CNC 8055 TC

Self-teaching manual

REF. 1010

FAGOR

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; *utelnet* 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

CHAPTER 1 THEORY ON CNC MACHINES.

1.1 Machine axes. 7
1.2 Machine reference zero and part zero. 8
1.3 Home search. 9
1.4 Travel limits. 10
1.5 Setting the part zero. 11
1.6 Work units. 12
1.7 Spindle speed. 13

CHAPTER 2 THEORY ON TOOLS.

2.1 The tool change. 17
2.2 The tool table. 18
2.3 Tool calibration. 25

CHAPTER 3 MANUAL PRACTICE.

3.1 Description of the screen and the keyboard. 31
3.1.1 Keyboard description. 32
3.1.2 Description of the standard screen. 34
3.1.3 Description of the auxiliary screen. 35
3.2 Home search. 36
3.3 Spindle. 37
3.3.1 Speed ranges (gears). 39
3.4 Jogging the axes. 40
3.4.1 Handwheels. 41
3.4.2 Jogging. 42
3.4.3 Moving an axis to a coordinate. 44
3.5 Tool change. 45
3.5.1 Tool change position. 46
3.5.2 Tool calibration. 47
3.5.3 How to change any data of the tool table. 50
3.5.4 Checking for proper calibration. 51

CHAPTER 4 OPERATIONS OR CYCLES.

4.1 Description of the screen and the keyboard. 55
4.2 Work modes. 57
4.3 Operations or cycles. 58
4.3.1 Editing an operation or cycle. 59
4.3.2 Simulating an operation or cycle. 63
4.3.3 Executing an operation or cycle. 67

CHAPTER 5 SUMMARY OF WORK CYCLES.

5.1 Positioning cycle. 73
5.2 Turning cycle. 74
5.3 Facing cycle. 75
5.4 Taper turning cycle. 76
5.5 Rounding cycle. 78
5.6 Threading cycle. 79
5.7 Grooving cycle. 82
5.8 Drilling and tapping cycles. 85
5.9 Profiling cycle. 86

FAGOR

·TC· Option

REF. 1010

CHAPTER 6 **CONVERSATIONAL PART PROGRAMS.**

6.1 What is a conversational part-program? ... 91
6.2 Editing a part-program. ... 92
6.3 Modifying a part-program. ... 95
6.4 Simulating/executing an operation. ... 99
6.5 Simulating/executing a part-program. ... 100
6.6 Simulating/executing a part-program starting with an operation. ... 101
6.7 Copying a part-program into another one. ... 102
6.8 Deleting a new part program. ... 103

APPENDIX **OTHER LATHE TYPE MACHINING OPERATIONS.**

A.1 Introduction. ... 107
A.2 Orienting spindle and live tool. ... 108
A.3 Multiple drilling. ... 109
A.4 Multiple tapping. ... 110
A.5 Slots. ... 111

FAGOR

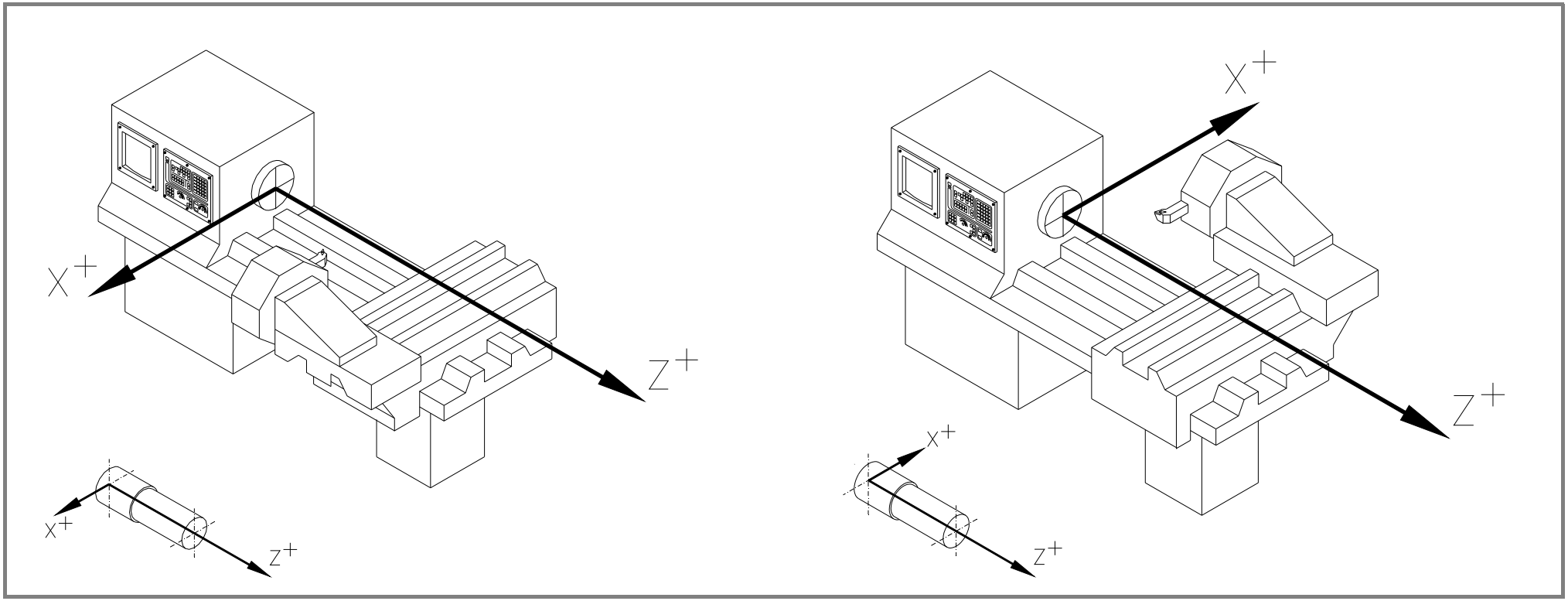
·TC· Option

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?.
- Ways to work with the spindle.

1.1 Machine axes.



Z axis: Longitudinal axis of the machine.

X axis: Cross axis of the machine.

1.2 Machine reference zero and part zero.

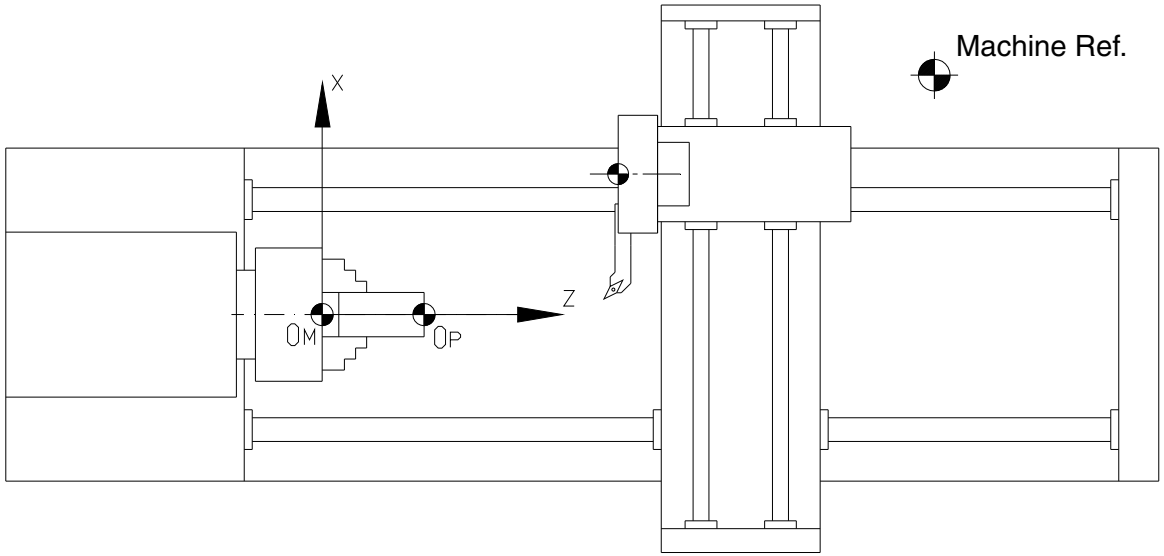
They are the references that the machine needs to work.

Machine reference zero (O_M)

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

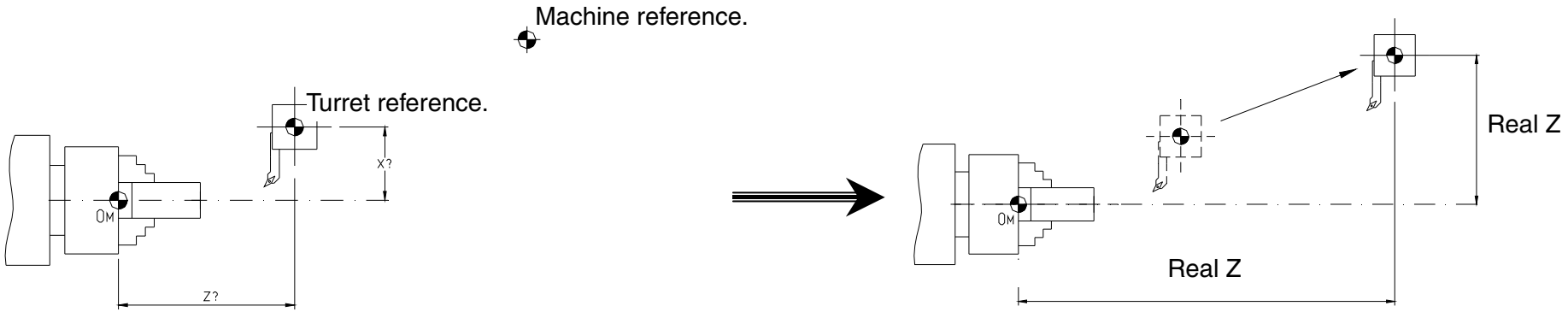
Part zero (O_P)

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.



Machine reference: It is the point to which the tool moves in a home search.

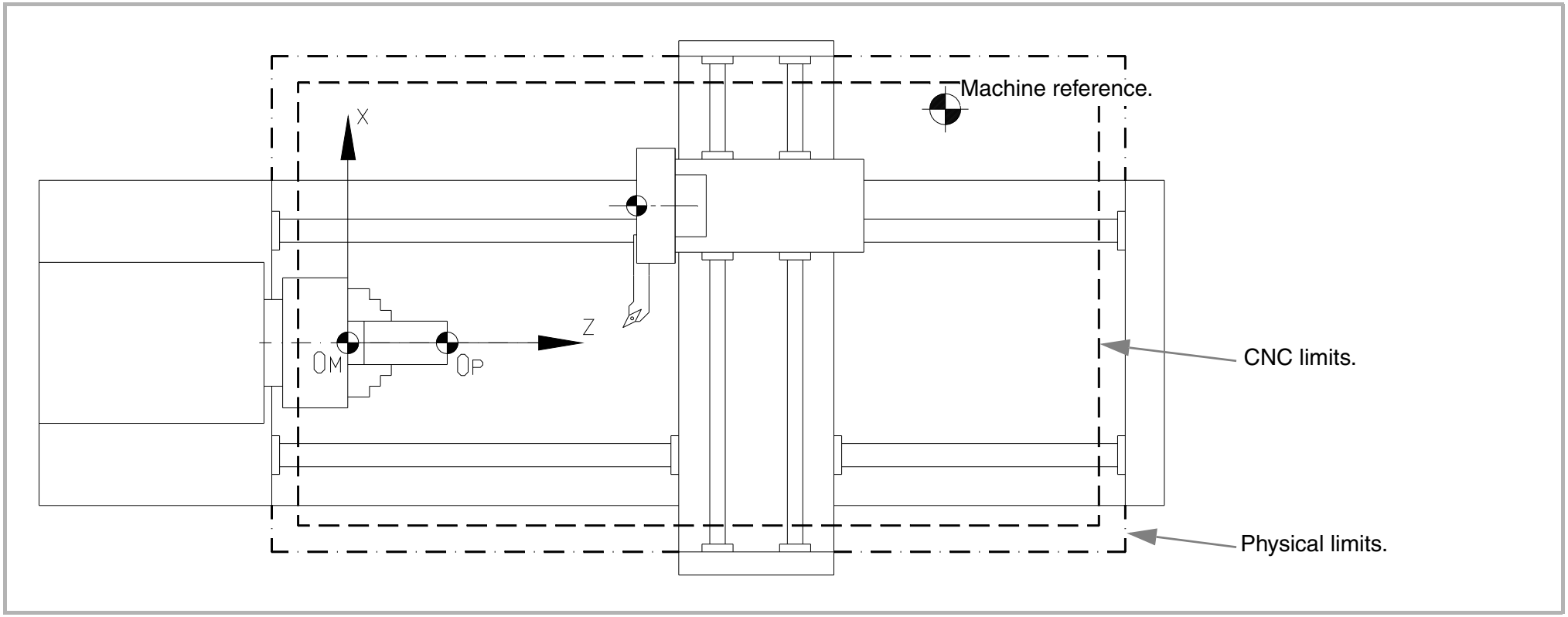
Turret reference: Point that moves with the turret. It is the point that moves in a home search.

NOTE: With the new feedback devices (distance-coded reference marks - Io -), 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.

