration.

- Presetting the part zero.



Withdraw the tool.

Position of the part zero.

- Start the spindle and touch the part surface with several tools and check the value on the screen. Although the tools are different, the value on the screen must be the same.



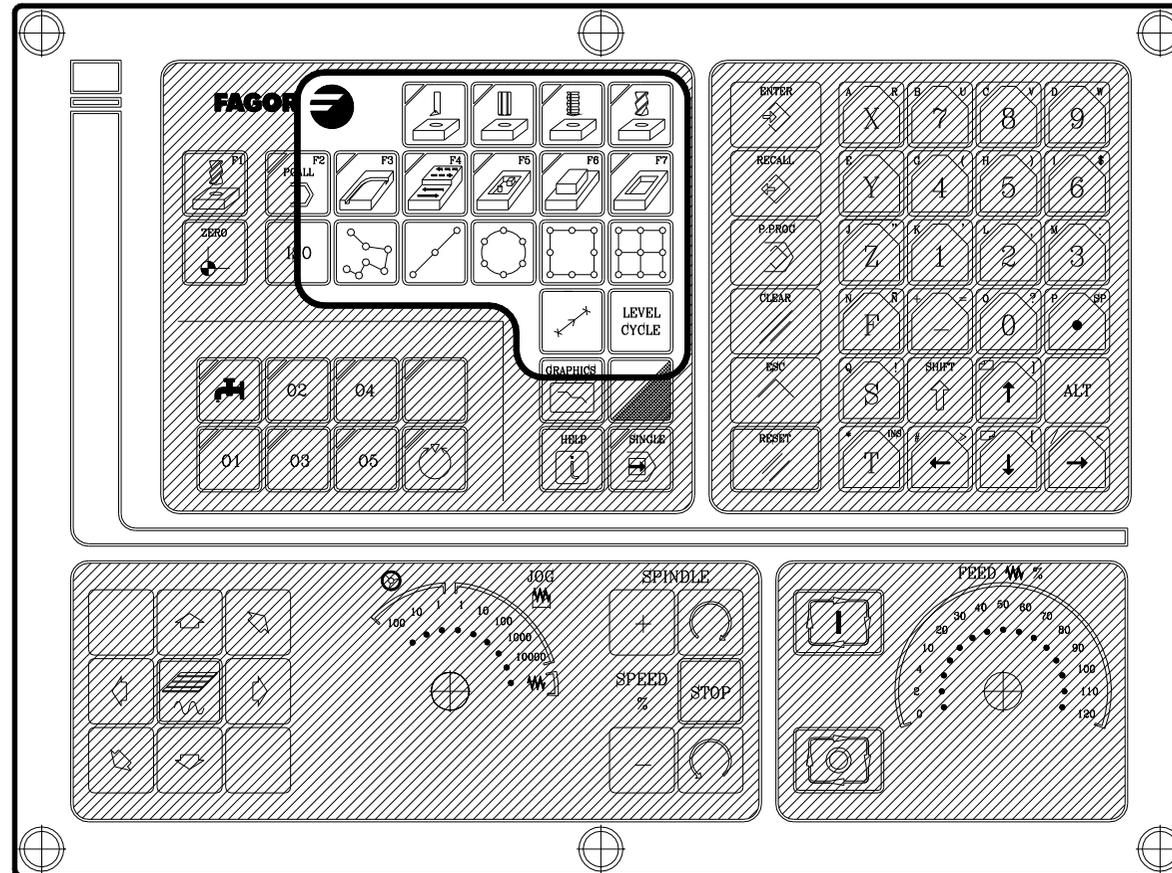
# 4

## OPERATIONS OR CYCLES.

- Keys associated with automatic operations.
- Different work modes.
- Editing the parameters of an operation.
- Simulating an operation.
- Executing an operation.

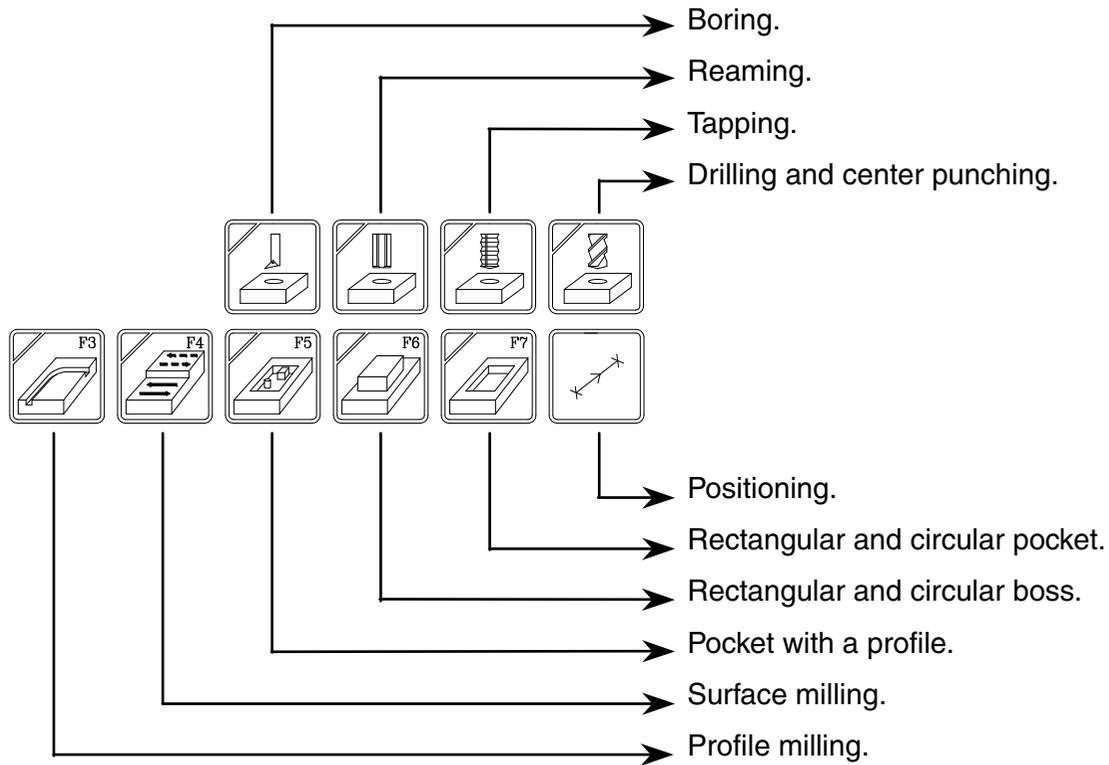


# 4.1 Description of the screen and the keyboard.

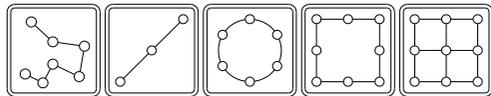


Position of the automatic function keys.

Operation keys.



Selecting the cycle level.

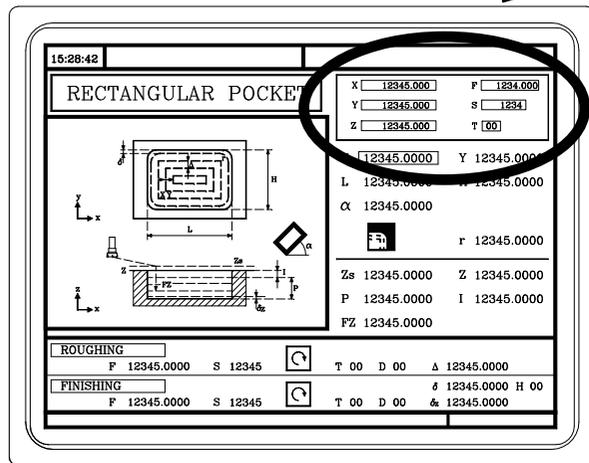


Keys for associating multiple positioning to a cycle so it may be repeated in several positions. Not all the cycles admit multiple positioning.

# 4.2 Work modes.

There are two work modes.

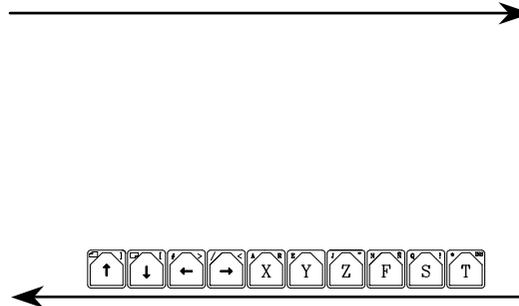
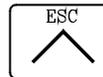
Editing mode.



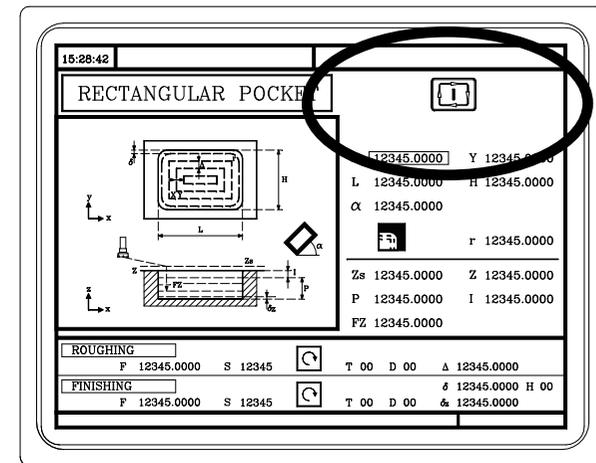
Editing the parameters of the operation or cycle.



Simulating an operation or cycle.



Execution mode.



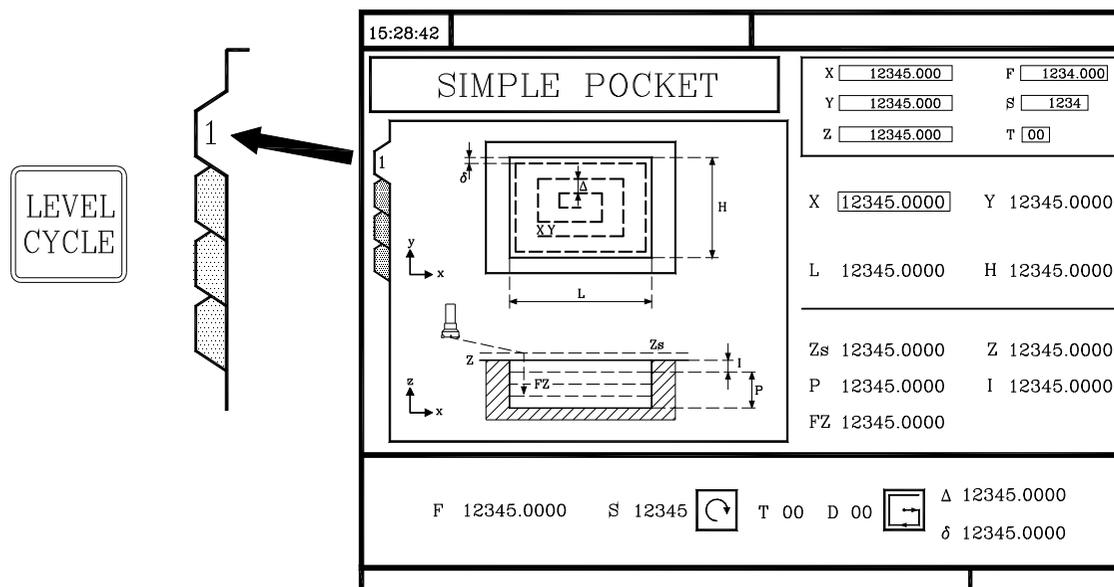
Simulating an operation or cycle.



Executing an operation or cycle.

### 4.3 Operations or cycles.

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



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

### 4.3.1 Editing an operation or cycle.

As an example, select the operation of a rectangular pocket.

**Work cycle.**

**Help graphics.**

**Machining conditions of the cycle.**

**Real position of the axes and cutting conditions.**

**Defining the geometry of the cycle.**

**RECTANGULAR POCKET**

15:28:42

X 12345.000 F 1234.000  
 Y 12345.000 S 1234  
 Z 12345.000 T 00

X 12345.0000 Y 12345.0000  
 L 12345.0000 H 12345.0000  
 α 12345.0000  
 r 12345.0000

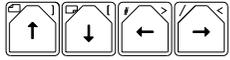
Zs 12345.0000 Z 12345.0000  
 P 12345.0000 I 12345.0000  
 FZ 12345.0000

ROUGHING	F 12345.0000	S 12345	T 00	D 00	Δ 12345.0000
FINISHING	F 12345.0000	S 12345	T 00	D 00	δ 12345.0000 H 00

Use the [LEVEL CYCLE] key to select the cycle level to be executed. Remember that not all the cycles have several levels.

## DEFINING THE DATA OF THE OPERATION OR CYCLE.

To select an icon, data or coordinate:



Move the cursor through the data or icons.



The CNC selects the first coordinate for that axis. Pressing it again selects the second coordinate.



The CNC selects the roughing feedrate. Pressing it again selects the finishing feedrate.



The CNC selects the roughing tool. Pressing it again selects the finishing tool.



The CNC selects the "S" roughing data. Pressing it again selects the "S" finishing data.

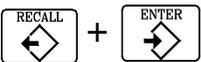
After this selection:



If it is a data, key in the new value and press [ENTER].



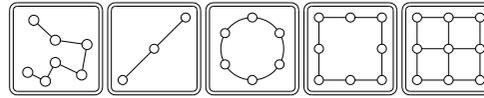
If it is an icon, press the two-color key until choosing the right one and press [ENTER].