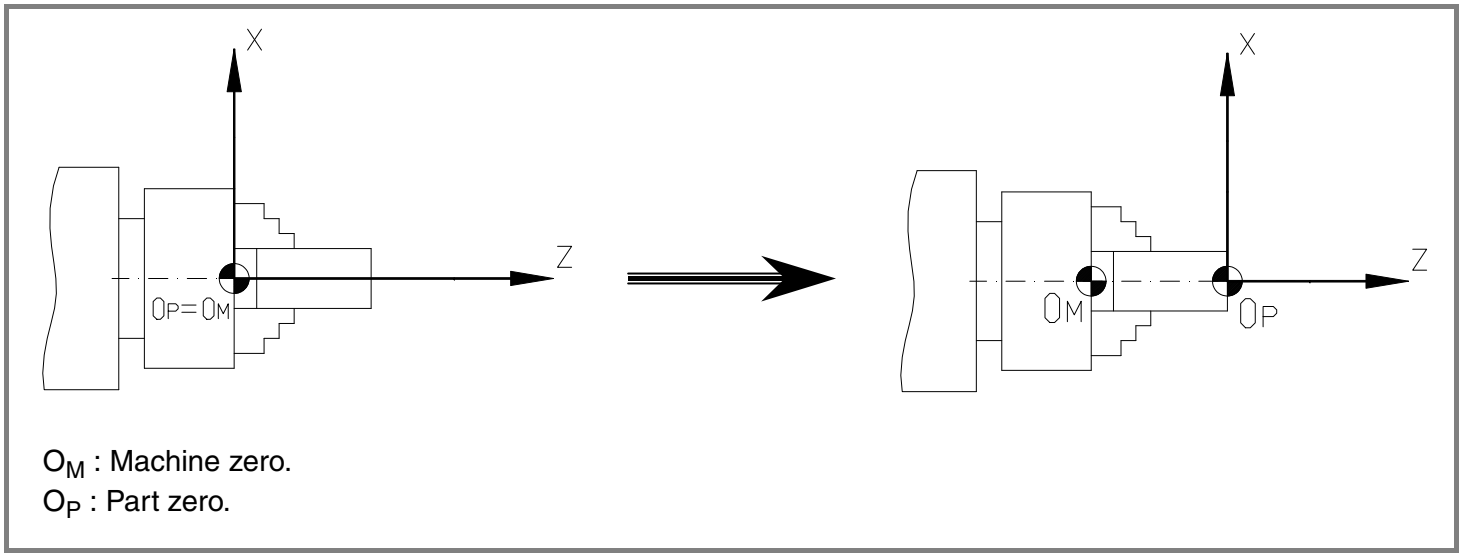


1.5 Setting the part zero.

The part zero is set on the Z axis.

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.

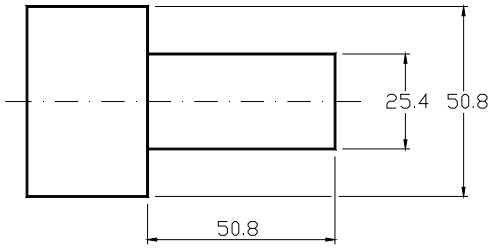


O_M : Machine zero.
O_P : Part zero.

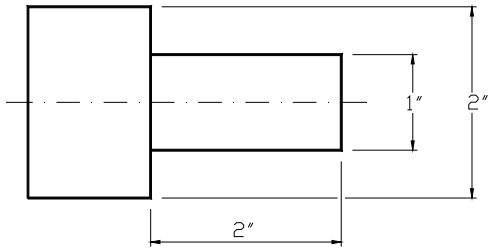
1.6 Work units.

Programming units.

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



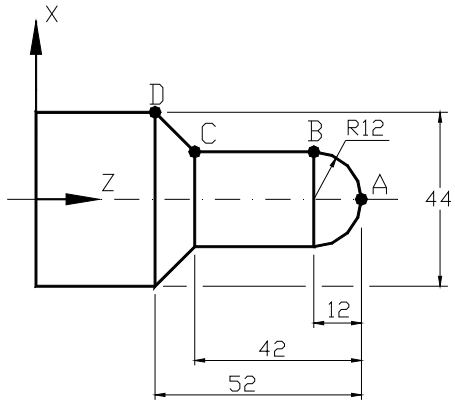
Millimeters.



Inches.

Programming units.

The units are set by the manufacturer and may be in radius or diameter.



	Radius.	Diameters.
A	X=0 Z=0	X=0 Z=0
B	X=12 Z=-12	X=24 Z=-12
C	X=12 Z=-42	X=24 Z=-42
D	X=22 Z=-52	X=44 Z=-52

Feedrate of the axes.

The units are set by the manufacturer and may be:

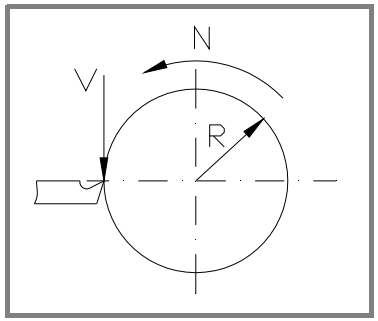
millimeters (inches) per minute:
The feedrate is independent from the spindle speed.

millimeters (inches) per turn:
The feedrate changes with the spindle speed.

It is recommended to work at constant surface speed (CSS) and feedrate in millimeters/turn. This will lengthen tool life and will improve the part finish.

1.7 Spindle speed.

There are two types of speeds; cutting (surface) speed and turning speed.



- Cutting (surface) speed (V).

  It is the linear speed of the contact points between the part and the tool.

- Turning speed (N).

  It is the angular speed of the part.

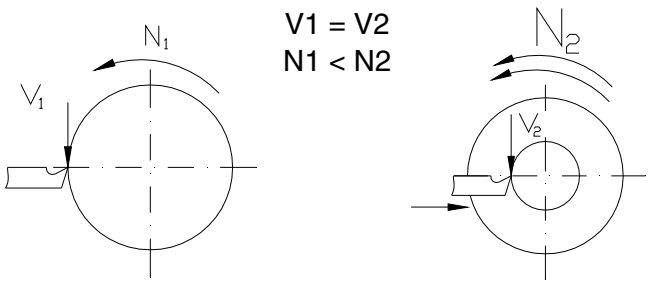
The relationship between the two is:

$$V = \frac{2 \times \Pi \times R \times N}{1000}$$

Regarding the way the spindle works, the CNC offers two choices.

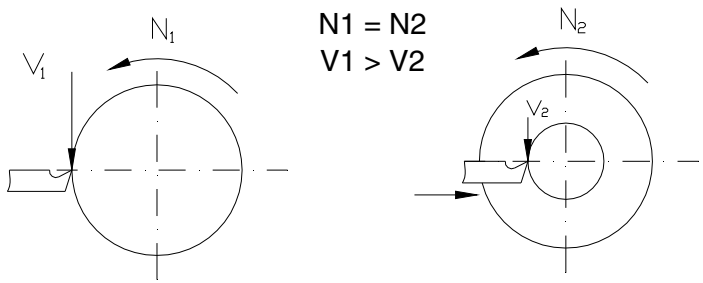
CSS: Constant surface speed.

The CNC maintains the surface (cutting) speed (V) constant. The turning speed (N) changes.



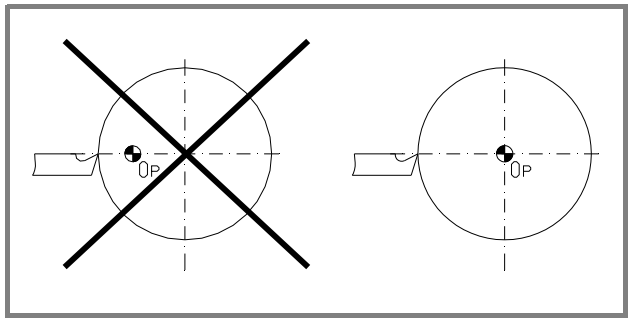
RPM: Constant turning speed.

The CNC maintains the turning speed (N) constant. The cutting (surface) speed (N) changes.

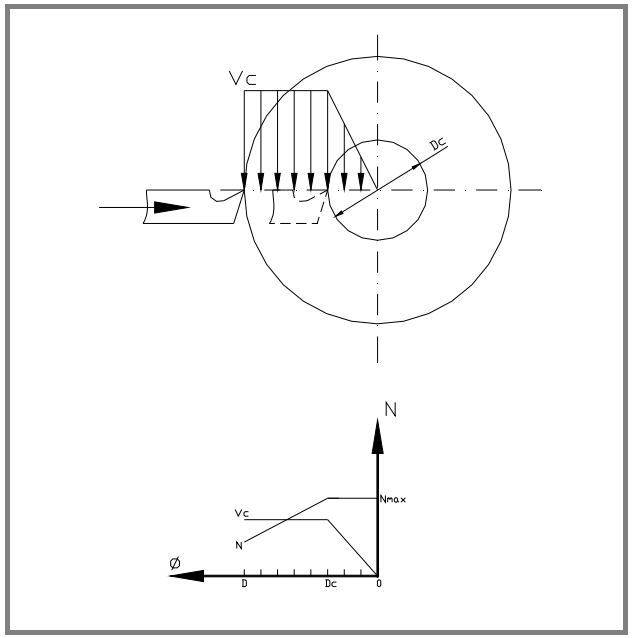


ABOUT CONSTANT SURFACE SPEED

To work at constant surface speed (CSS), consider these two aspects.



Part zero must be in the part rotation axis so that the calculated turning speed is the same as the best cutting speed.



A maximum turning speed must be calculated because the turning speed increases as the diameter decreases and a certain speed should not be exceeded on part with a large diameter.

The CNC works at constant surface speed (Vc) and from diameter Dc on, (when N=Nmax), it works at constant turning speed.

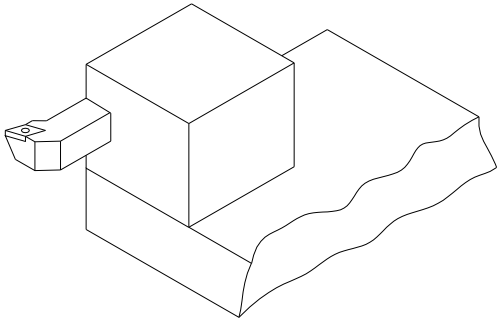
2 THEORY ON TOOLS.

- 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.

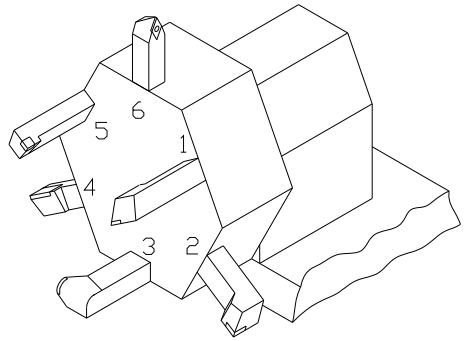
2.1 The tool change.

The tools that the CNC can use are located in the tool holding turret. This turret may have either a manual tool changer or an automatic tool changer.

- When the turret has manual tool change, it is done like on a conventional machine.
- When the has an automatic tool changer, all the tools are located on the turret and when changing a tool the CNC will rotate the turret until the new tool is placed in the work condition.



Manual change turret.



Automatic change turret.

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 data.
- The tool type must also be defined.

Diamond shaped tool.

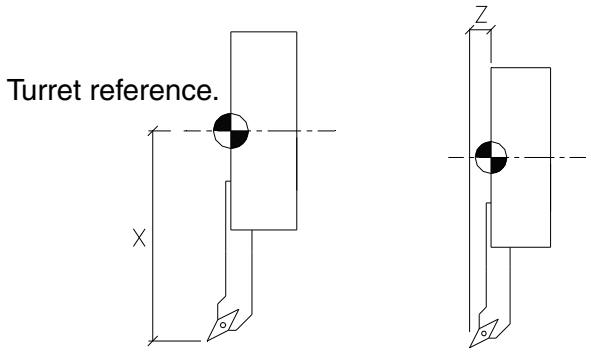
Threading tool.

Square tool.

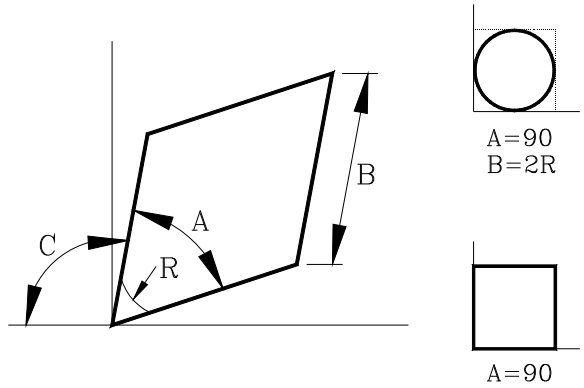
Round tool.

Live tool.

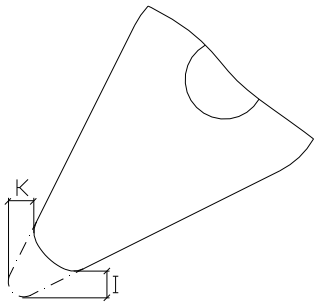
- ·X· Tool length along the X axis (in radius).
- ·Z· Tool length along the Z axis.



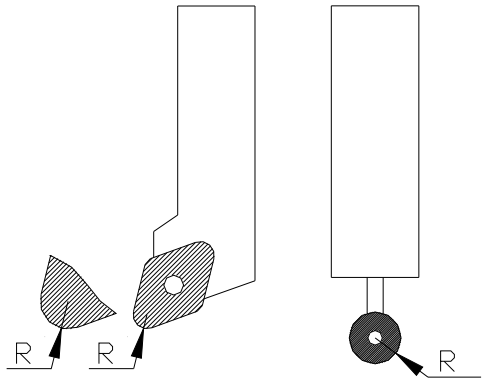
- ·A· Cutter angle.
- ·B· Cutter width.
- ·C· Cutting angle.



- ·I· Y axis wear offset.
- ·K· Z axis wear offset.

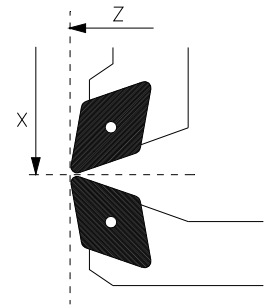


- ·R· Tool radius.

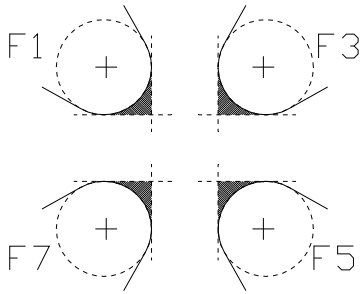


- ·F· Location code or the way the tool has been calibrated.

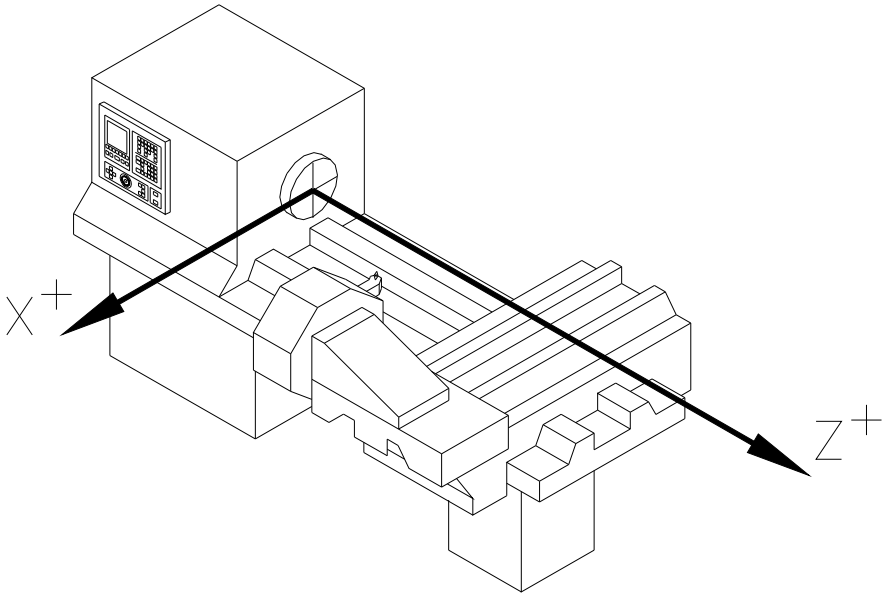
  Once the tool dimensions are known;

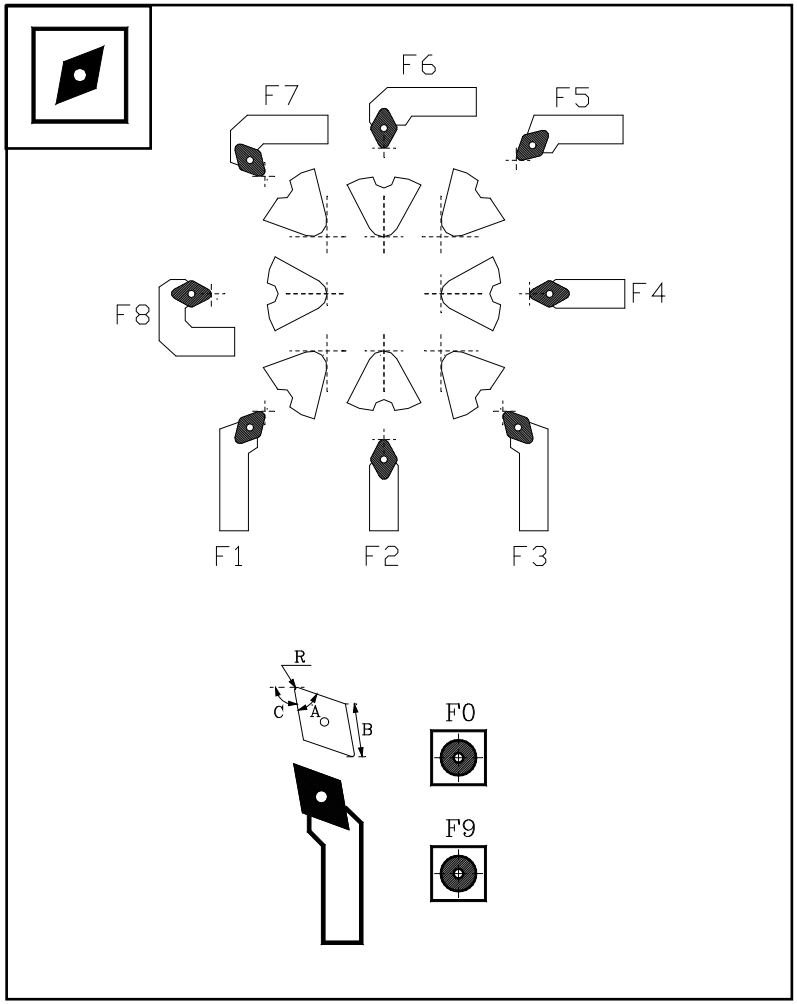


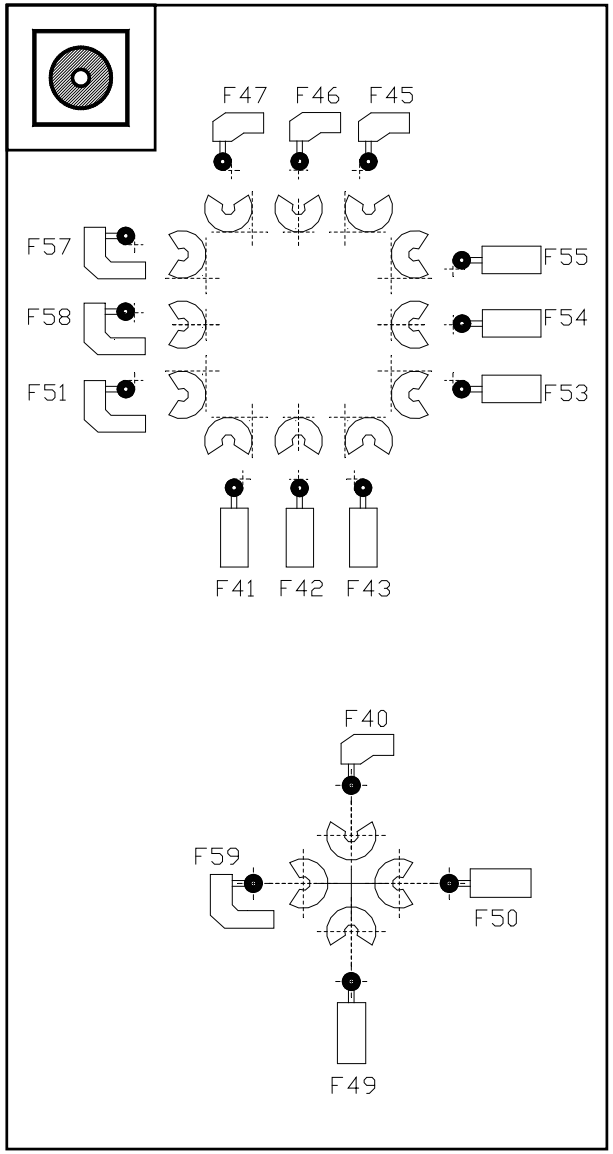
The CNC must know the location code or calibration point of that tool in order to apply tool radius compensation. The location code depends on the orientation of the machine axes.

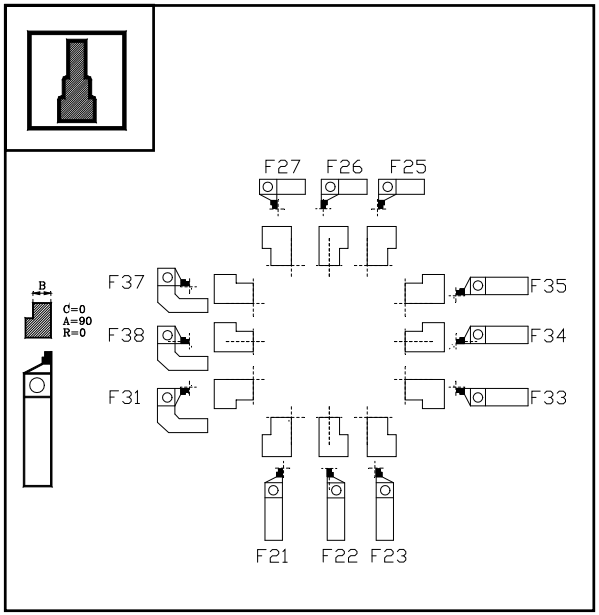


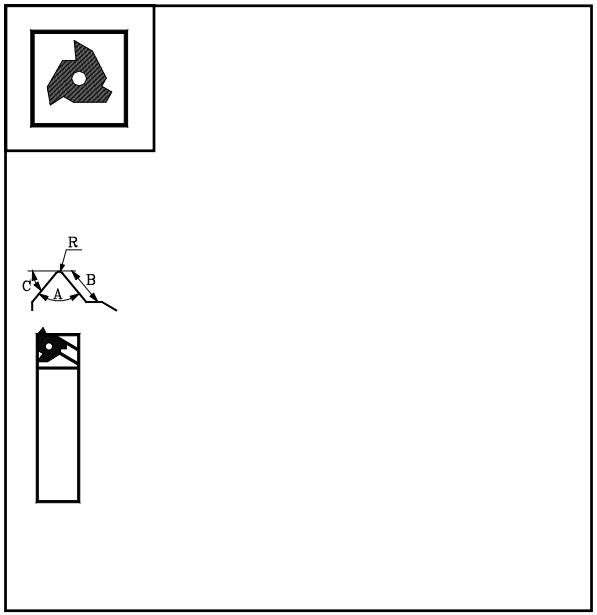
MOST USUAL TOOL LOCATION CODES (TOOL TYPES).