If it is a coordinate, there are two choices:

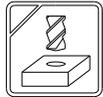
- Key in the new value and press [ENTER].
- Assign the current cursor position to the data. Press [RECALL]+[ENTER].

## ASSOCIATING A MULTIPLE POSITIONING WITH AN OPERATION.

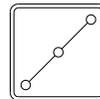


Some cycles may be carried out in the tool position or may be associated a multiple positioning so the cycle may be repeated in several locations. Multiple positioning may be associated with the following cycles.

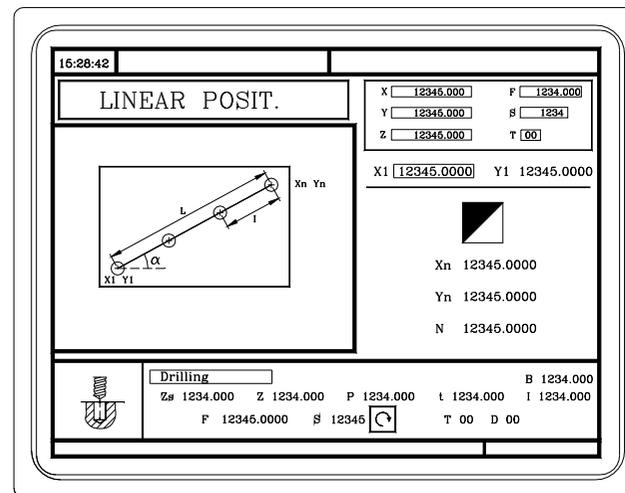
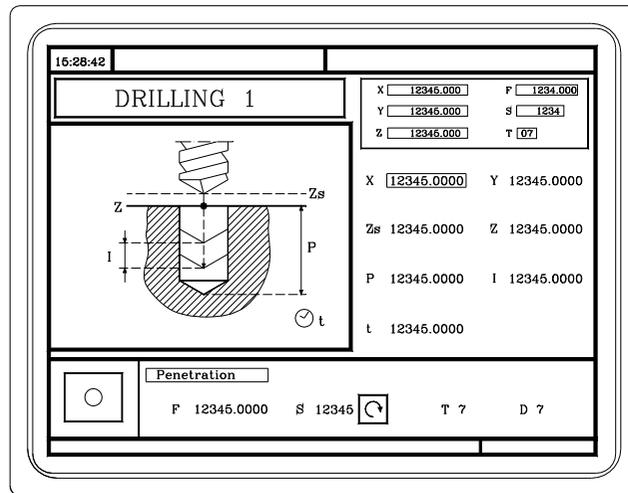
- Boring, reaming, tapping, drilling and center punching operations.
- 2D and 3D pockets.
- Bosses.



Operation or cycle.



Multiple positioning.



Each positioning may be defined in various ways. To select the right data group, place the cursor over the icon and press the two-color key.



## 4.3.2 Simulating an operation or cycle.

Simulation may be used to check the tool paths on the screen.



The CNC shows the graphic simulation window and its associated softkey menu. To access the various functions, use the keys [F1] through [F7].



To start the simulation, press the [START] key.

The simulation speed may be varied with the Feedrate Override switch.

Other useful keys are.



Interrupt the simulation.



If the simulation is interrupted, it resumes it.



If the simulation is interrupted, it cancels it definitely.



Any of these two keys quits the simulation mode.

## GRAPHIC OPTIONS. TYPE OF GRAPHIC.

- "3D" graphics.

It uses color lines to draw the tool path in three dimensions.

- "XY, XZ, YZ" graphics.

It uses color lines to draw the tool path in the selected plane.

- "Combined" graphics.

It divides the screen in four quadrants and draws in each quadrant the graph corresponding to each plane XY, XZ, YZ and in three-dimensions (3D).

- "Top view" graphics.

It draws a solid graph of the XY plane in different shades of gray that indicate the depth of the part. It also shows two sections (XZ and YZ) of the part.

- "Solid" graphics.

It draws a solid three-dimensional image. It starts with an initial block (raw stock). The simulation shows how the tool removes material. It shows the resulting shape of the part.

## GRAPHIC OPTIONS. DISPLAY AREA.

With this option it is possible to define the display area by defining the maximum and minimum values for each axis.



Selecting the coordinates.

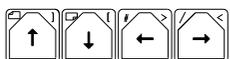


Validating the new values.

## GRAPHIC OPTIONS. ZOOM.

It may be used to enlarge or shrink the drawing or part of it. The new display area is selected using a window super-imposed on the shown tool path.

To enlarge or shrink the drawing, use the softkeys "ZOOM+" and "ZOOM-".



To move the zoom window.



Validating the new values.



To draws the selected area.

To return to the original display area, select the option: "initial value".

## GRAPHIC OPTIONS. GRAPHIC PARAMETERS.

Simulation speed: To select the percentage of simulation speed to apply.

Tool path colors: To change the colors of the tool paths on "3D", "XY, XZ, YZ", "Combined" graphics.

Colors of the solid graphics: To change the colors of the tool and the part in "Top view" and "Solid" graphics.

## **GRAPHIC OPTIONS. CLEAR SCREEN.**

To delete the screen. While in "Solid" graphics mode, it shows the part without being machined (raw stock).

## 4.3.3 Executing an operation or cycle.

Once all the data has been defined, press [ESC]. The CNC screen shows the "cycle start" symbol indicating that the operation may be executed.



This key may be used to choose whether the operations may be executed from beginning to end or one pass at a time.



It starts the execution of the operation or cycle.

Once the execution has started:



Interrupt the execution.



If the execution is interrupted, it resumes it.



If the execution is interrupted, it cancels it definitely.



It goes into graphics mode.

The execution may be interrupted any time, except during the threading operation. In this case, the execution will stop at the end of the threading operation.

## TOOL INSPECTION.

With this option, it is possible to interrupt the execution of an operation to inspect the tool, change its wear, replace it, etc.



- Interrupt the execution.



- Depending on the manufacturer, accessing tool inspection on some machines may require pressing also the [T] key.
- The top of the CNC screen shows the message "INSPECTION". Within the "tool inspection" mode, it is possible to move the axes (jog or handwheels), check the tool, change the tool, stop and start the spindle, change the tool wear, etc.



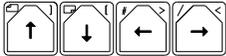
- Repositioning the axes. If more than one axis has been moved, the CNC will request the repositioning command.
- Resuming execution.

## MODIFYING TOOL WEAR.

This option may be used to change the I, K values. The values entered are incremental and are added to the ones stored earlier. This option may be executed during tool inspection or while the machine is running.



- The CNC goes into tool calibration and shows the data for that tool.



- Select the I data with the cursor.



- Key in the I value and press [ENTER].

- Select the K data with the cursor.



- Key in the K value and press [ENTER].

- To change the offset of another tool, select it.



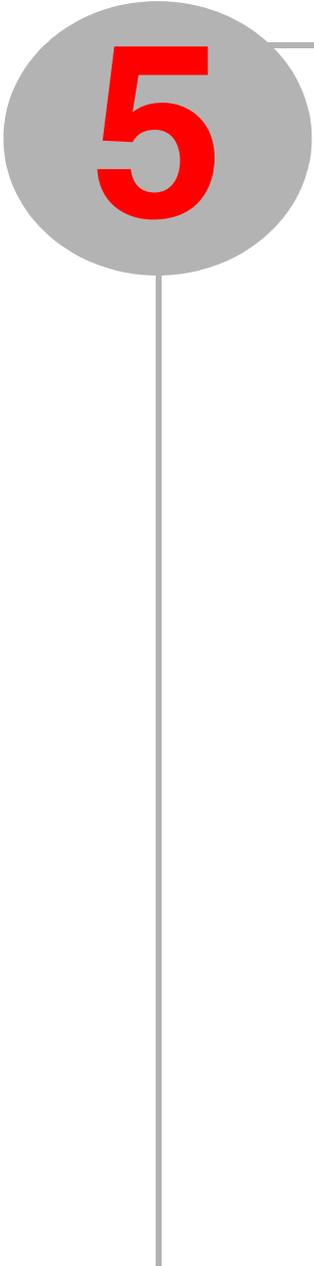
+ (Tool number)

+



- Press [ESC] to end it.

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



**5**

---

**SUMMARY OF WORK CYCLES.**



# 5.1 Profile milling operation.



15:28:42

### PROFILE 1

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	00

X1 12345.0000 Y1 12345.0000

P1 X 12345.0000 r 12345.0000  
Z 12345.0000

P2 X 12345.0000 C 12345.0000  
Z 12345.0000

Xn 12345.0000 Yn 12345.0000

Zs 12345.0000 Z 12345.0000

P 12345.0000 I 12345.0000

FZ 12345.0000

ROUGHING		↻	✎						
F	12345.0000	S	12345	T	00	D	00		
FINISHING		↻	✎						
F	12345.0000	S	12345	T	00	D	00	delta	12345.0000



15:28:42

### PROFILE MILLING

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	00

X 12345.0000 Y 12345.0000

PROFILE 000

Zs 12345.0000 Z 12345.0000

P 12345.0000 I 12345.0000

FZ 12345.0000

ROUGHING		↻	✎						
F	12345.0000	S	12345	T	00	D	00		
FINISHING		↻	✎						
F	12345.0000	S	12345	T	00	D	00	delta	12345.0000

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

At this cycle level, the profile is defined using the profile editor.

# 5.2 Surface milling and slot milling operation.



15:28:42

### BIDIR. MILLING ON X

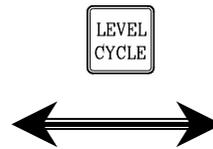
X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	00

X1	12345.0000	Y1	12345.0000
L	12345.0000	H	12345.0000
E	12345.0000		

Zs	12345.0000	Z	12345.0000
P	12345.0000	I	12345.0000
FZ	12345.0000	delta_z	12345.0000

ROUGHING

F 12345.0000 S 12345 ↻ T 00 D 00 Δ 12345.0000



15:28:42

### SLOT MILLING

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	00

X	12345.0000	Y	12345.0000
L	12345.0000	H	12345.0000
E	12345.0000	alpha	12345.0000

Zs	12345.0000	Z	12345.0000
P	12345.0000	I	12345.0000
FZ	12345.0000		

ROUGHING

F 12345.0000 S 12345 ↻ T 00 D 00 ↔ Δ 12345.0000

FINISHING

F 12345.0000 S 12345 ↻ T 00 D 00 ↔ delta 12345.0000 N 00

delta 12345.0000

# 5.3 Pocket cycle with a profile



15:28:42

### 2D PROFILE POCKET

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	00

PROFILE 000

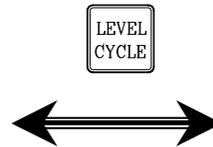
Zs 12345.0000    Z 12345.0000

P 12345.0000    I 12345.0000

FZ 12345.0000

ROUGHING			T 00 D 00		$\beta$ 0.0000	$\Delta$ 12345.0000
F	12345.0000	S	12345			
FINISHING			T 00 D 00	$\delta$ 12345.0000	N 00	
F	12345.0000	S	12345	$\Theta$ 0.0000	$\phi$ 12345.0000	

Pocket with 2D profile.



15:28:42

### 3D PROFILE POCKET

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	00

**3D POCKET 288**

P. XY 2

P. XZ1 3    P. XZ2 0

P. XZ3 0    P. XZ4 0

Zs 12345.0000    Z 12345.0000

P 12345.0000    I1 12345.0000

FZ 12345.0000    I2 12345.0000

ROUGHING			T 00 D 00	$\beta$ 0.0000	$\Delta$ 12345.0000
F	12345.0000	S	12345		
FINISHING			T 00 D 00	$\delta$ 12345.0000	R 0.0000
F	12345.0000	S	12345	$\epsilon$ 12345.0000	

Pocket with 3D profile.

# 5.4 Rectangular and circular boss cycles.



15:28:42

### RECTANGULAR BOSS

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	00
X	12345.0000	Y	12345.0000
L	12345.0000	H	12345.0000
alpha	12345.0000	Q	12345.0000
r	12345.0000		
Zs	12345.0000	Z	12345.0000
P	12345.0000	I	12345.0000
FZ	12345.0000		

ROUGHING				↻
F	12345.0000	S	12345	T 00 D 00 Δ 12345.0000
FINISHING				↻
F	12345.0000	S	12345	δ 12345.0000 H 00 δz 12345.0000

Rectangular boss.



15:28:42

### CIRCULAR BOSS

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	00
Xc	12345.0000	Yc	12345.0000
R	12345.0000	Q	12345.0000
Zs	12345.0000	Z	12345.0000
P	12345.0000	I	12345.0000
FZ	12345.0000		

ROUGHING				↻
F	12345.0000	S	12345	T 00 D 00 Δ 12345.0000
FINISHING				↻
F	12345.0000	S	12345	δ 12345.0000 H 00 δz 12345.0000

Circular boss.

# 5.5 Rectangular and circular pocket cycles.



15:28:42

### SIMPLE POCKET

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	00

X 12345.0000 Y 12345.0000

L 12345.0000 H 12345.0000

---

Zs 12345.0000 Z 12345.0000

P 12345.0000 I 12345.0000

FZ 12345.0000

F	12345.0000	S	12345		T	00	D	00	Δ	12345.0000
δ 12345.0000										

Simple pocket.