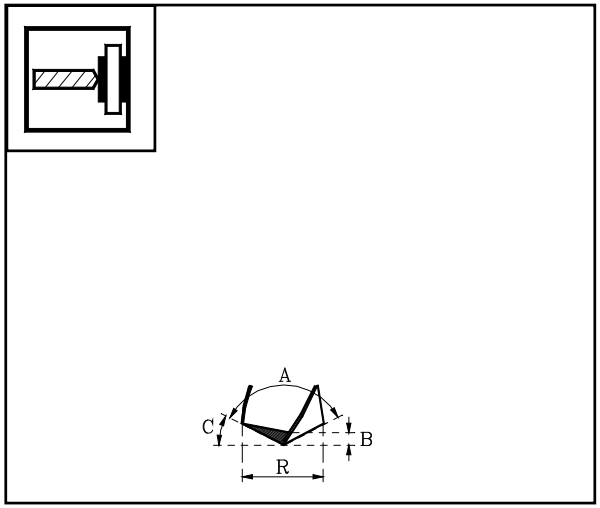




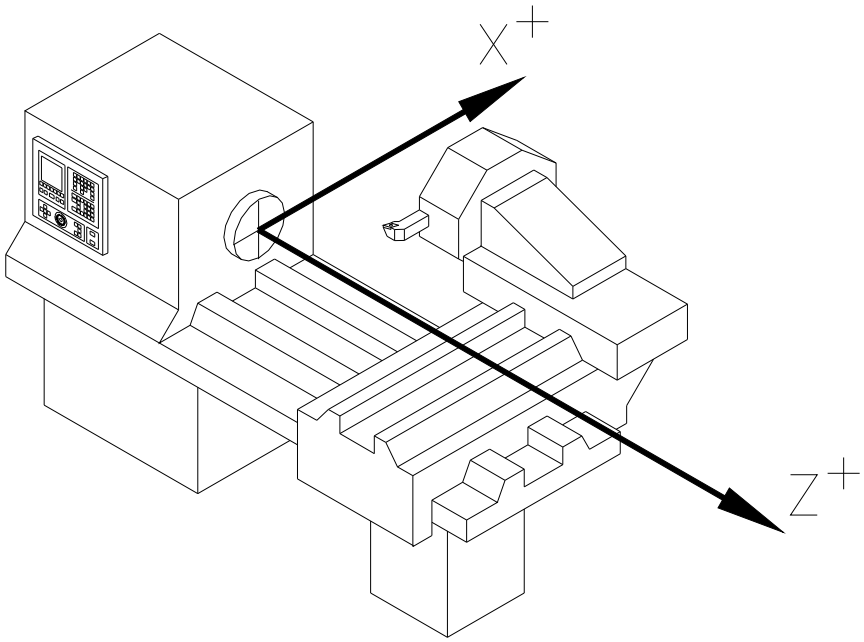

F47
F46
F45
F57
F55
F58
F54
F51
F53
F41
F42
F43
F40
F59
F50
F49

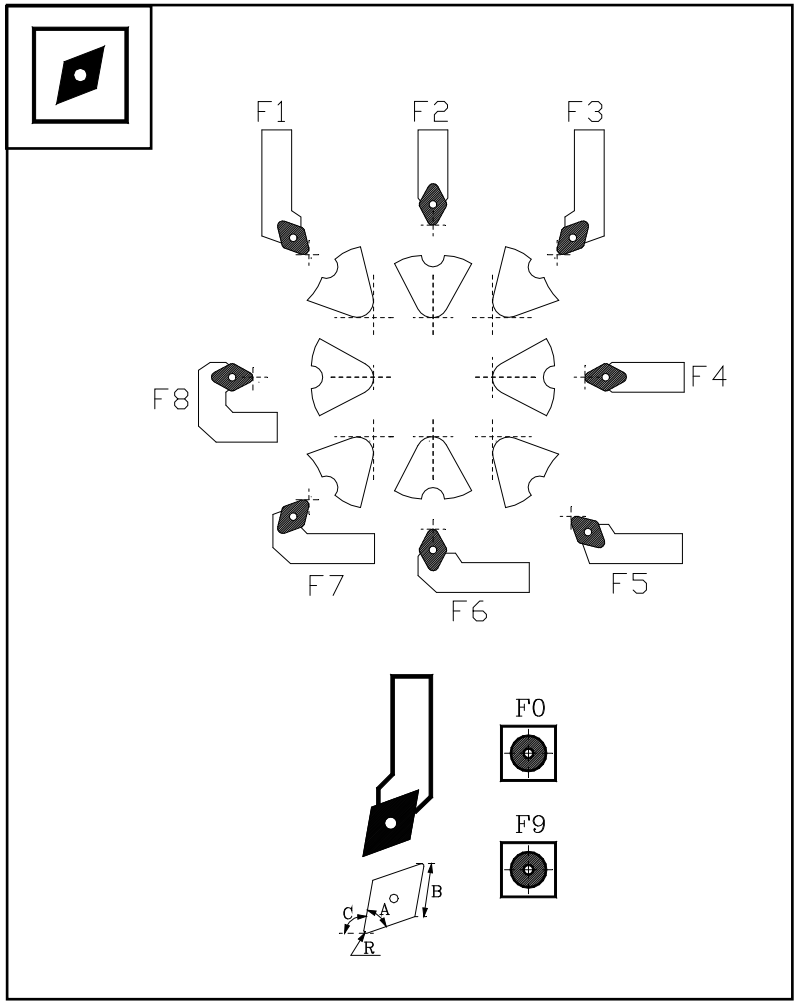

F27
F26
F25
B
C=0
A=90
R=0
F37
F35
F38
F34
F31
F33
F21
F22
F23

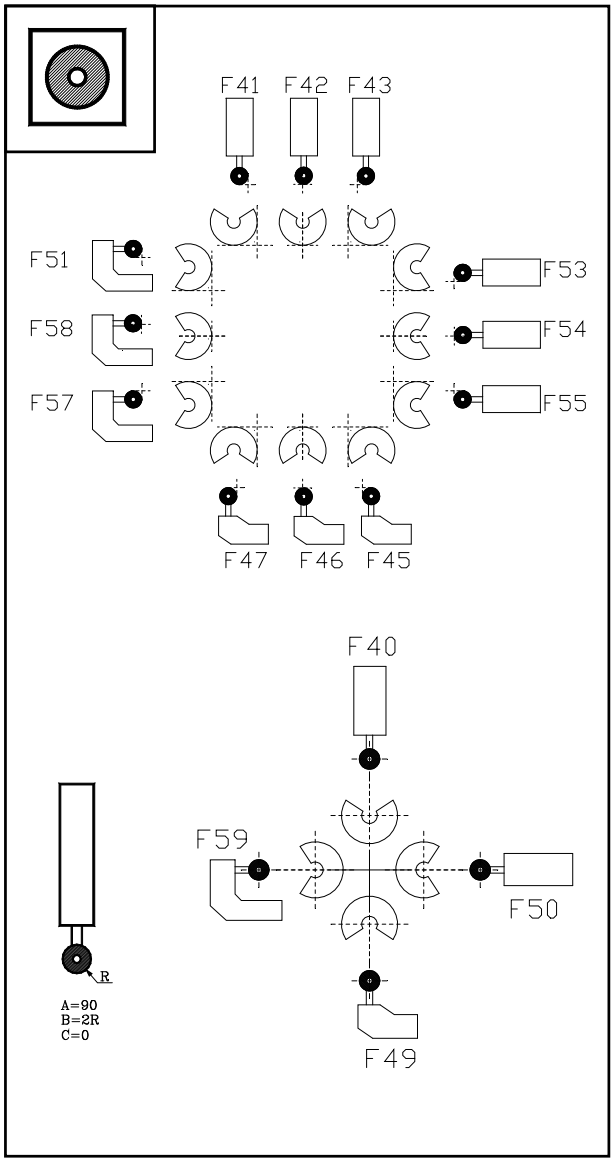

R
C
A
B

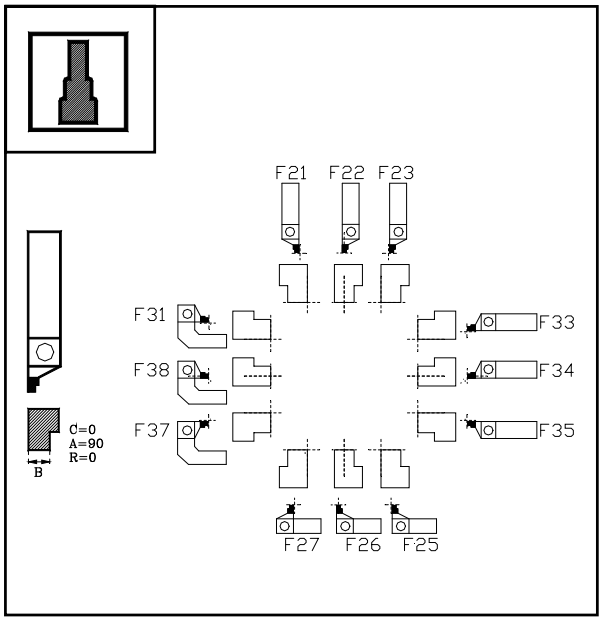

A
C
B
R

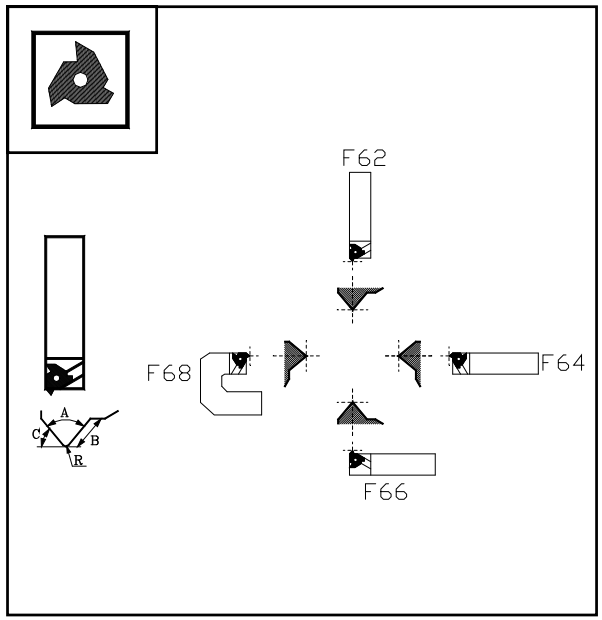
MOST USUAL TOOL LOCATION CODES (TOOL TYPES).

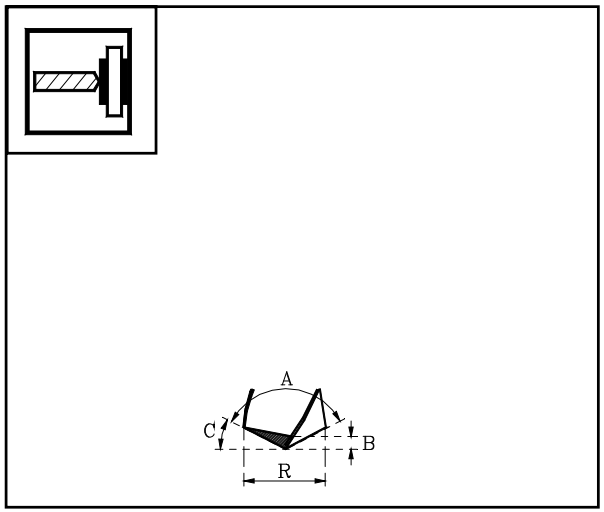





F41
F42
F43
F51
F53
F58
F54
F57
F55
F47
F46
F45
F40
F59
F50
R
A=90
B=2R
C=0
F49


F21
F22
F23
F31
F33
F38
F34
F37
F35
C=0
A=90
R=0
B
F27
F26
F25


F62
F68
F64
A
C
B
R
F66

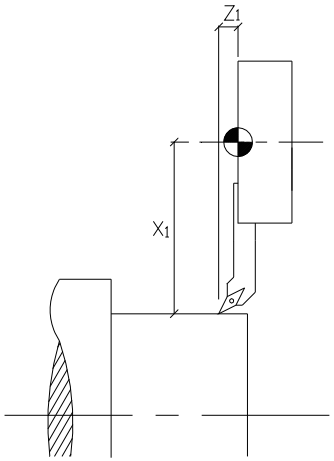

A
C
B
R

2.3 Tool calibration.

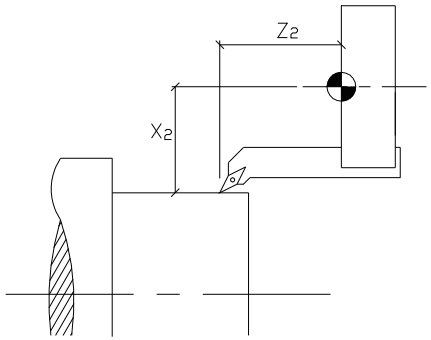
Tool calibration is the operation the CNC uses to define the dimensions of the tool. It is very important to carry out this operation properly so the parts are made with the right dimensions and so when changing the tool the CNC keeps controlling the same point.

X	00044.000
Z	-00043.331

T1



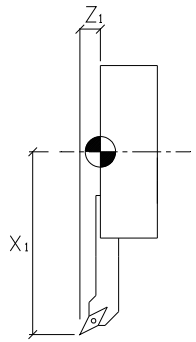
T2

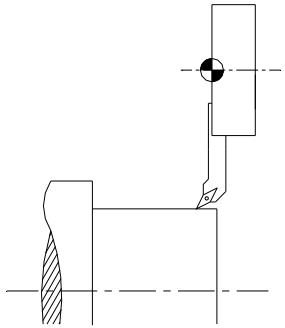


X	00044.000
Z	-00043.331

Different tool dimensions. Same point.

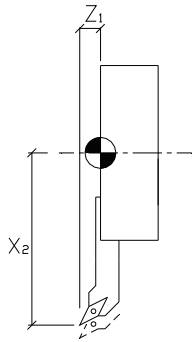
DEFECTS DUE TO WRONG TOOL LENGTH CALIBRATION.

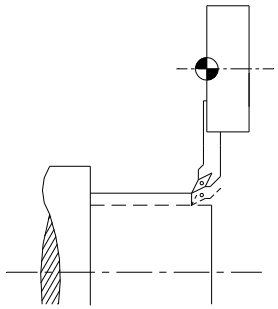




·X1· Actual (real) length.
·Z1· Actual (real) length.

Part with the right dimensions.

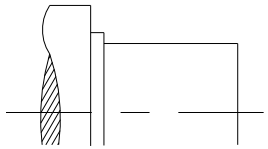




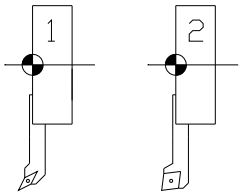
·X2· Wrong length.

·X2· < Actual (real) length.

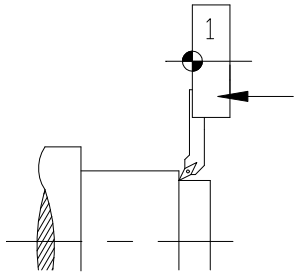
Part with the wrong dimensions.

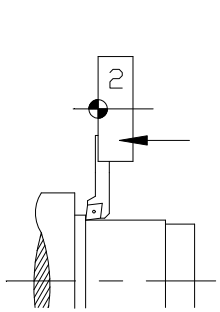


Part to be machined.

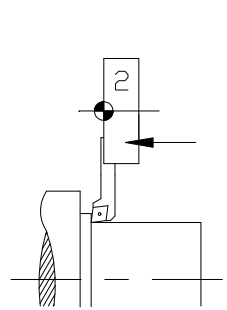


Tools.





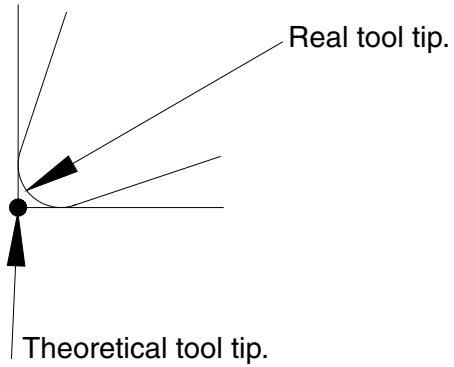
Wrong machining.
Tools calibrated wrong.



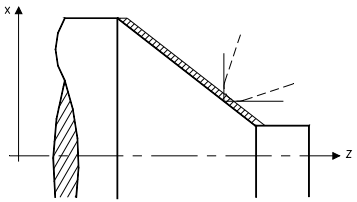
Correct machining.
Tools calibrated properly.

DEFECTS DUE TO WRONG LOCATION CODE DEFINITION.

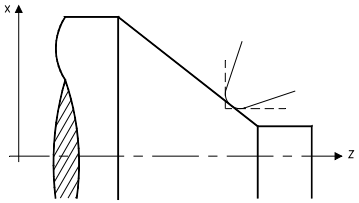
The tool has a theoretical tip and a real one.



When working with the theoretical tip, machining ridges remain. To prevent this, the CNC works with the real tip.

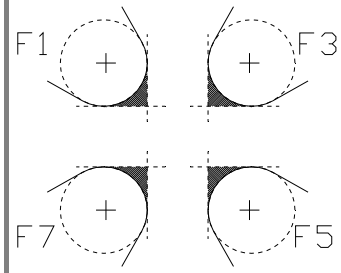


Working with the theoretical tool tip.



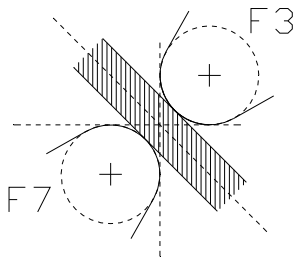
Working with the real tool tip.

To work with the theoretical tool tip, the CNC must know the calibration point of that tool (location code) in order to be able to apply tool radius compensation.

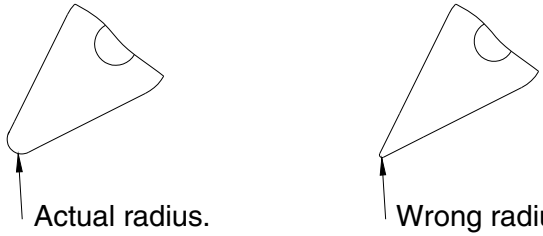


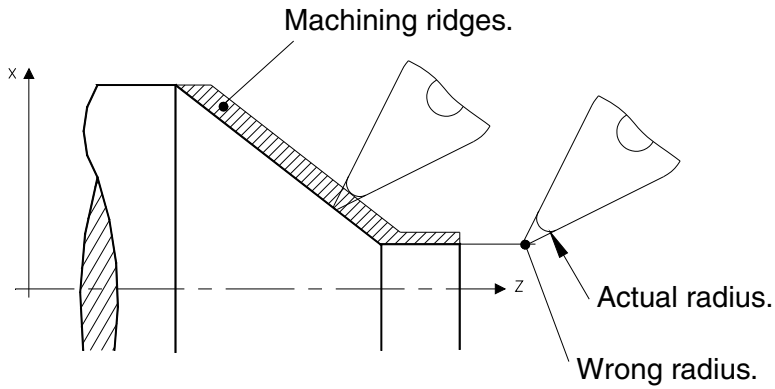
Knowing the location code, the CNC can compensate for the shaded part (radius compensation).

If the wrong location code is entered, the error increases instead of being compensated.



DEFECTS DUE TO WRONG TOOL RADIUS CALIBRATION.





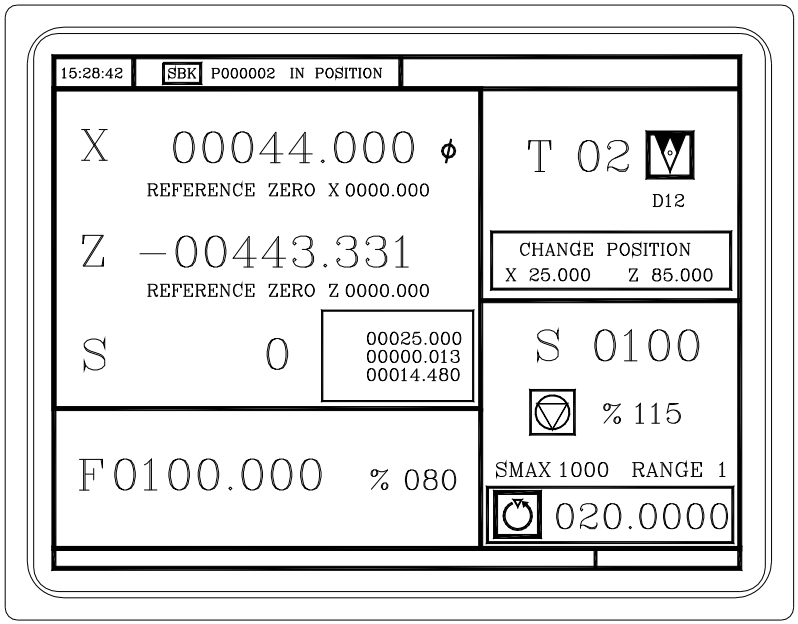
The residual stock is due to the difference between the two radii.

3 MANUAL PRACTICE.

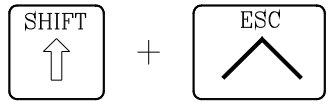
- Description of the screen and the keyboard.
- How to do a home search.
- Working with the spindle.
- Ways to jog the axes (handwheels, continuous jog, incremental jog).
- Tools.

3.1 Description of the screen and the keyboard.

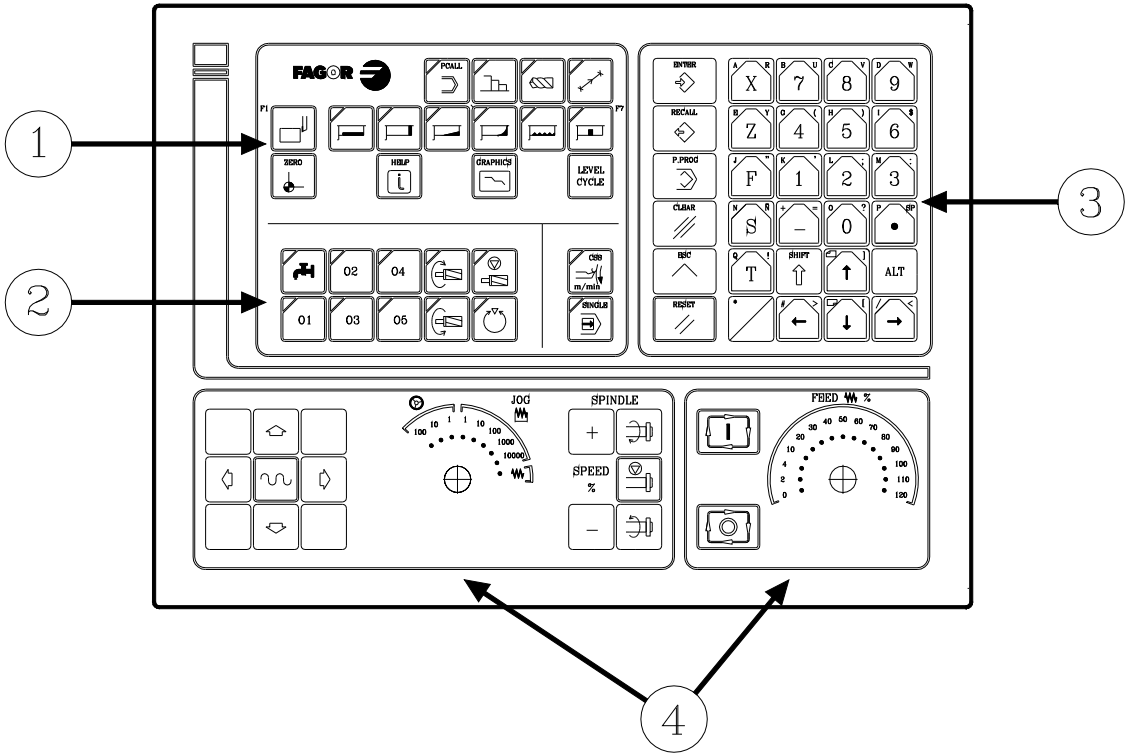
On power-up, the CNC shows the following screen.



If the CNC does not show this screen, the CNC may be in ·T· 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, live tool and spindle work mode (RPM/CSS).

3· Alphanumeric keys and command keys.

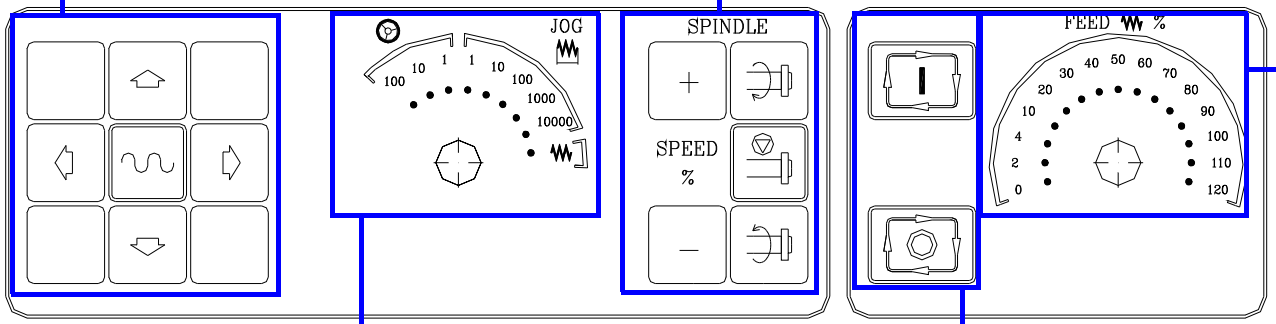
4· Operator panel.

Operator panel description.

Jog keyboard for manual movement of axes.

Selecting the turning direction of the spindle and starting it.

Percentage variation of the spindle speed.



Percentage variation of the feedrate.

Work mode selector.

Continuous jog.

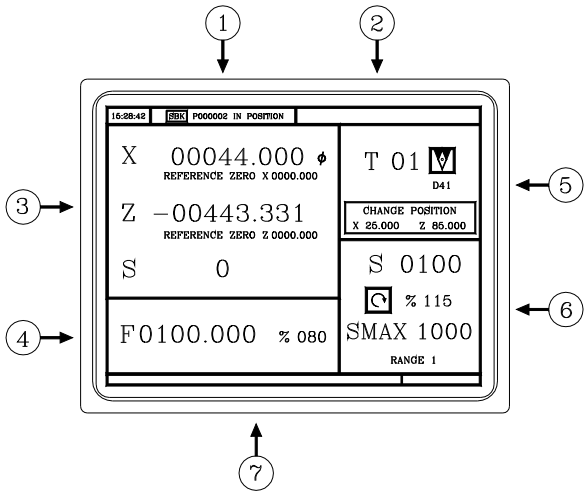
Incremental jog.

Handwheels.

[START] key.

[STOP] key.

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.

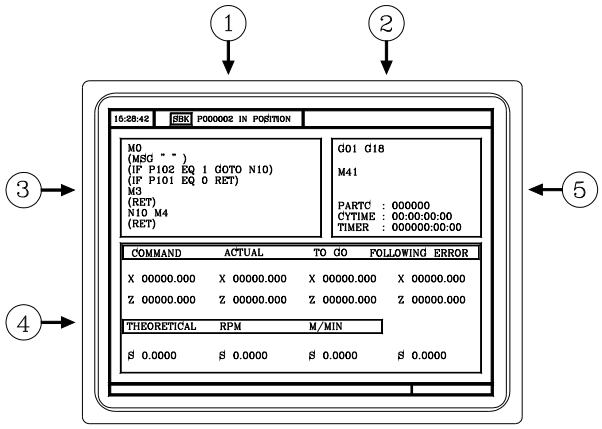
5· Tool information. Active tool and tool change position.

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 (REST) and the difference between the theoretical position and the real position (FOLLOWING ERROR).

Spindle information. Programmed theoretical speed, speed in rpm and feedrate in meters per minute.

5· Active G and M functions.

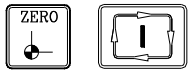
Number of consecutive parts executed with the program (PARTC), part execution time (CYTIME) and PLC timer.

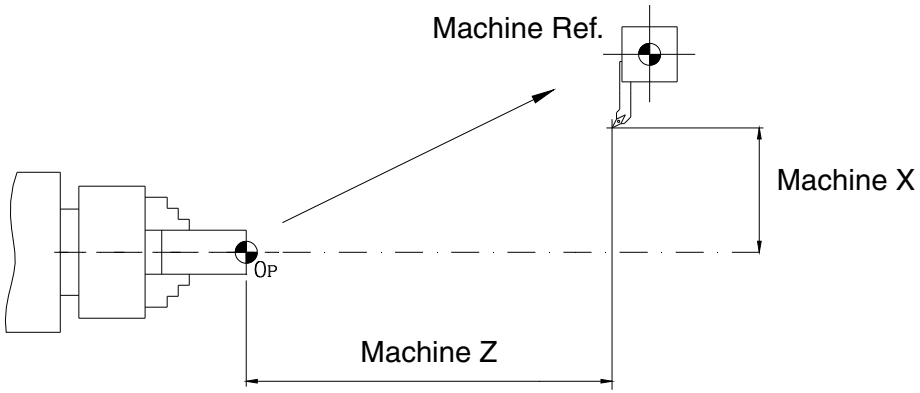
3.2 Home search.

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.

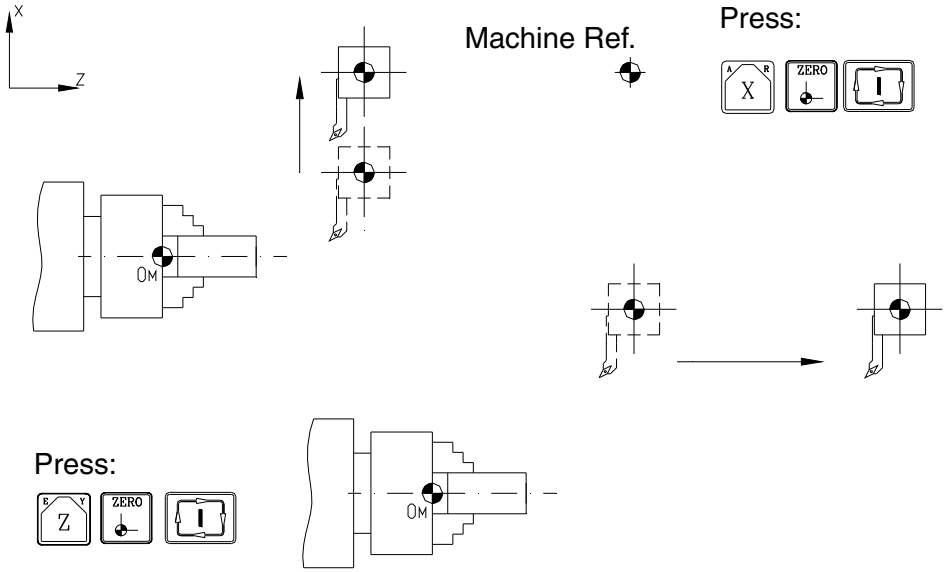
Press: 



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

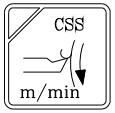
Manual home search.