15:28:42

### RECTANGULAR POCKET

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	00

X 12345.0000 Y 12345.0000

L 12345.0000 H 12345.0000

α 12345.0000

r 12345.0000

Zs 12345.0000 Z 12345.0000

P 12345.0000 I 12345.0000

F'Z 12345.0000

<b>ROUGHING</b>										
F	12345.0000	S	12345		T	00	D	00	Δ	12345.0000
<b>FINISHING</b>										
F	12345.0000	S	12345		T	00	D	00	δ	12345.0000 H 00
δz 12345.0000										

Rectangular pocket.



At this level, it is possible to choose the type of corner and the inclination angle of the pocket with respect to the abscissa axis.

15:28:42

### CIRCULAR POCKET 1

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	00

Xc	12345.0000
Yc	12345.0000
R	12345.0000

---

Zs	12345.0000	Z	12345.0000
P	12345.0000	I	12345.0000
FZ	12345.0000		

<b>ROUGHING</b>	F 12345.0000	S 12345		T 00	D 00	β 0.0000	Δ 12345.0000
<b>FINISHING</b>	F 12345.0000	S 12345		T 00	D 00	δ 12345.0000	N 00

Circular pocket.

LEVEL  
CYCLE

⇒

15:28:42

### CIRCULAR POCKET 2

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	00

Xc	12345.0000	Yc	12345.0000
Re	12345.0000	Ri	12345.0000

---

Zs	12345.0000	Z	12345.0000
P	12345.0000	I	12345.0000
FZ	12345.0000		

<b>ROUGHING</b>	F 12345.0000	S 12345		T 00	D 00	β 0.0000	Δ 12345.0000
<b>FINISHING</b>	F 12345.0000	S 12345		T 00	D 00	δ 12345.0000	N 00

Pre-empted pocket.

LEVEL  
CYCLE

⇒

# 5.6 Positioning.



15:28:42

### POSITIONING 1

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	00

X 12345.0000  
Y 12345.0000  
Z 12345.0000

F 12345.000 S 12345 T 00 D 00



15:28:42

### POSITIONING 2

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	00

X 12345.0000 Y 12345.0000  
Z 12345.0000

Func.Aux.Anter. Func.Aux.Post.

****	****
****	****
****	****
****	****

F 12345.000 S 12345 T 00 D 00

At this cycle level, it is possible to define the auxiliary functions that will be executed before or after the movement.

# 5.7 Boring operation.



15:28:42

### BORING 1

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	04

X	12345.0000	Y	12345.0000
Zs	12345.0000	Z	12345.0000
P	12345.0000	t	12345.0000

Penetration: F 12345.0000 S 12345 T 4 D 4

Exit: [Upward arrow icon]



15:28:42

### BORING 2

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	04

X	12345.0000	Y	12345.0000
Zs	12345.0000	Z	12345.0000
P	12345.0000	t	12345.0000

$\alpha_s$  12345.0000

$\Delta x$  12345.0000  $\Delta y$  12345.0000

Penetration: F 12345.0000 S 12345 T 4 D 4

This level is only available when using spindle orientation. This level may be used, after the quail has penetrated, to orient the spindle, retract the quail before the exit movement, thus preventing the part from scratching.

# 5.8 Reaming operation.



15:28:42

## REAMING

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	05

X	12345.0000	Y	12345.0000
Zs	12345.0000	Z	12345.0000
P	12345.0000	I	12345.0000

Penetration

F	12345.0000	S	12345	T	5	D	5
---	------------	---	-------	---	---	---	---

# 5.9 Tapping operation.



15:28:42

### TAPPING

X	00044.000	F	1.000
Y	00022.003	S	150
Z	-00397.490	T	3

X	0.0000	Y	0.0000
Zs	0.0000	Z	0.0000
P	0.0000	t	0.0000

Penetration

F	0.000	S	150	T	3	D	3
---	-------	---	-----	---	---	---	---

# 5.10 Drilling and center punching operation.



15:28:42

### DRILLING 1

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	[07]

X 12345.0000 Y 12345.0000  
Zs 12345.0000 Z 12345.0000  
P 12345.0000 I 12345.0000  
t 12345.0000

Penetration

F 12345.0000 S 12345 T 7 D 7

Drilling.



15:28:42

### DRILLING 2

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	[07]

X 12345.0000 Y 12345.0000  
Zs 12345.0000 Z 12345.0000  
P 12345.0000 I 12345.0000  
t 12345.0000 B 12345.0000

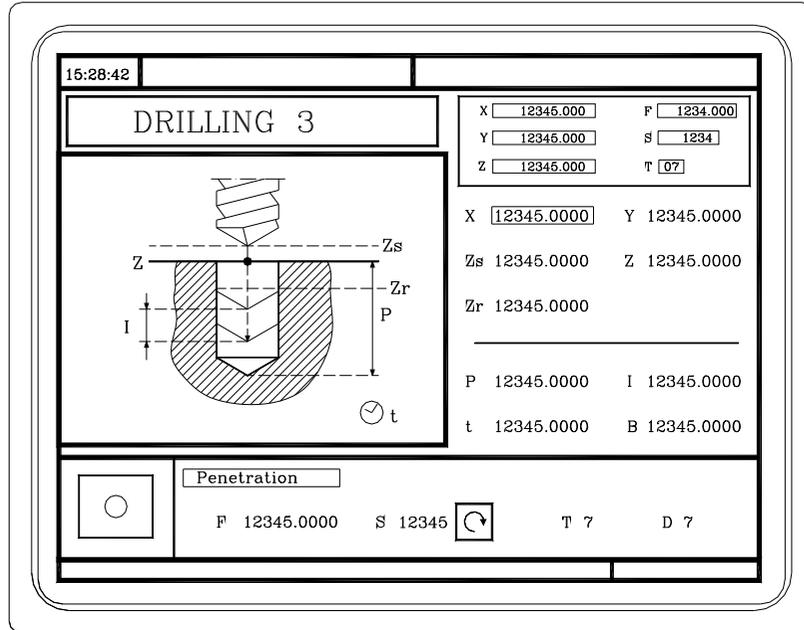
Penetration

F 12345.0000 S 12345 T 7 D 7

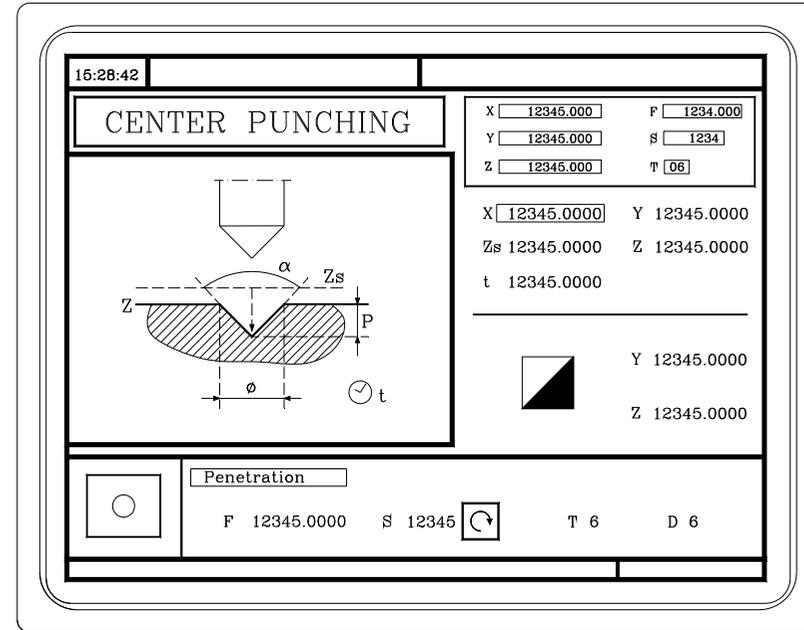
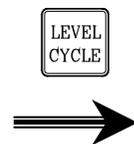
Drilling.



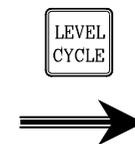
At this level, it is possible to set the withdrawal distance (B) for the tool after each penetration (drilling peck).



Drilling.



Center punching.



At this level, it is possible to set the withdrawal coordinate (Zr) after each penetration.

# 5.11 Multiple positioning in several locations.



15:28:42

**RANDOM POSIT.**

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	00

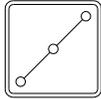
  

X1	12345.0000	Y1	12345.0000
X1	12345.0000	Y1	12345.0000
X1	12345.0000	Y1	12345.0000
X1	12345.0000	Y1	12345.0000
X1	12345.0000	Y1	12345.0000
X1	12345.0000	Y1	12345.0000

	Drilling	B	1234.000						
Zs	1234.000	Z	1234.000	P	1234.000	t	1234.000	I	1234.000
F	12345.0000	S	12345		T	00	D	00	

# 5.12 Multiple positioning in a straight line.



15:28:42

### LINEAR POSIT.

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	00

X1 12345.0000    Y1 12345.0000

Xn 12345.0000  
Yn 12345.0000  
N 12345.0000

Drilling		B	1234.000
Zs	1234.000	Z	1234.000
P	1234.000	t	1234.000
I	1234.000	F	12345.0000
S	12345	T	00
D	00		

# 5.13 Multiple positioning in an arc.



15:28:42

### ARC POSIT.

X 12345.000 F 1234.000  
Y 12345.000 S 1234  
Z 12345.000 T 00

X1 12345.0000 Y1 12345.0000

Xc 12345.0000  
Yc 12345.0000  
N 12345.0000  
T 12345.0000

Drilling B 1234.000  
Zs 1234.000 Z 1234.000 P 1234.000 t 1234.000 I 1234.000  
F 12345.0000 S 12345 T 00 D 00

# 5.14 Multiple positioning in a rectangular pattern.



15:28:42

RECTANGULAR POSIT.

X 12345.000 F 1234.000  
Y 12345.000 S 1234  
Z 12345.000 T 00

X1 12345.0000 Y1 12345.0000

LX 12345.0000 LY 12345.0000  
NX 12345.0000 NY 12345.0000

$\alpha$  12345.0000  $\beta$  12345.0000

Drilling B 1234.000  
Zs 1234.000 Z 1234.000 P 1234.000 t 1234.000 I 1234.000  
F 12345.0000 S 12345 T 00 D 00

# 5.15 Multiple positioning in a grid pattern.



15:28:42

GRID PATTERN POSIT.

X <input type="text" value="12345.000"/>	F <input type="text" value="1234.000"/>
Y <input type="text" value="12345.000"/>	S <input type="text" value="1234"/>
Z <input type="text" value="12345.000"/>	T <input type="text" value="00"/>
X1 <input type="text" value="12345.0000"/> Y1 <input type="text" value="12345.0000"/>	
LX <input type="text" value="12345.0000"/>	LY <input type="text" value="12345.0000"/>
NX <input type="text" value="12345.0000"/>	NY <input type="text" value="12345.0000"/>
alpha <input type="text" value="12345.0000"/>	beta <input type="text" value="12345.0000"/>

Drilling

B

Zs

Z

P

t

I

F

S

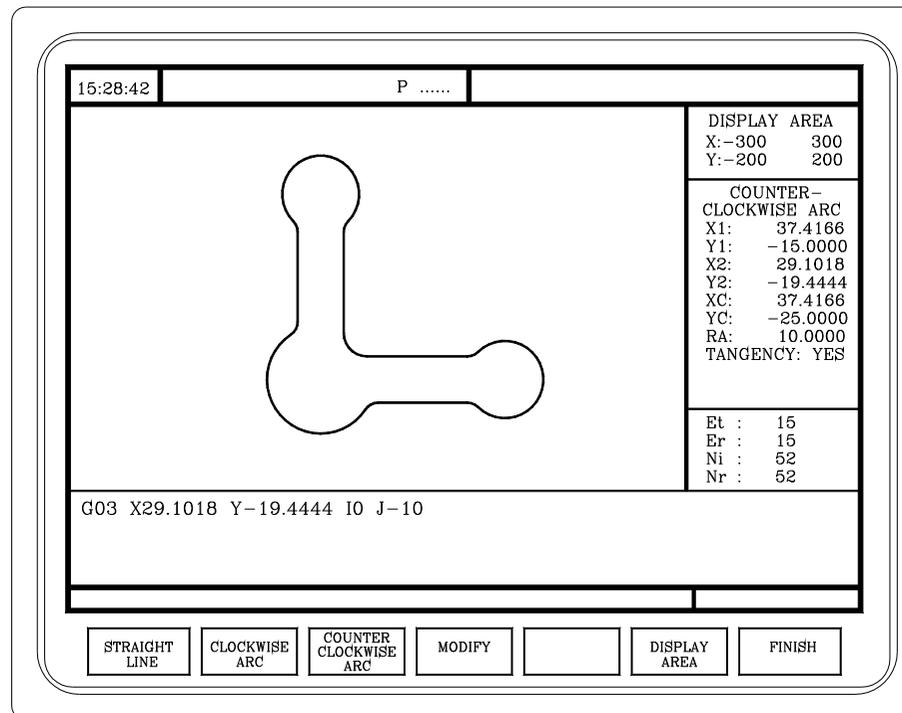
T

D

## 5.16 Profile editor.

With the profile editor, it is possible to define the profiles in the "profile milling" and "pocket with a profile" cycles.

With the profile editor, it is possible to define straight and curve sections of the profile (the editor solves intersection and tangent related problems) and then change these sections adding rounding, chamfers and tangential entries and exits.