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



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

3.3 Spindle.



The spindle of a CNC machine can work in two modes. The constant surface speed mode is selected with the [CSS] key.

- Working in RPM mode (revolutions per minute).
- Working in CSS (constant surface speed) mode.

In CSS mode, the turning speed changes depending on the position of the X axis.

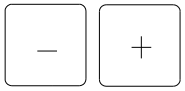
- If X decreases, the turning speed increases.
- If X increases, the turning speed decreases.

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

Spindle clockwise. Clockwise.

Stops the spindle.

Spindle counterclockwise. Counterclockwise.



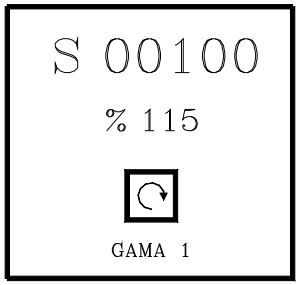
Increase or decrease the percentage of spindle speed.

Working in RPM mode (revolutions per minute).

To select the turning speed, press:

+ Speed +

The CNC displays the following information.



Selected speed.

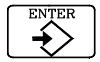
Percentage applied.

Turning direction.

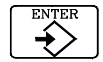
Active spindle range (gear).

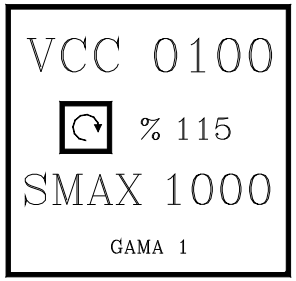
Working in CSS (constant surface speed) mode.

To select the cutting (surface) speed, press:

+ Cutting (surface) speed + 

To select the maximum turning speed, press:

+ + Cutting (surface) speed + 



Selected speed.

Turning direction and percentage applied.

Maximum turning speed.

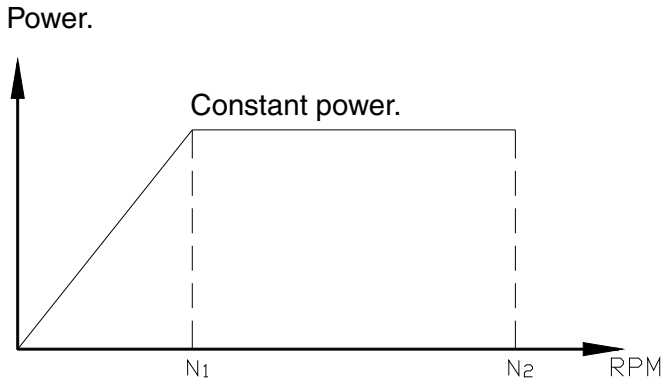
Active spindle range (gear).

A work gear (range) must be selected before programming the cutting speed. A gear change cannot be done during a machining operation.

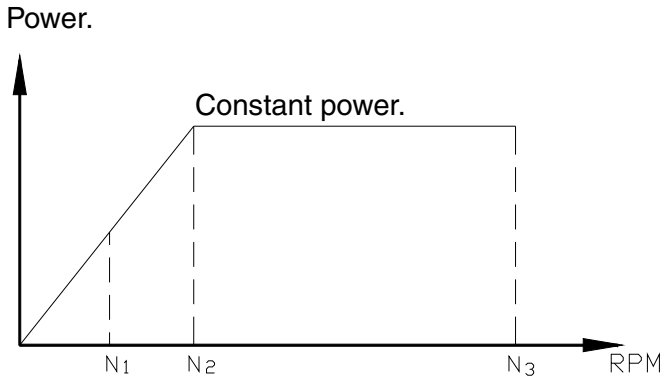
The CNC assumes by default the active gear. Once the gear has been chosen, go into CSS mode.

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·



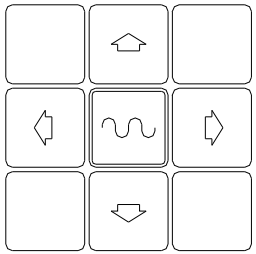
SPEED RANGE (GEAR) ·2·

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

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

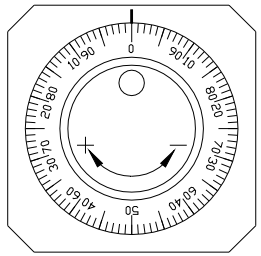
3.4 Jogging the axes.

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.

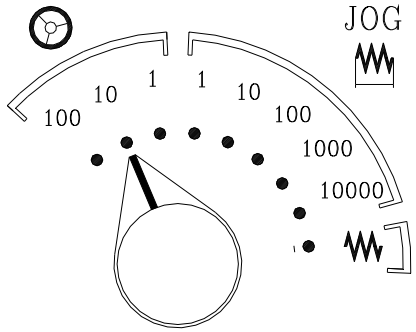


It may have one, two or three handwheels. It follows the turning direction of the handwheel.

Handwheels.

Use the selector switch to select the type of feed.

Movement with handwheels.

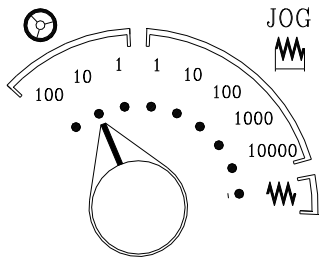


Movement in incremental jog.

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.

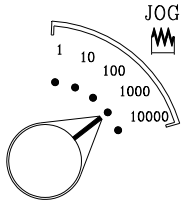
  The machine moves one axis with each handwheel.

3.4.2 Jogging.

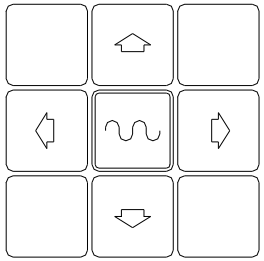
Incremental jog.

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.

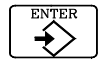
If the feedrate is defined in millimeters per turn, there must be a speed selected at the spindle.

Continuous jog. Feedrate in millimeters per minute.

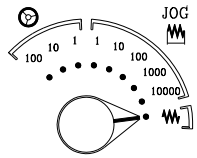
While a jog key is kept pressed, the axes move at the selected feedrate, considering the selected percentage (0% to 120%).

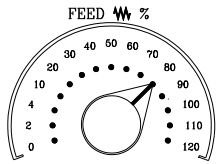
- Enter the desired feedrate.

+ 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.

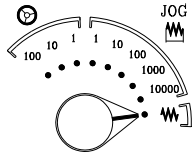
Continuous jog. Feedrate in millimeters per turn.

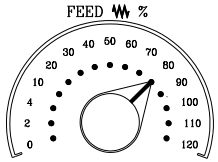
In this mode, the feedrate depends on the status of the spindle (stopped or turning).

- Enter the desired feedrate.

+ 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.

- Spindle status.

  If the spindle is turning, the CNC moves the axes at the selected feedrate.

  If the spindle is stopped, but at the selected turning speed S, the CNC calculates the theoretical feedrate (in mm/min) and moves the axes at that feedrate.

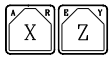
  If the spindle is stopped and no spindle speed S has been selected, the axes will not move.

- 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.

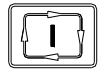
4· Press [START].

Machine with automatic tool changer.

The CNC manages the tool change.

1· Press the [T] key.

2· Enter the tool number.

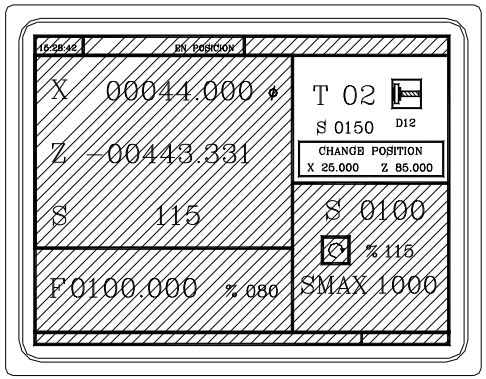


3· Press [START].

4· The CNC will rotate the tool-holder until the new tool is in the work position.

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, and Z, the values of the point where the tool change will take place.

T + X + (X value) + ENTER

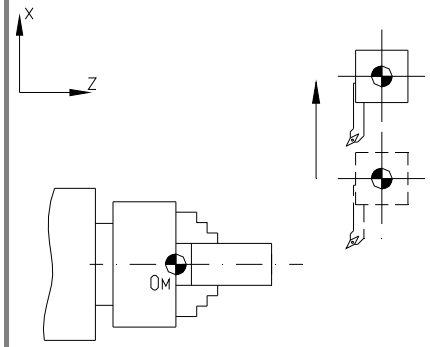
T + Z + (Z value). + ENTER

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.

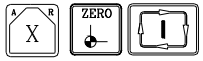
3.5.2 Tool calibration.

- The machine must be homed before calibrating the tools.

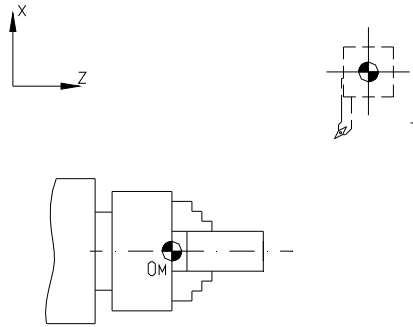
Homing the X axis.



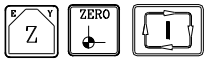
Machine Ref.

Press: 

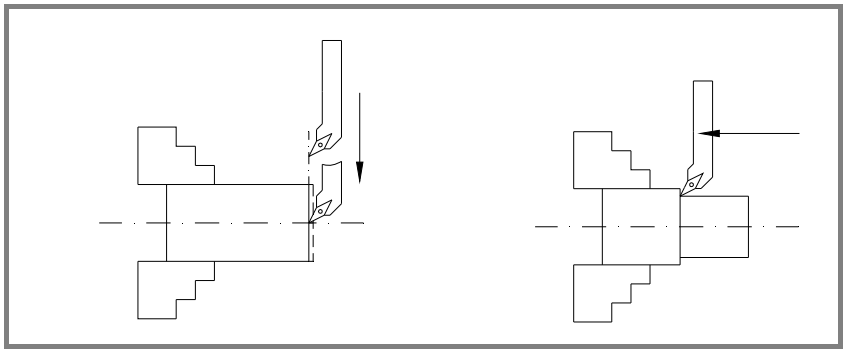
Homing the Z axis.



Machine Ref.

Press: 

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

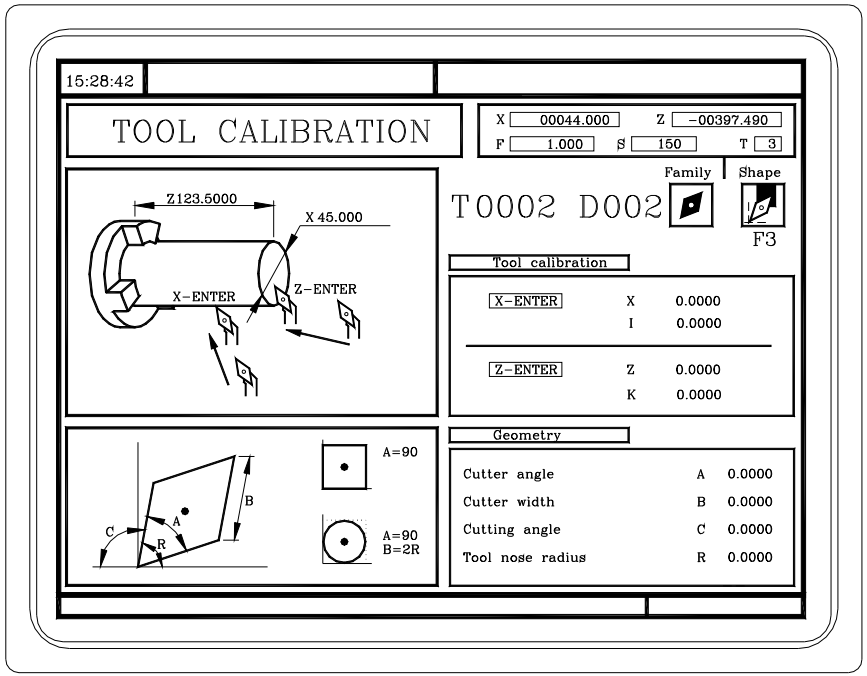


• Go into tool calibration mode. The CNC shows the screen for tool calibration.

Work mode.

Help graphics and part dimensions used to calibrate the tool.

Help graphics.



Real position of the axes and cutting conditions.

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

Tool length and wear.

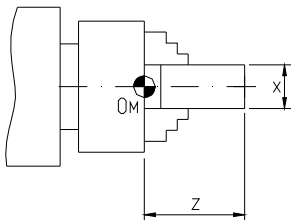
Tool geometry.

Use the keys to move the cursor.

Use the two-color key to change an icon.

• Calibrate the tool as follows.

1· Measure the part.