# 6

## CONVERSATIONAL PART PROGRAMS.

- What is a conversational part-program?.
- How to edit a conversational part-program.
- How to modify a conversational part-program (insert or delete operations).
- Simulating/executing an operation.
- Simulating/executing a part-program starting with an operation.
- Simulating/executing a part-program.
- Copying a part-program.
- Deleting a new part program.



# 6.1 What is a conversational part-program?

A conversational part-program consists in a set of operations ordered sequentially. Each operation is defined separately and they are then saved one after another in a program. The name of the part-program may be a number between 1 and 899999.

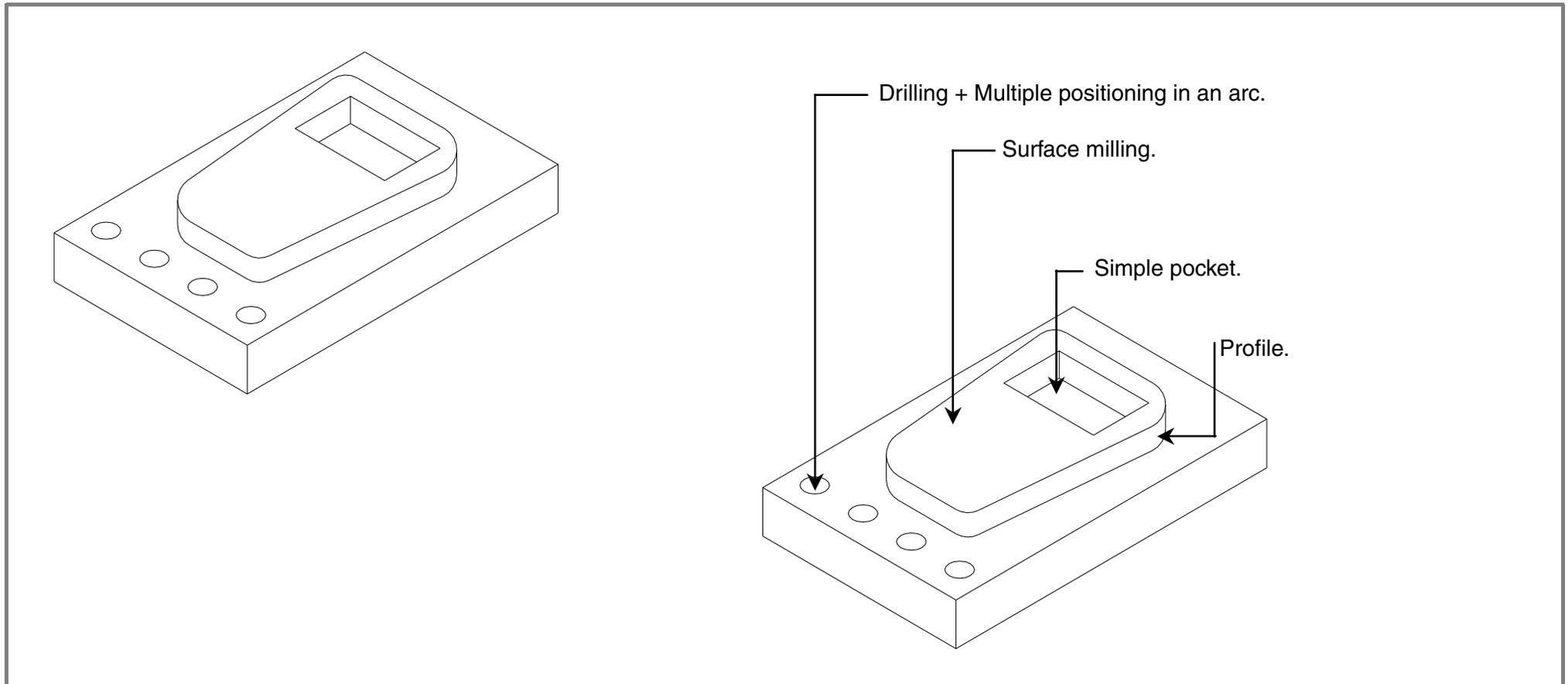
The diagram on the left shows a cylindrical part with several features: a surface mill on top, a circular pocket, a circular boss, and a rectangular pocket. Arrows point to these features with labels: "Surface milling.", "Circular pocket.", "Drilling + Multiple positioning in an arc.", "Circular boss.", and "Rectangular pocket.".

The screenshot on the right shows a conversational programming interface. At the top, it displays the time "15:28:42" and the program name "P000555". Below this is a table with two columns: "PART - PROGRAMS" and "CYCLES".

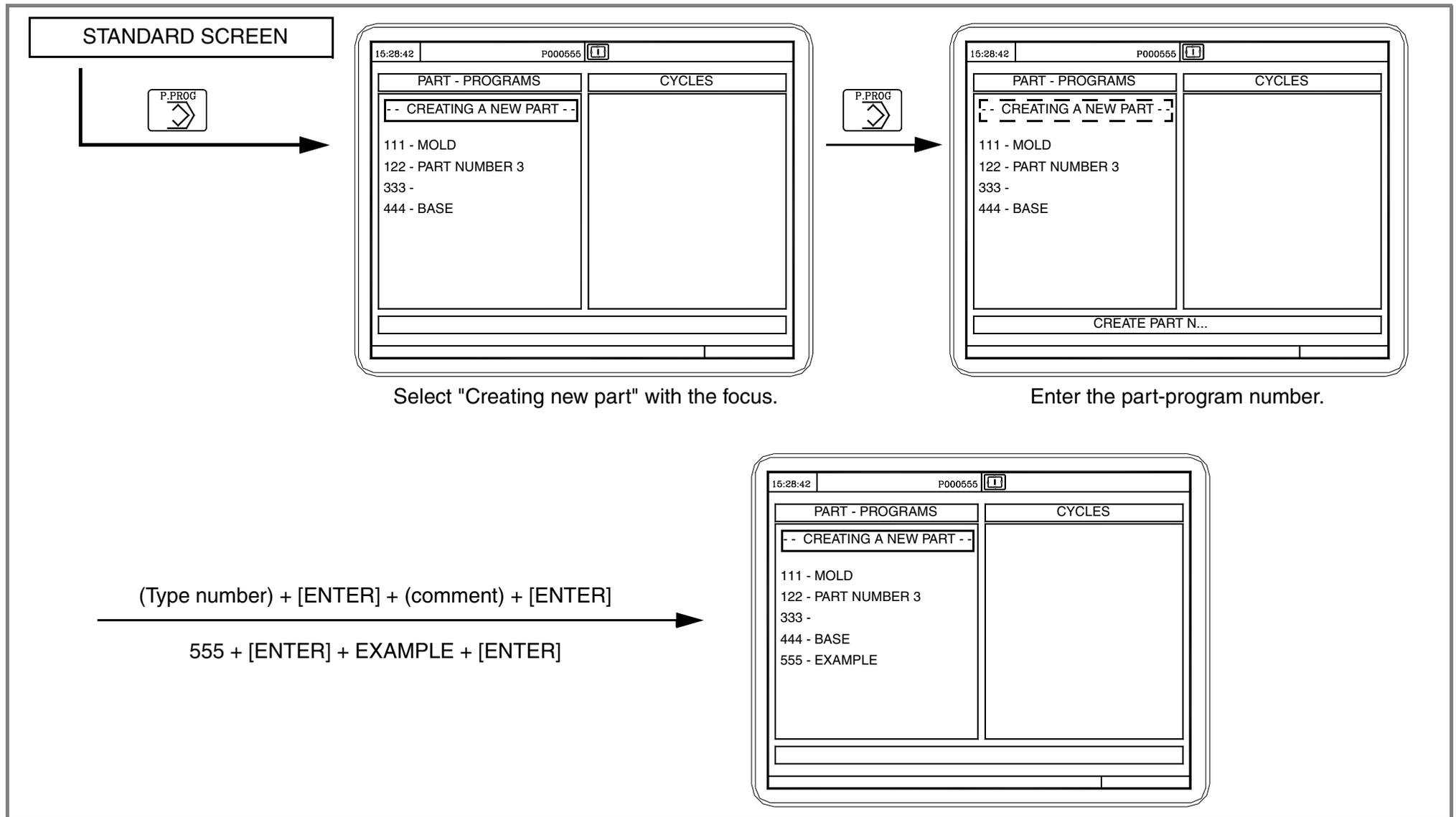
PART - PROGRAMS	CYCLES
--- CREATING A NEW PART ---	1 - POSITIONING 1
111 - MOLD	2 - BIDIR SURF MILL IN X
122 - PART NUMBER 3	3 - CIRCULAR POCKET
333 -	4 - CIRCULAR BOSS
	5 - RECTANGULAR POCKET
	6 - DRILLING1+POSIT. IN ARC

## 6.2 Editing a part-program.

To edit the part-program, first select the operations required to execute the part and the order (sequence) in which they will be executed. A part may be executed in different ways.

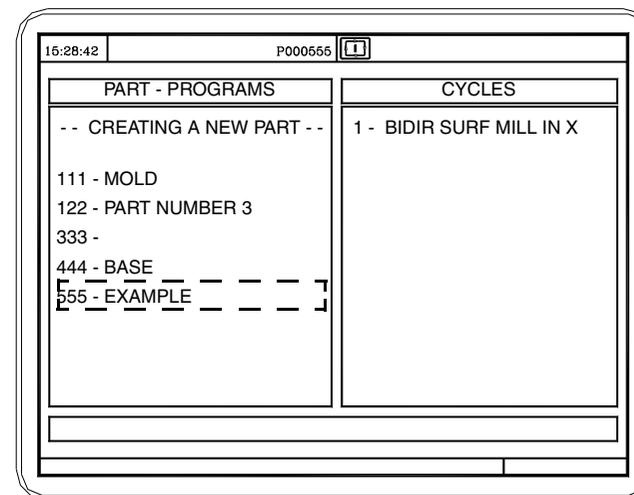
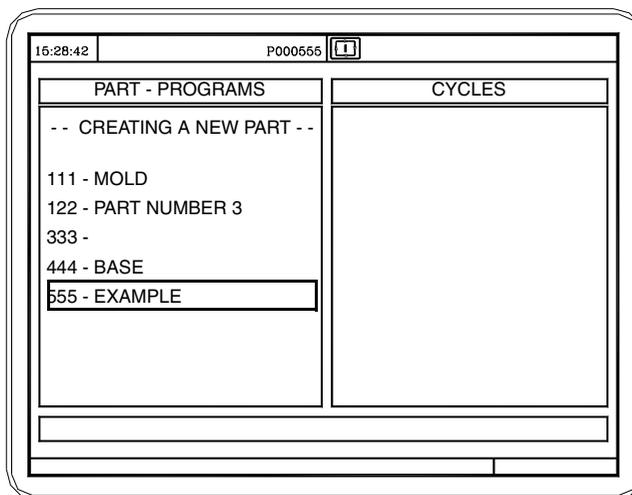


Once the sequence of operations has been chosen, the part program is created by editing the operations one by one.



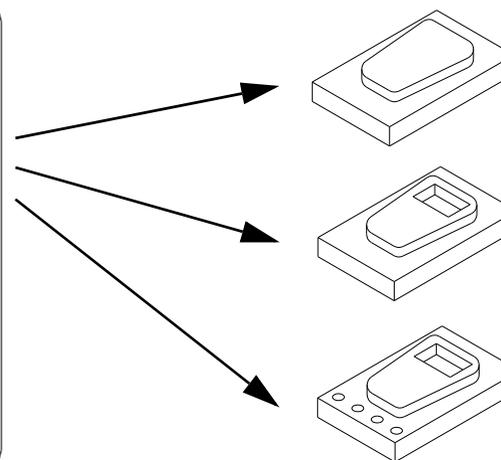
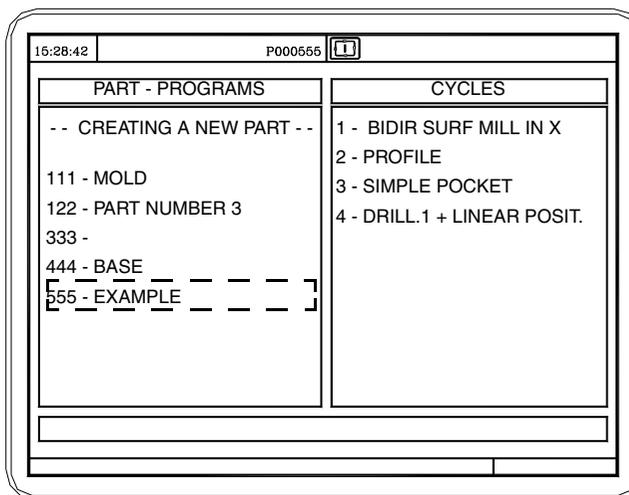


Select an operation and define its parameters. When the operation is defined, press [P.PROG].



Select the program with the focus.

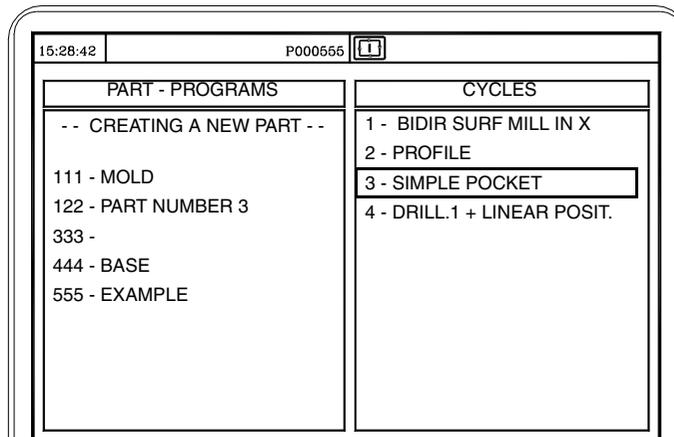
Repeat these steps for the other operations. In this case, the part-program will be as follows.



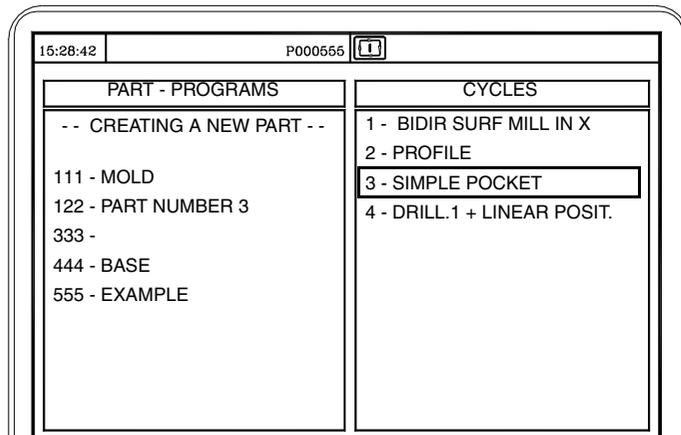
### 6.3 Modifying a part-program.

#### Replacing an operation.

Select an operation and press [RECALL].



The CNC shows the operation with all the data. Modify the parameters like in editing mode. After modifying the data, press [P.PROG].



The CNC will request an option. Select the "REPLACE" option and press [ENTER].



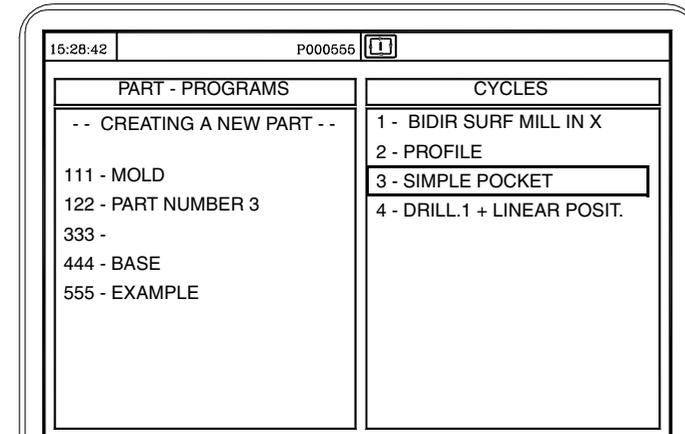
The new operation replaces the previous one.

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

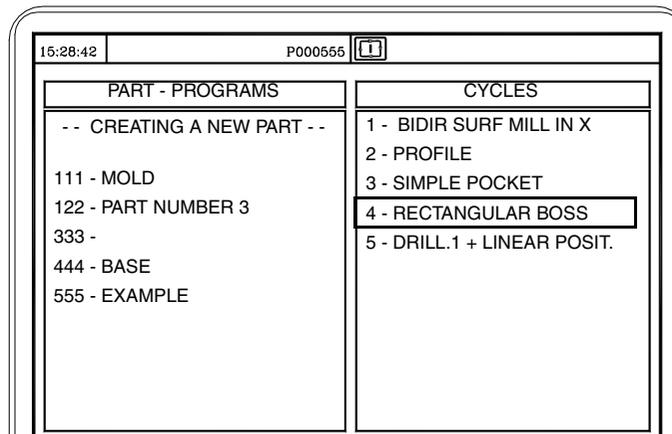
## Inserting an operation.

Select an operation.

Define the parameters of the operation and press [P.PROG] to access the part-program.

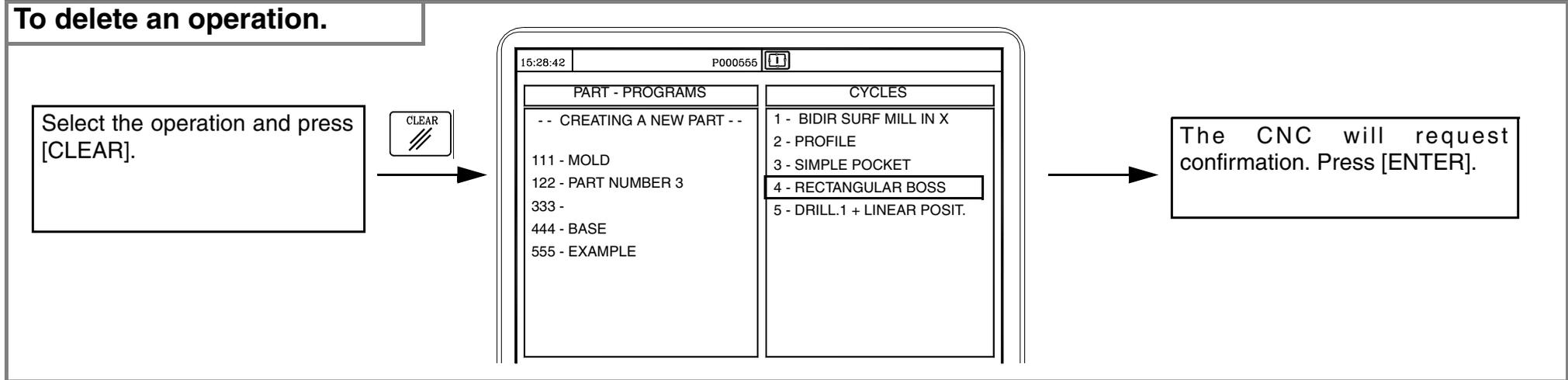


Select the position where the operation is to be inserted and press [ENTER].



The new operation is inserted after the selected position.

Operations can also be deleted from a part-program.



The position of an operation can also be changed.

## Changing the position of an operation.

Select the operation to be relocated and press the two-color key.

PART - PROGRAMS	CYCLES
-- CREATING A NEW PART --	1 - BIDIR SURF MILL IN X
111 - MOLD	2 - PROFILE
122 - PART NUMBER 3	3 - SIMPLE POCKET
333 -	4 - DRILL.1 + LINEAR POSIT.
444 - BASE	
555 - EXAMPLE	



Select the new position for the operation and press [ENTER].

PART - PROGRAMS	CYCLES
-- CREATING A NEW PART --	1 - BIDIR SURF MILL IN X
111 - MOLD	2 - PROFILE
122 - PART NUMBER 3	3 - SIMPLE POCKET
333 -	4 - DRILL.1 + LINEAR POSIT.
444 - BASE	
555 - EXAMPLE	

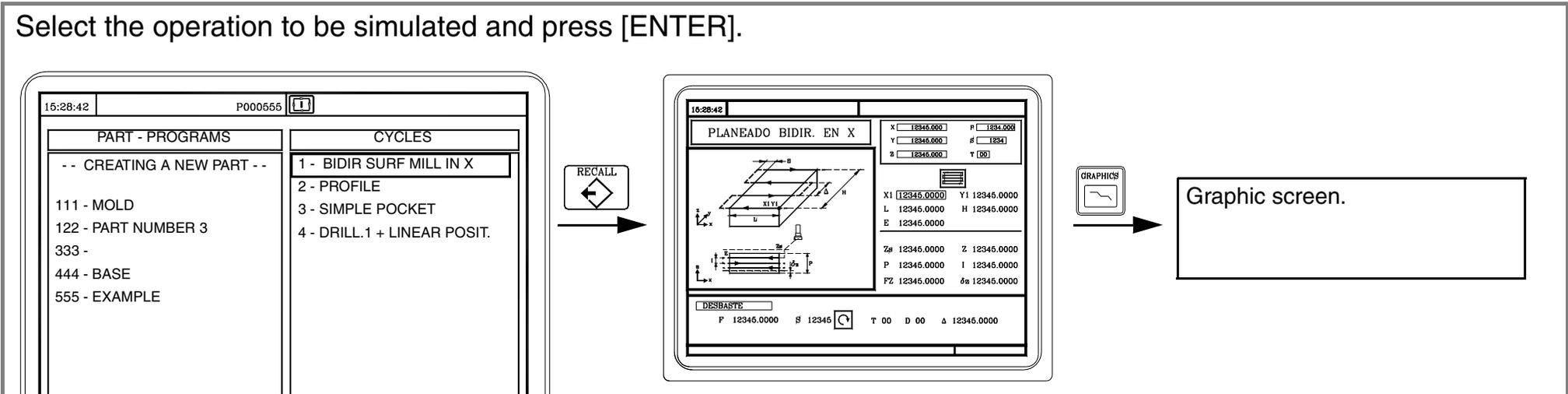


PART - PROGRAMS	CYCLES
-- CREATING A NEW PART --	1 - PROFILE
111 - MOLD	2 - SIMPLE POCKET
122 - PART NUMBER 3	3 - BIDIR SURF MILL IN X
333 -	4 - DRILL.1 + LINEAR POSIT.
444 - BASE	
555 - EXAMPLE	

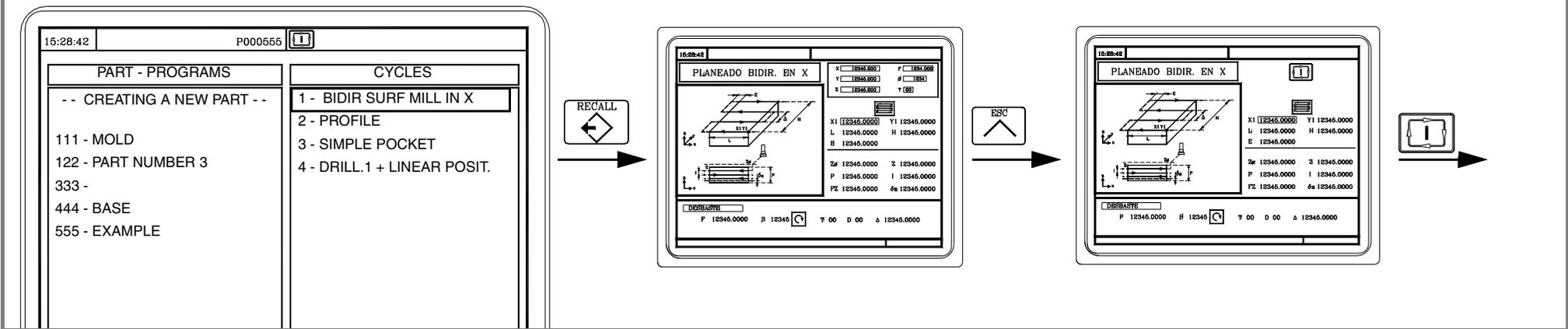
The new operation is inserted after the selected position.

# 6.4 Simulating/executing an operation.

Select the operation to be simulated and press [ENTER].

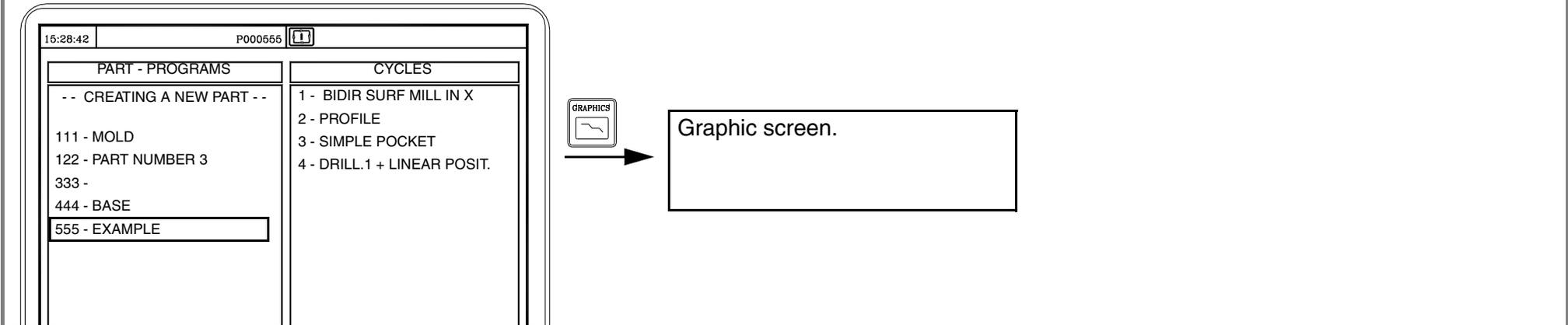


Select the operation to be simulated and press [ENTER].