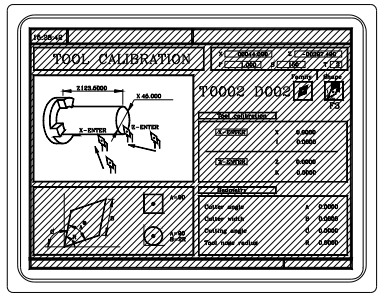


Part dimensions.

• Go to the window for tool calibration.

• Enter the X value.

• 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.

T + (Tool number) + ENTER

4· Jog the axes until touching the part along the X axis.

Press [X]+[ENTER]. The CNC calculates and shows the new length. The offset value (I) is reset to 0.

Press [Z]+[ENTER]. The CNC calculates and shows the new length. The offset value (K) is reset to 0.

5· Enter the rest of the data (angle, width, radius and location 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.

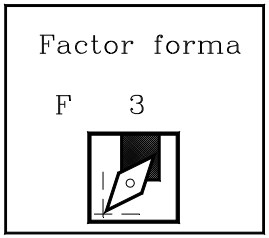
To change the tool data (T, D, A, B, C, R, location code, I, K) get into tool calibration mode and display the tool data.

+ (Tool number) +

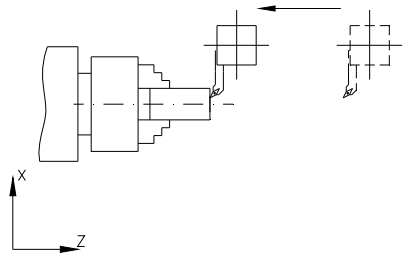
The CNC shows the data for that tool. To change them, place the cursor on the value to be modified, key in the new value and press [ENTER]. To select an icon, use the two-color key and press [ENTER]. To exit the calibration option, press [ESC].

3.5.4 Checking for proper calibration.

- Presetting the part zero.

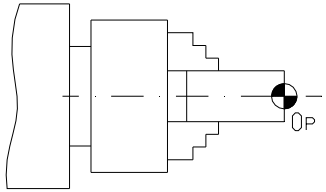


Select a tool; for example with location code 3.



Approach the tool in Z.

Press:



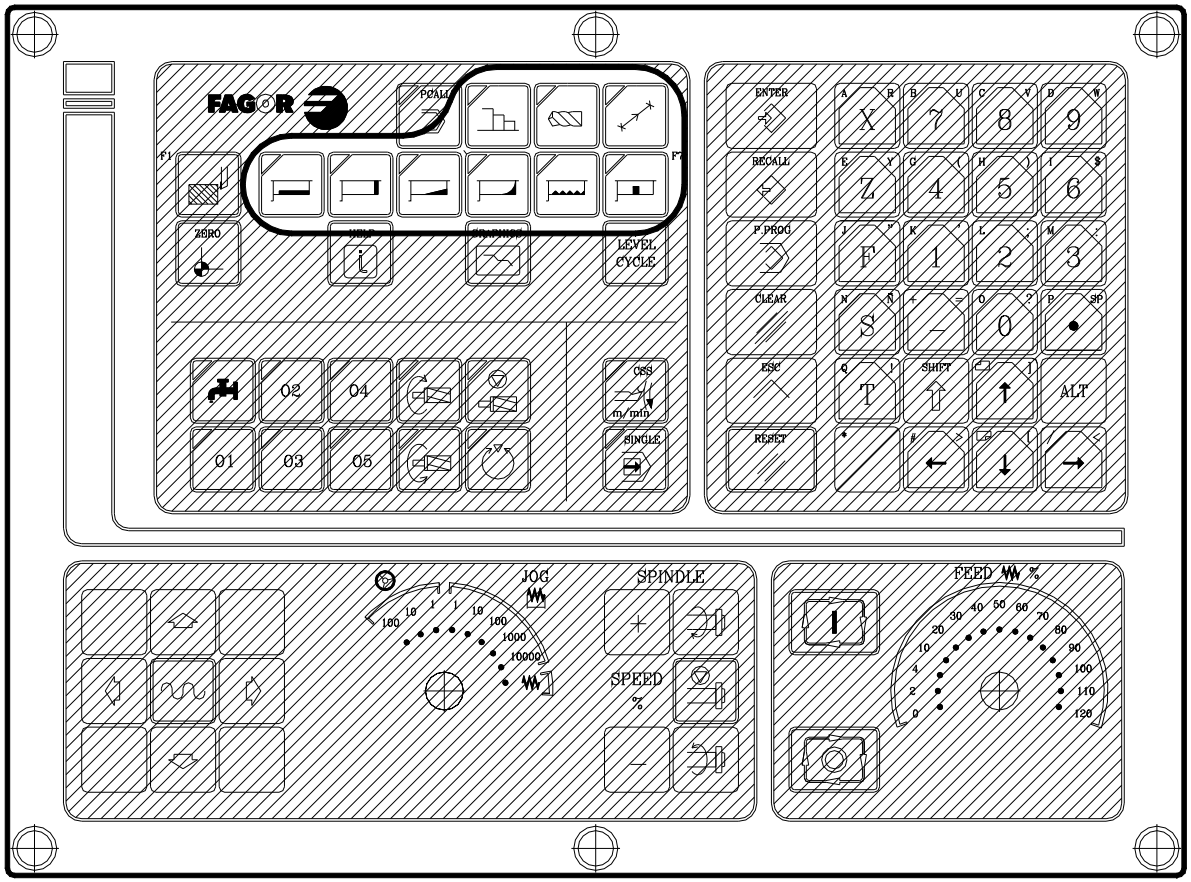
Withdraw the tool. The part zero is set.

- Start the spindle and touch the diameter of the part 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.

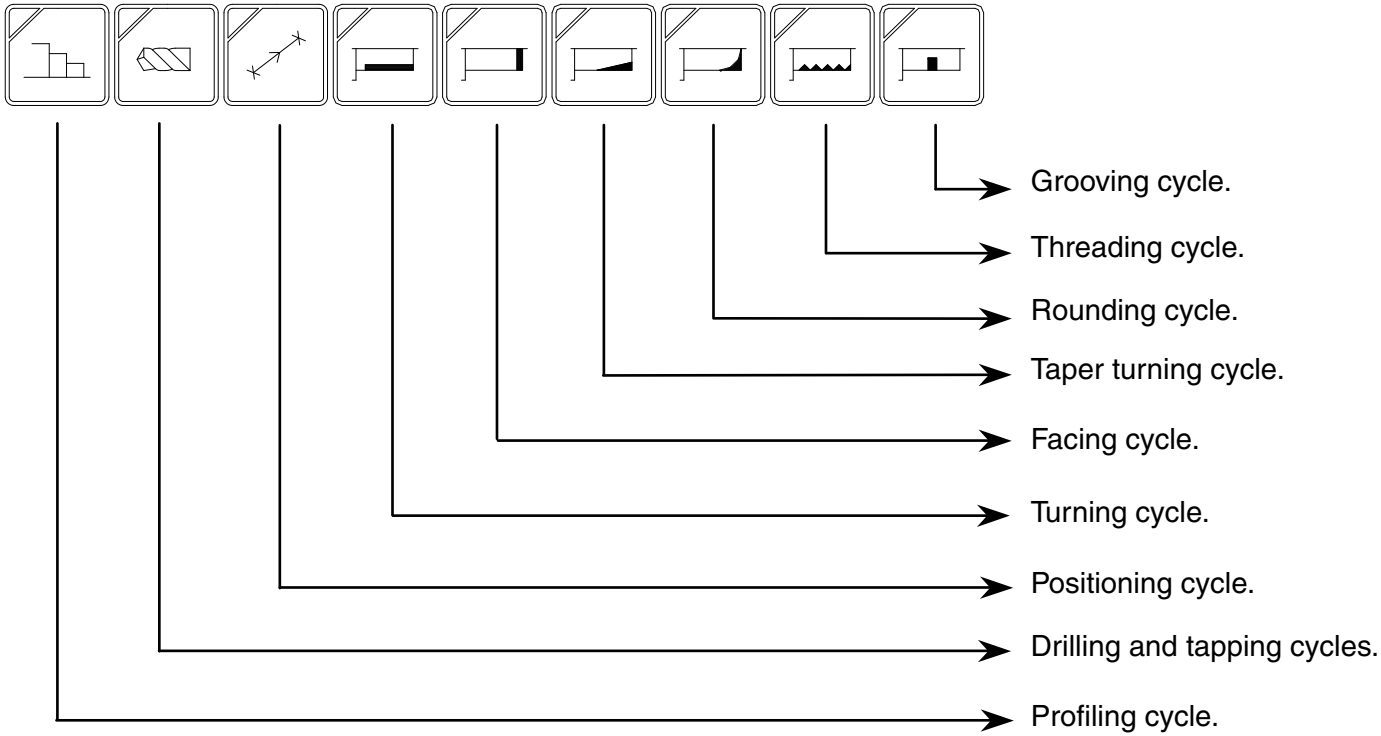
- Keys associated with automatic operations.
- Different work modes.
- Editing the parameters of an operation.
- Simulating an operation.
- Executing an operation.

4.1 Description of the screen and the keyboard.



Position of the automatic function keys.

Operation keys.

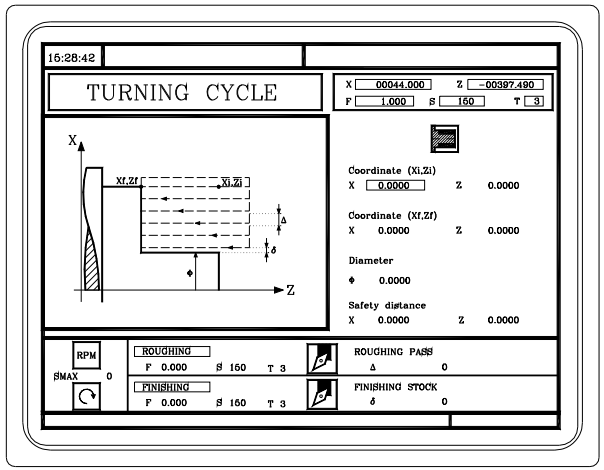


LEVEL CYCLE Selecting the cycle level.

4.2 Work modes.

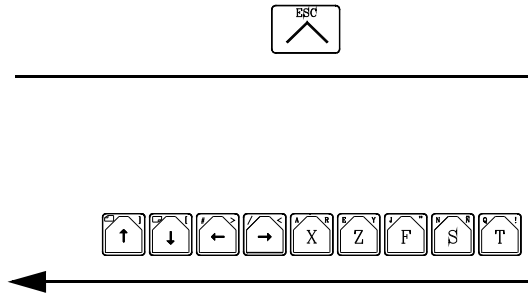
There are two work modes.

Editing mode.

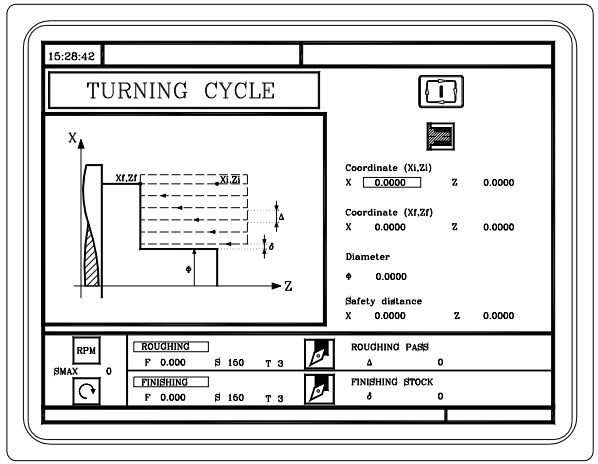


Editing the parameters of the operation or cycle.

GRAPHICS Simulating an operation or cycle.



Execution mode.

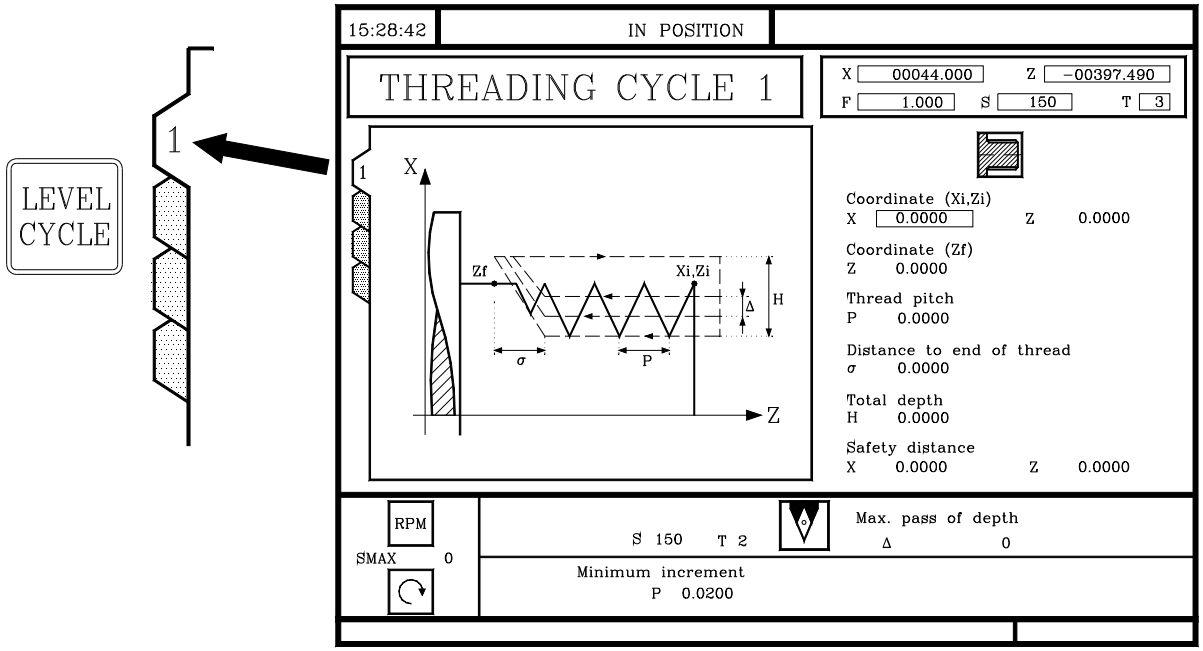


GRAPHICS Simulating an operation or cycle.