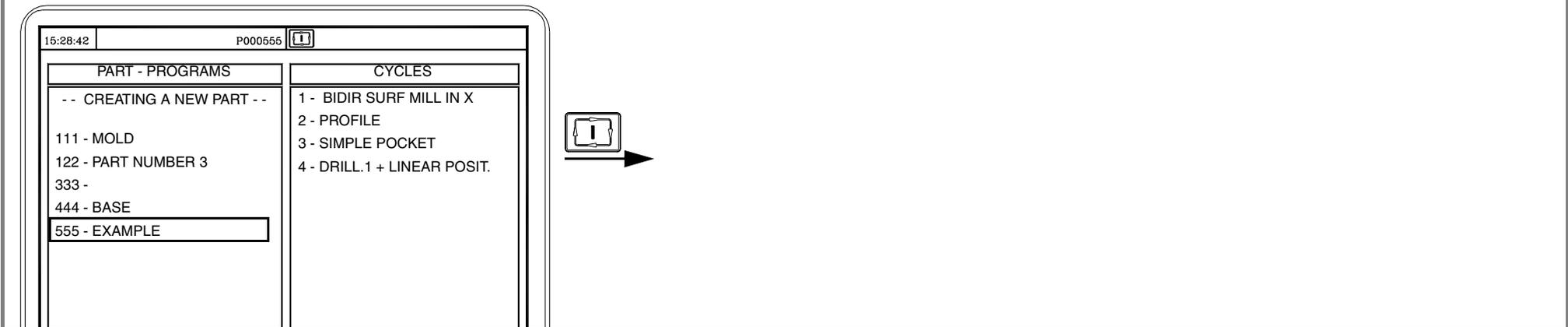


## 6.5 Simulating/executing a part-program.

Select the part-program to be simulated and press [GRAPHICS].

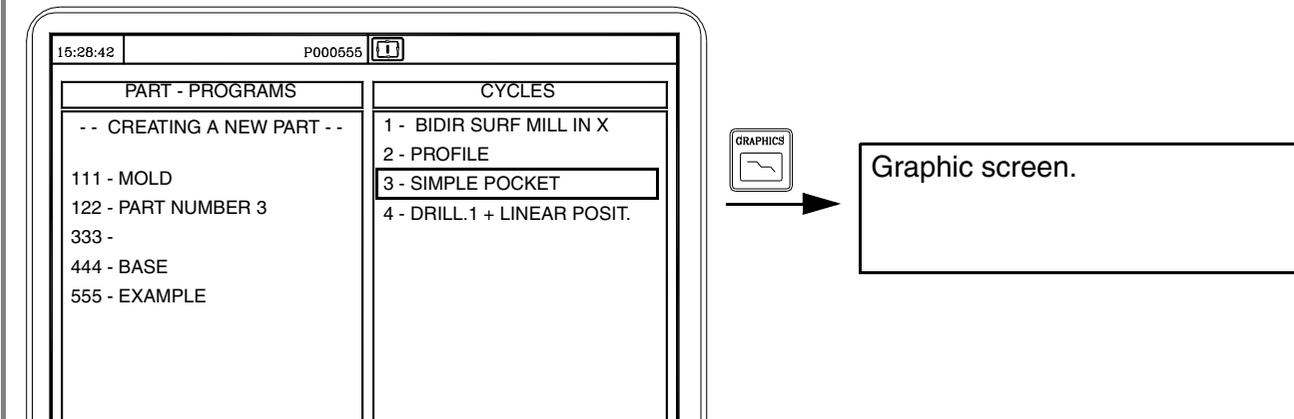


Select the part-program to be executed and press [START].

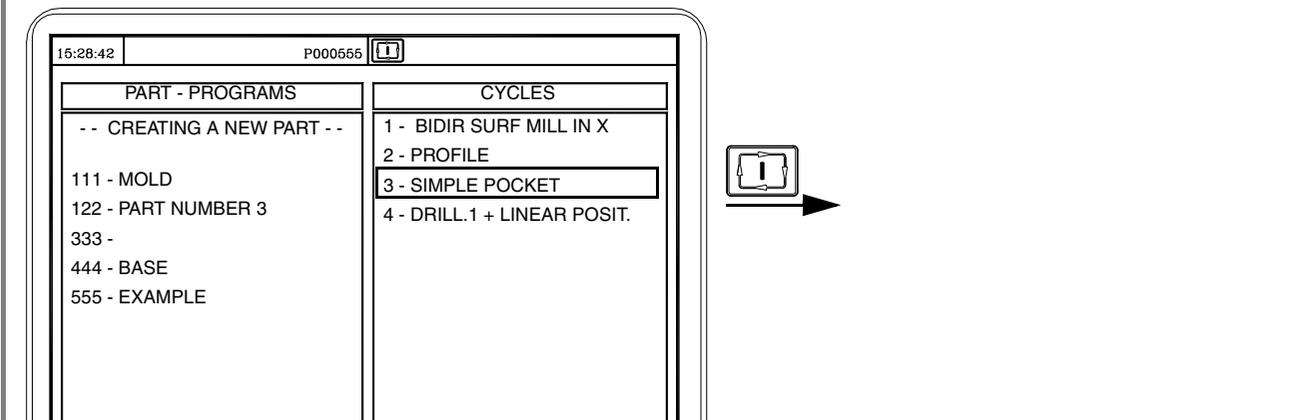


## 6.6 Simulating/executing a part-program starting with an operation.

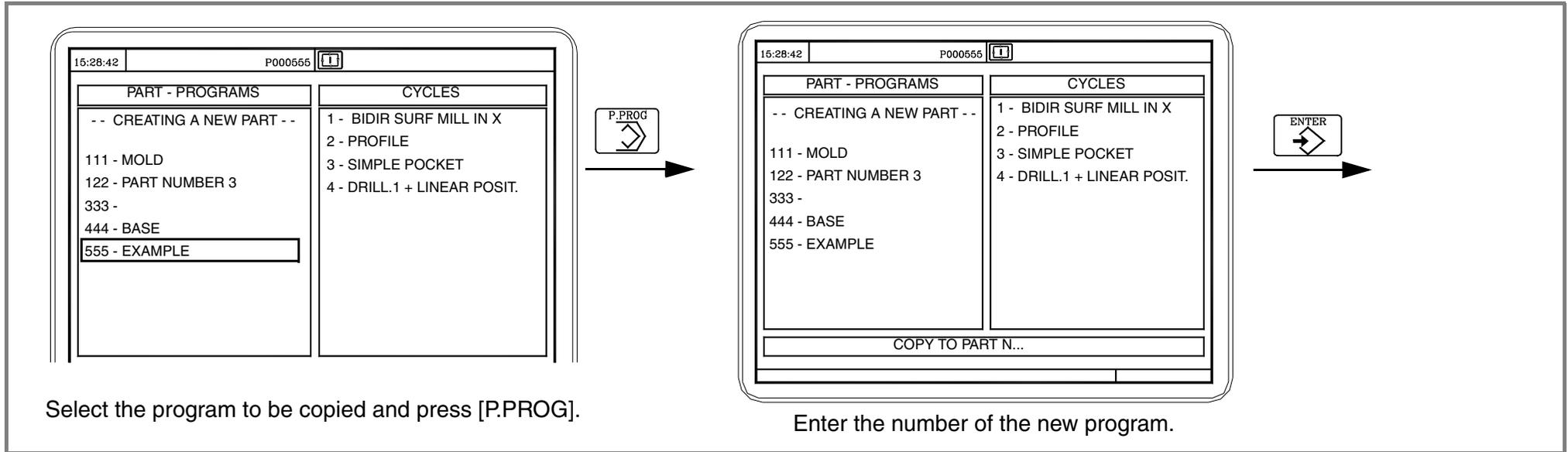
Select the first operation to start simulating and press [GRAPHICS].



Select the first operation to start executing and press [START].



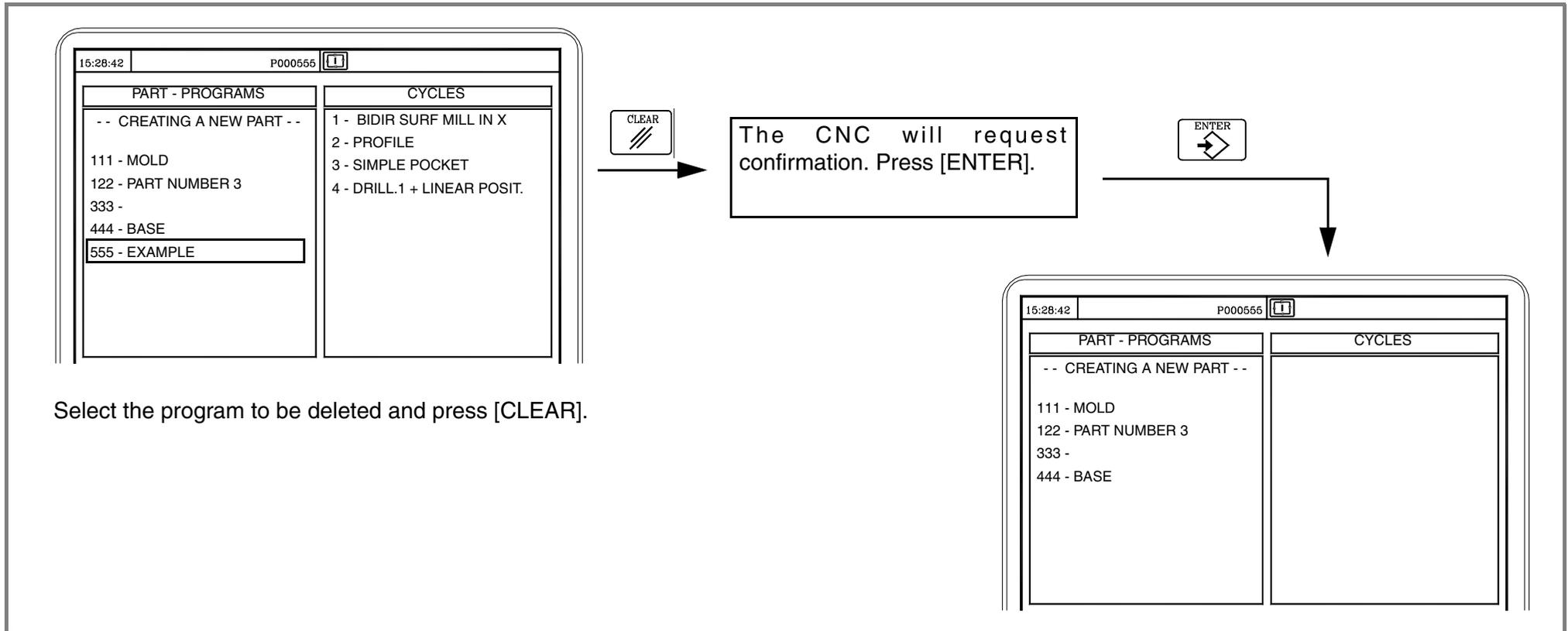
# 6.7 Copying a part-program into another one.



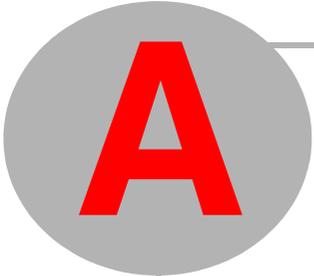
Select the program to be copied and press [P.PROG].

Enter the number of the new program.

# 6.8 Deleting a new part program.







**A**

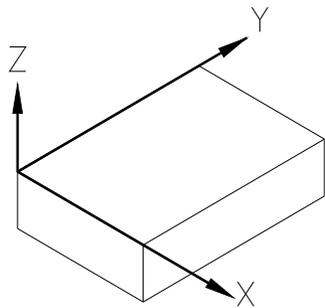
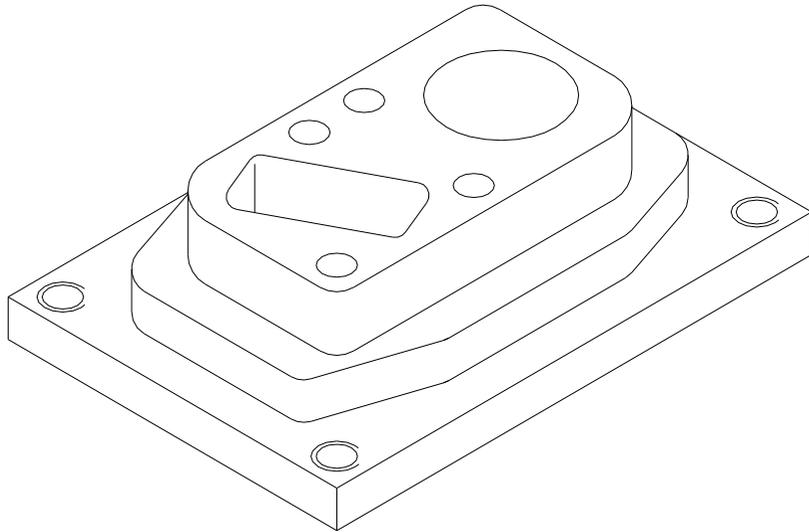
---

**PROGRAMMING EXAMPLE.**





## Step 0: Part to be machined.



## PRIOR CONSIDERATIONS.

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

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

- T1: End mill  $\text{Ø}40$ .
- T2: End mill  $\text{Ø}25$ .
- T3: End mill  $\text{Ø}10$ .
- T4: Center punching drill bit.
- T5: Drill  $\text{Ø}8$ .
- T6: Drill bit  $\text{Ø}5$ .
- T7: Tap M-6.

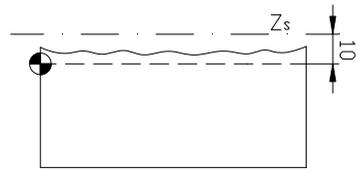
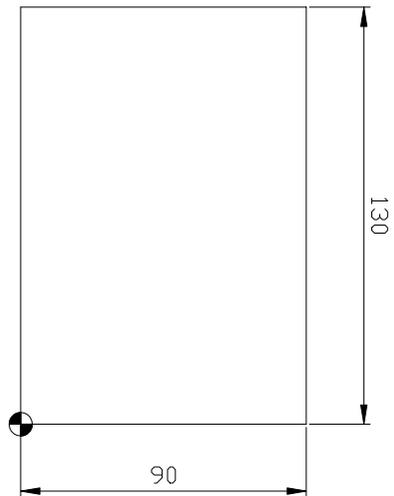
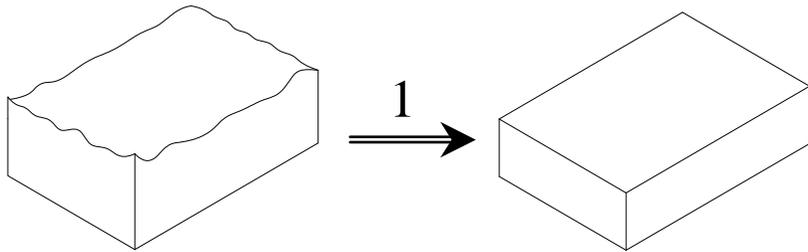
The spindle speed and feedrate are approximate, they may be different from the ones shown here.



This symbol represents the position of the part zero.

# .MC. Option

## Step 1: Surface milling.



F4

15:28:42

BIDIR. MILLING ON X

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	[00]

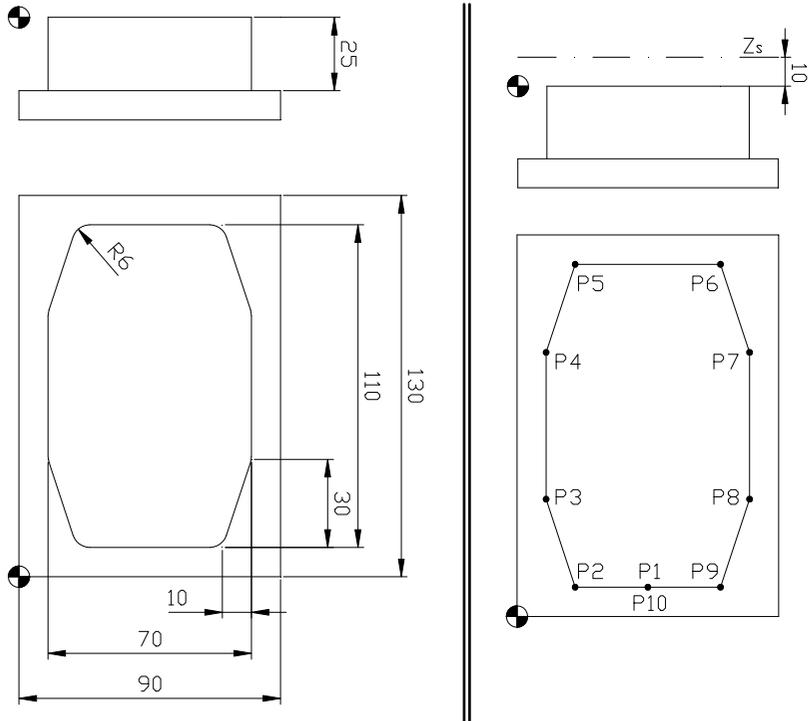
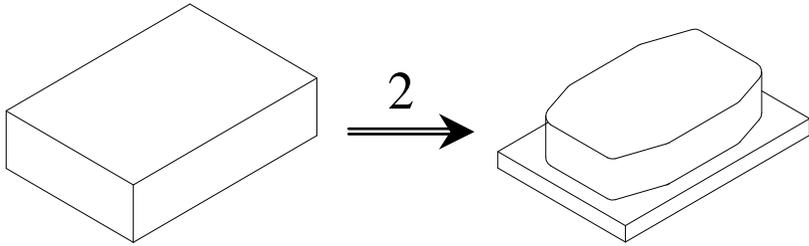
X1	0.0000	Y1	0.0000
L	90.0000	H	130.0000
E	25.0000		
Zs	10.0000	Z	4.0000
P	4.0000	I	2.2500
FZ	90.0000	δz	0.5000

ROUGHING

F 110.0000 S 720

T 1 D 1 Δ 15.0000

## Step 2: machining of the profile



LEVEL  
CYCLE

15:28:42

**PROFILE 1**

X	12345.000	P	1234.000
Y	12345.000	β	1234
Z	12345.000	T	00

X1	45.0000	Y1	-30.0000
P1 X	45.0000	r	30.0000
Y	10.0000		
P2 X	20.0000	r	6.0000
Y	10.0000		
Xn	45.0000	Yn	-30.0000
Zs	10.0000	Z	0.0000
P	25.0000	I	5.0000
PZ	90.0000		

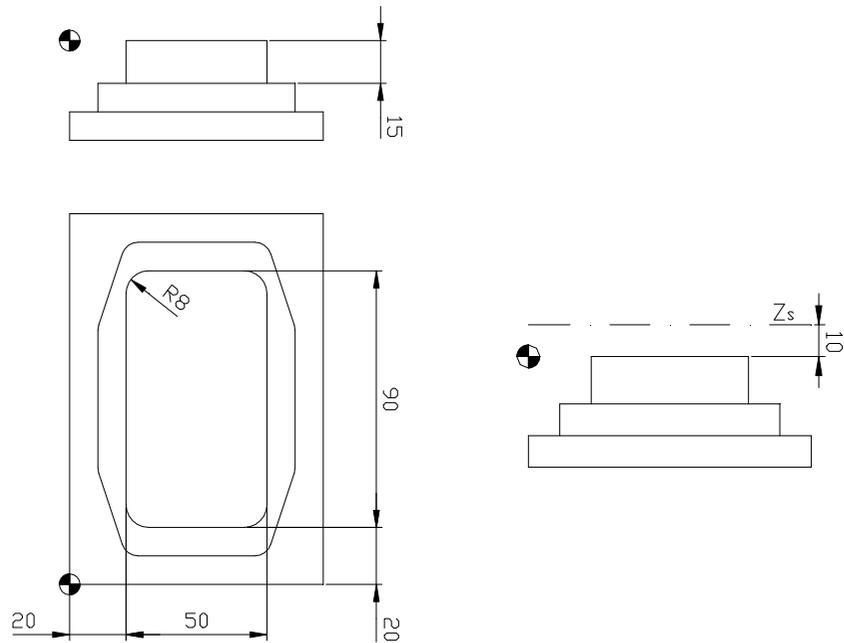
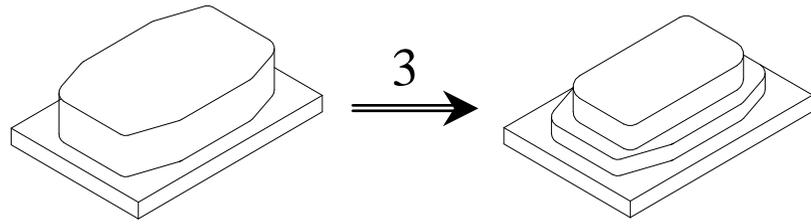
ROUGHING					
F	150.0000	S	1020	τ	2 D 2
FINISHING					
F	180.0000	β	1200	τ	2 D 2 δ 0.2000

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

# .MC. Option

## Step 3: Rectangular boss.



F6

LEVEL  
CYCLE

15:28:42

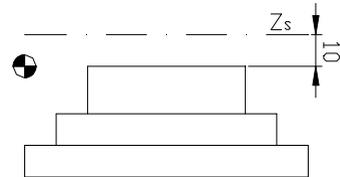
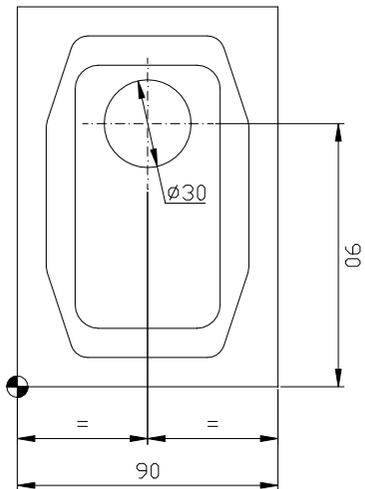
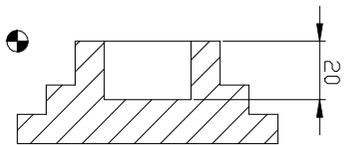
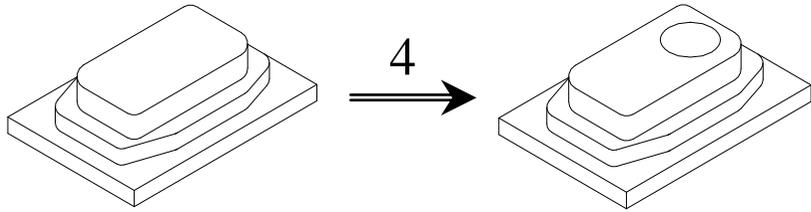
RECTANGULAR BOSS

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	00

X	20.0000	Y	20.0000
L	50.0000	H	90.0000
alpha	0.0000	Q	15.0000
r	8.0000		
Zs	10.0000	Z	0.0000
P	15.0000	I	5.0000
FZ	90.0000		

ROUGHING			
F	375.0000	S	2500
T	3	D	3
delta	2.0000		
FINISHING			
F	420.0000	S	2800
T	3	D	3
delta	0.2000	N	1
delta_z	0.2000		

## Step 4: Circular pocket.



F7

LEVEL  
CYCLE

15:28:42

CIRCULAR POCKET

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	00

Xc	45.0000
Yc	90.0000
R	15.0000

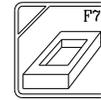
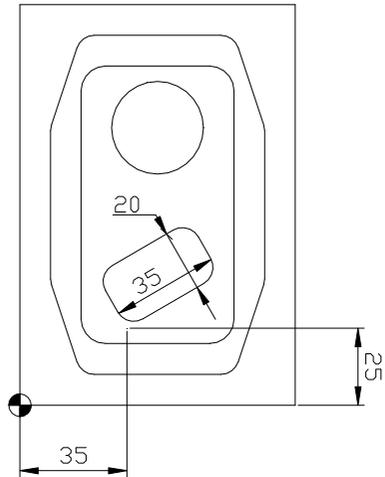
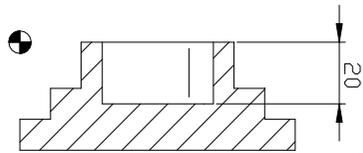
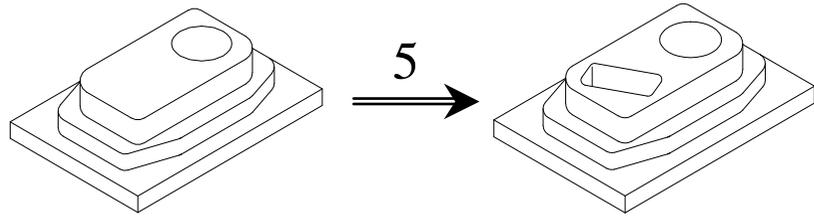
Zs	10.0000	Z	0.0000
P	20.0000	I	5.0000
FZ	100.0000		

ROUGHING				↻
F	375.0000	S	2500	T 3 D 3 Δ 2.0000
FINISHING				↻
F	420.0000	S	2800	δ 0.2000 N 1 δs 0.2000

# .MC. Option

## Step 5: Rectangular pocket.



15:28:42

RECTANGULAR POCKET

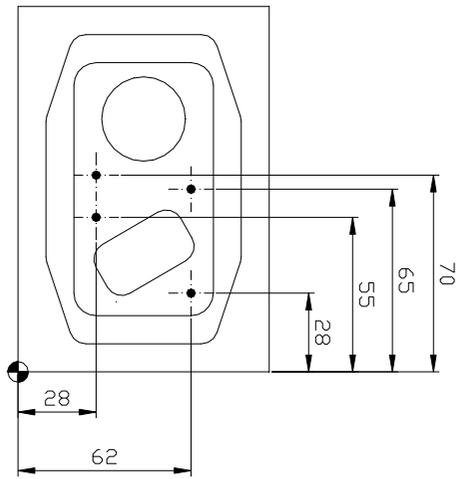
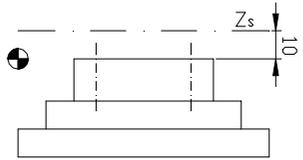
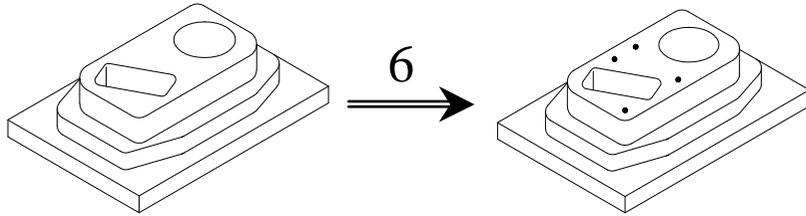
X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	00

X	35.0000	Y	25.0000
L	35.0000	H	20.0000
$\alpha$	30.0000		

r	6.0000		
Zs	10.0000	Z	0.0000
P	20.0000	I	5.0000
FZ	100.0000		

<b>ROUGHING</b>	F 375.0000	S 2500	↻	T 3 D 3 $\Delta$ 2.0000
<b>FINISHING</b>	F 420.0000	S 2800	↻	$\delta$ 0.2000 N 1
				$\delta_z$ 0.2000

## Step 6: Center punching + multi-point positioning.



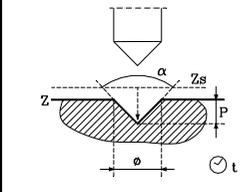


LEVEL  
CYCLE

15:28:42

X	12345.000	F	1234.600
Y	12345.000	S	1234
Z	12345.000	T	00

### CENTER PUNCHING



X	62.0000	Y	28.0000
Zs	10.0000	Z	0.0000
t	1.0000		

P	6.0000
---	--------

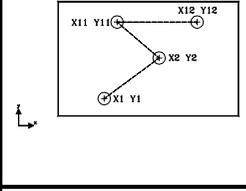
Penetration

F	125.0000	S	2100	T	4	D	4
---	----------	---	------	---	---	---	---

15:28:42

X	12345.000	F	1234.600
Y	12345.000	S	1234
Z	12345.000	T	00

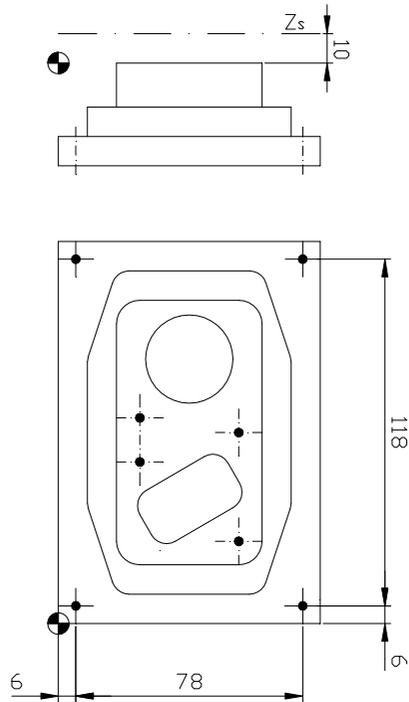
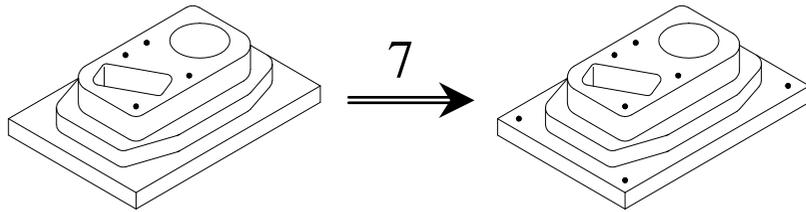
### RANDOM POSIT.



X1	62.0000	Y1	28.0000
X2	28.0000	Y2	55.0000
X3	28.0000	Y3	70.0000
X4	62.0000	Y4	65.0000
X5	62.0000	Y5	65.0000
X6	0.0000	Y6	0.0000

Zs	10.0000	Z	0.0000	P	6.0000	t	1.0000
F	125.0000	S	2100	T	4	D	4

## Step 7: Center point + Multiple positioning in a rectangular pattern.



LEVEL  
CYCLE

16:28:42

CENTER PUNCHING

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	00

X	6.0000	Y	6.0000
Z#	10.0000	Z	-25.0000
t	1.0000		

P	6.0000
---	--------

Penetration			
F	125.0000	S	2100
T	4	D	4

16:28:42

RECTANGULAR POSIT.

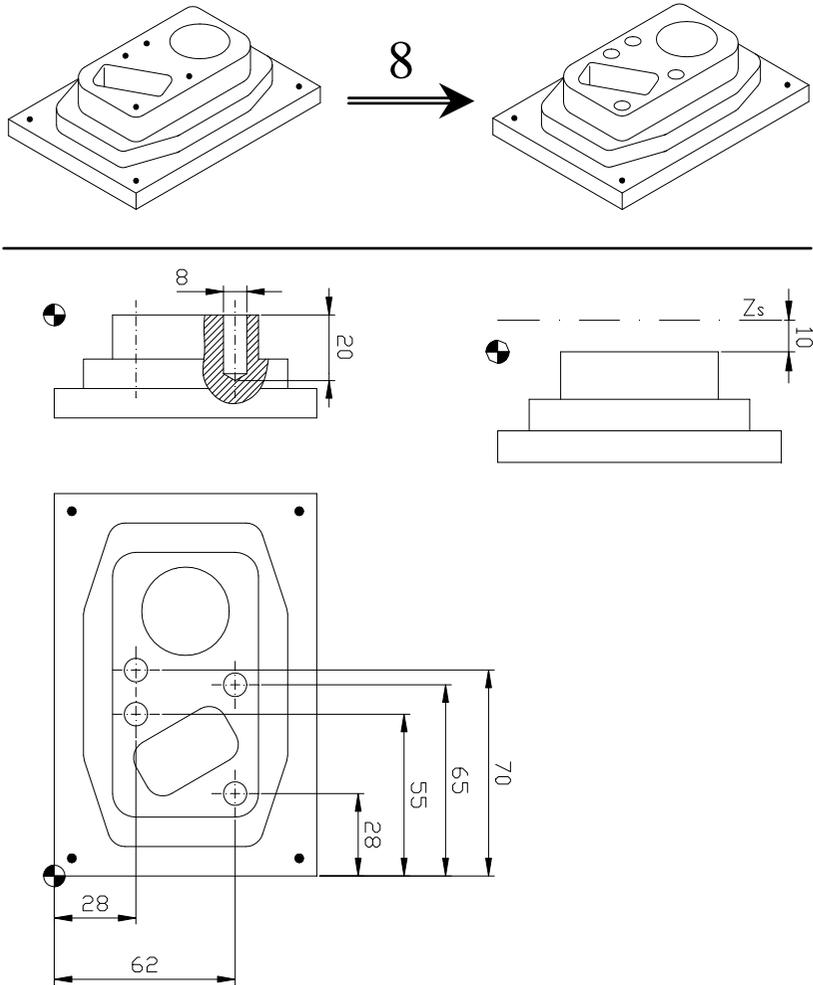
X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	00

X1	6.0000	Y1	6.0000
Lx	78.0000	Ly	118.0000
Nx	2	Ny	2
$\alpha$	0.0000	$\beta$	90.0000

Center punching			
Z#	10.0000	Z	-25.0000
P	6.0000	t	1.0000
F	125.0000	S	2100
T	4	D	4

## Step 8: Drilling + multi-point positioning.



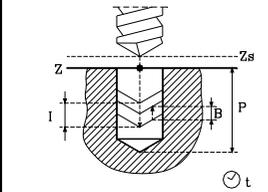


LEVEL  
CYCLE

15:28:42

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	07

### DRILLING 2



X	62.0000	Y	28.0000
Zs	10.0000	Z	0.0000
P	20.0000	I	8.0000
t	1.0000	B	3.0000

Penetration

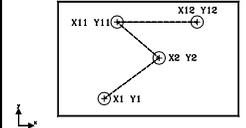
F	120.0000	S	1190	T	5	D	5
---	----------	---	------	---	---	---	---



15:28:42

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	07

### RANDOM POSIT.

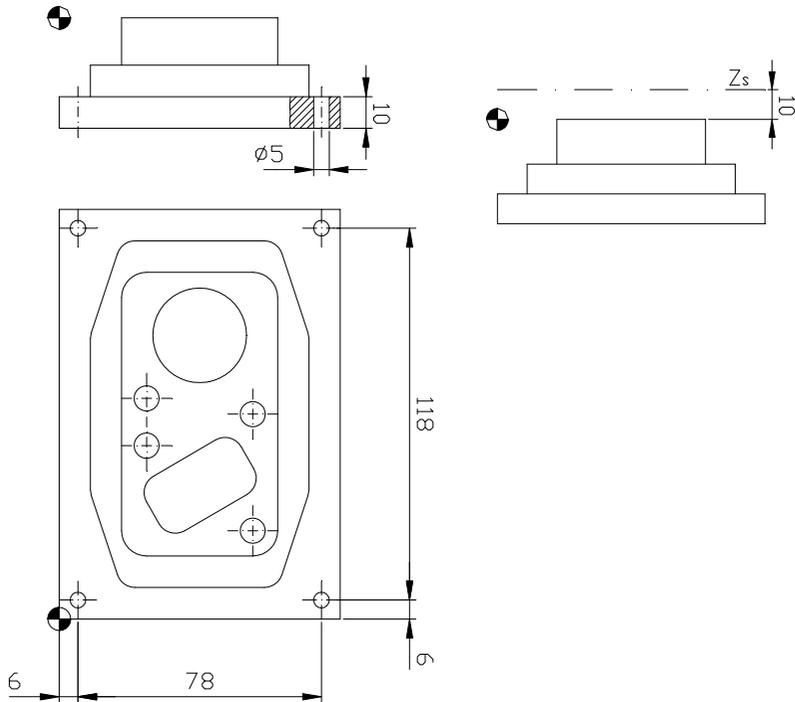
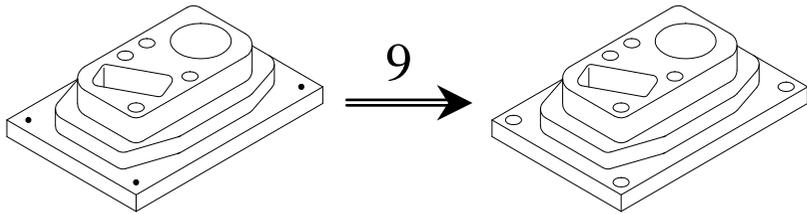


X1	62.0000	Y1	28.0000
X2	28.0000	Y2	55.0000
X3	28.0000	Y3	70.0000
X4	62.0000	Y4	65.0000
X5	62.0000	Y5	65.0000
X6	0.0000	Y6	0.0000

Drilling 2

Zs	10.0000	Z	0.0000	P	20.0000	t	1.0000	B	3.0000
F	120.0000	S	1190	T	5	D	5		

## Step 9: Drilling + Multiple positioning in a rectangular pattern.



LEVEL  
CYCLE

16:28:42

DRILLING 2

X	12345.000	P	1234.000
Y	12345.000	S	1234
Z	12345.000	T	07

X	6.0000	Y	6.0000
Zs	10.0000	Z	-25.0000
P	15.0000	I	8.0000
t	0.0000	B	1.0000

Penetration			
F	190.0000	S	1900
T	6	D	6

16:28:42

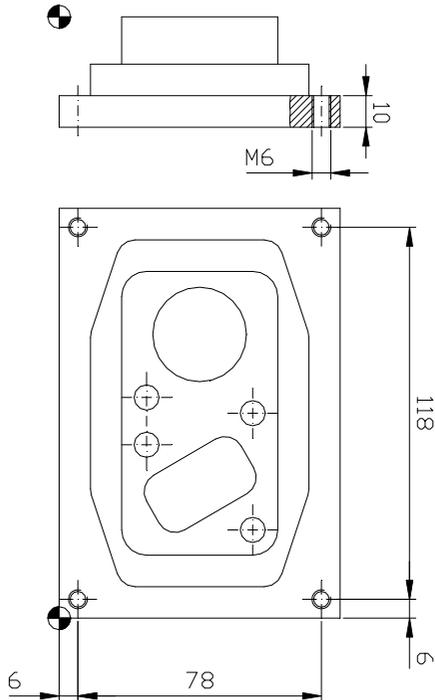
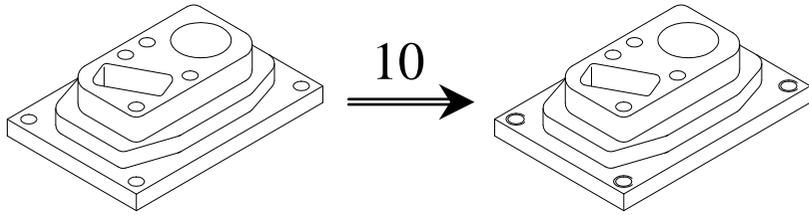
RECTANGULAR POSIT.

X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	07

X1	6.0000	Y1	6.0000
Lx	78.0000	Ly	118.0000
Nx	2	Ny	2
α	0.0000	β	90.0000

Drilling 2			
Zs	10.0000	Z	-25.0000
P	15.0000	t	0.0000
B	1.0000	I	8.0000
F	190.0000	S	1900
T	6	D	6

## Step 10: Tapping + Multiple positioning in a rectangular pattern.

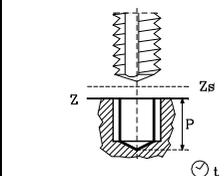




LEVEL  
CYCLE

15:28:42

### TAPPING



X	00044.000	F	1.000
Y	00022.000	S	150
Z	-00397.400	T	3

X	6.0000	Y	6.0000
Zs	10.0000	Z	-25.0000
P	15.0000	t	1.0000

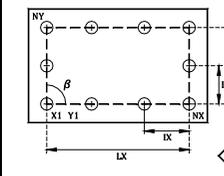
Penetration			
F	250.0000	S	250
T	7	D	7



15:28:42

### RECTANGULAR POSIT.



X	12345.000	F	1234.000
Y	12345.000	S	1234
Z	12345.000	T	00

X1	6.0000	Y1	6.0000
Lx	78.0000	Ly	118.0000
Nx	2	Ny	2
alpha	0.0000	beta	90.0000

Tapping			
Zs	10.0000	Z	-25.0000
P	15.0000	t	1.0000
F	250.0000	S	250
T	7	D	7