Executing an operation or cycle.

4.3 Operations or cycles.

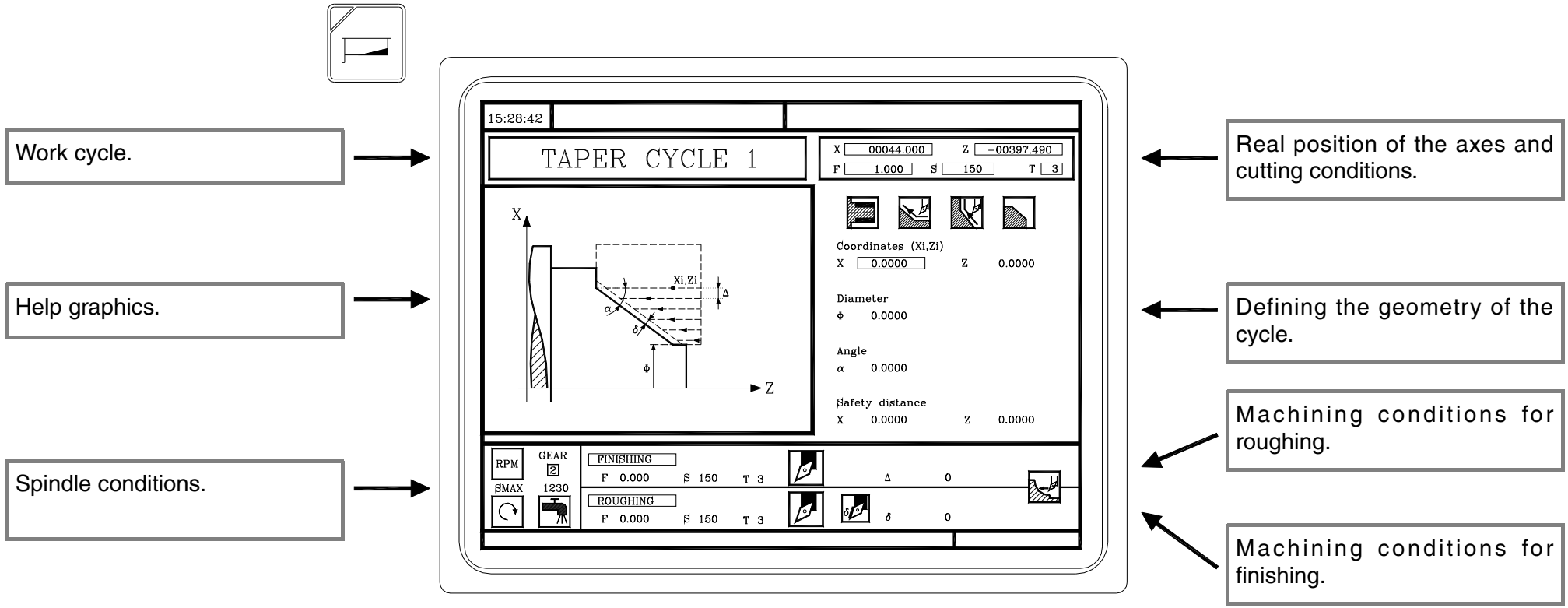
All the cycles have several editing levels. Each level has its own screen and the main window of the cycle indicates, with tabs, the available levels and which one is selected.



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.

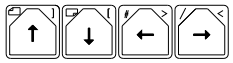
As an example, select the taper turning cycle.



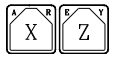
Use the [LEVEL CYCLE] key to select the cycle level to be executed. Remember that not all the cycles have several levels.

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.

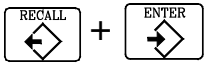
The CNC selects the roughing tool. Pressing it again selects the finishing tool.

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

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].

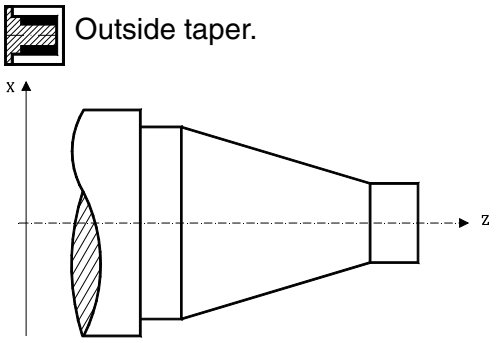


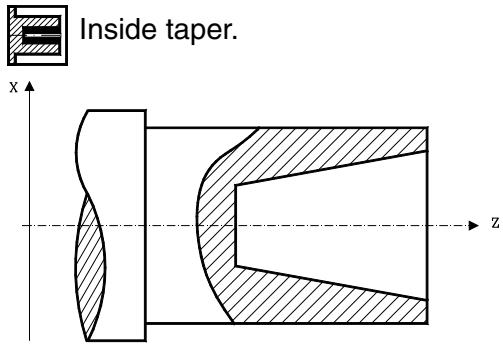
If it is a coordinate, there are two choices:

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

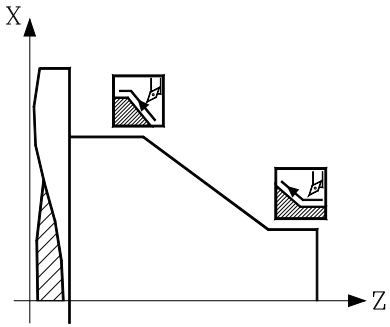
The icons show the following data: When selecting an icon, the CNC shows its meaning at the bottom of the screen.

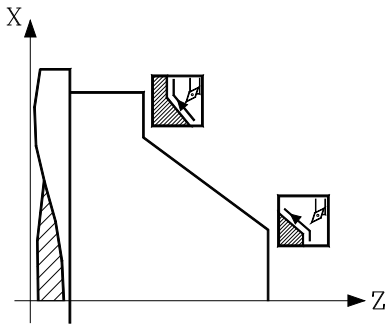
- Type of taper.



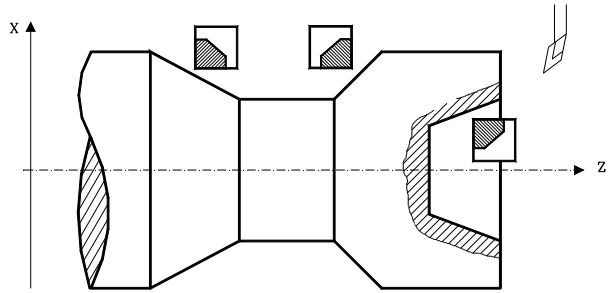


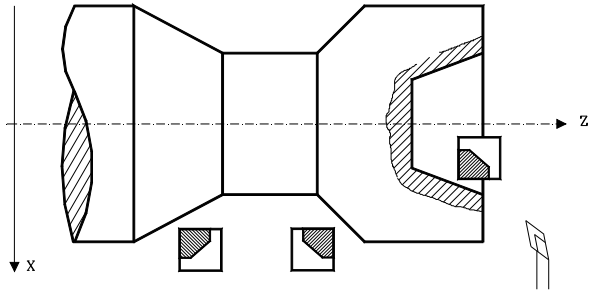
- Type of section before and after the taper.



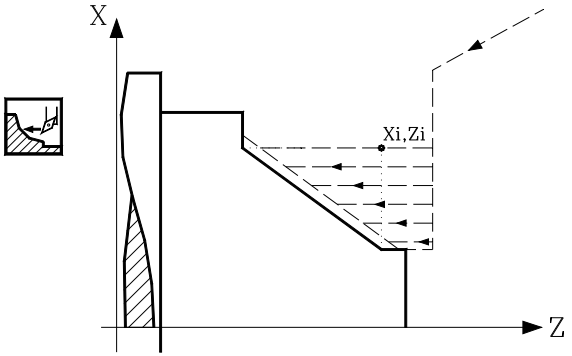


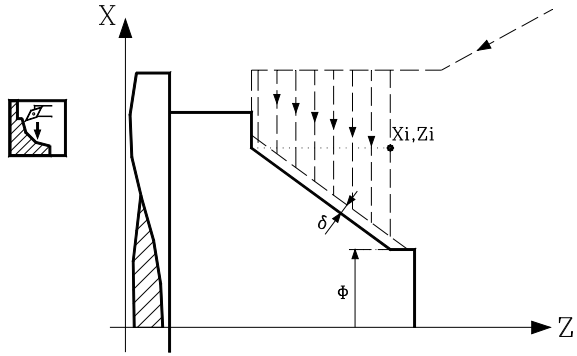
• Work quadrant. This icon indicates the type of corner to be machined.





• Machining direction.





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.

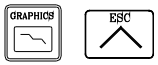
The simulation speed may be varied with the Feedrate Override switch.

Other useful keys are.

Interrupt the simulation.

If the simulation is interrupted, it resumes it.

If the simulation is interrupted, it cancels it definitely.

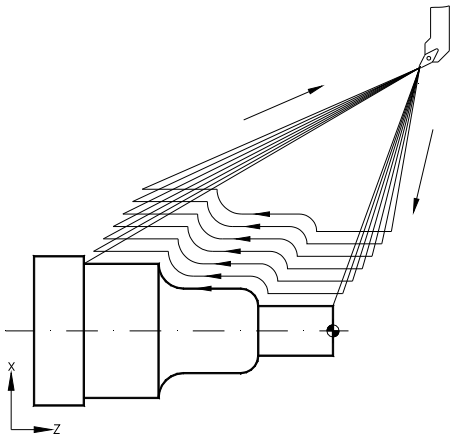


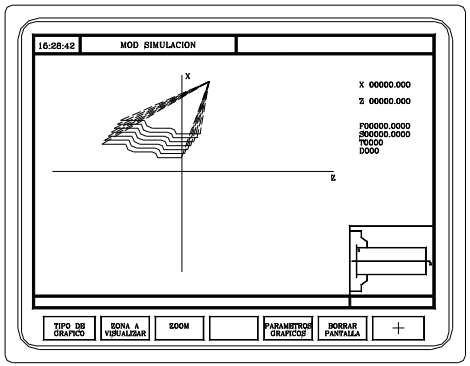
Any of these two keys quits the simulation mode.

GRAPHIC OPTIONS. TYPE OF GRAPHIC.

- "3D" graphics.

  It uses color lines to draw the tool path. When selecting this type of graphics, the screen only shows half of the part. This is because it only shows the path of the tool tip (not the part itself).





Graphic representation.

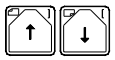
To later check the part dimensions later on the simulation screen, it must be done with a tool whose tip (nose) radius R=0.

- "Solid" graphics.

  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.



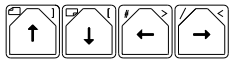
Selecting the coordinates.

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 in XZ graphics.

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

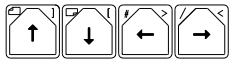
- Depending on the manufacturer, accessing tool inspection on some machines may require pressing also the [T] key.
- The top of the CNC screen shows the message "INSPECTION". Within the "tool inspection" mode, it is possible to move the axes (jog or handwheels), check the tool, change the tool, stop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

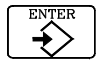
- Key in the I value and press [ENTER].

- 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) +

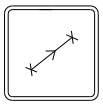


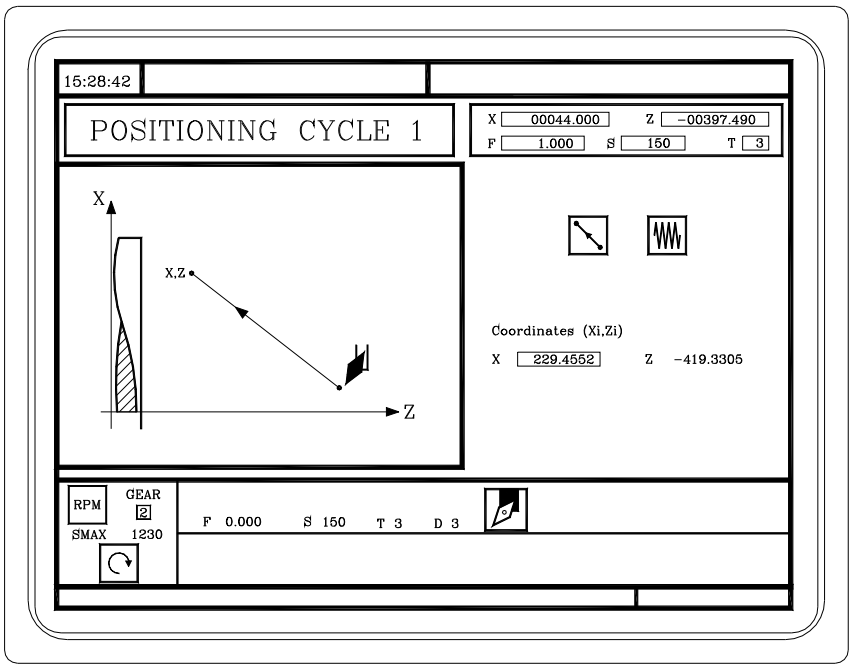
- Press [ESC] to end it.

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

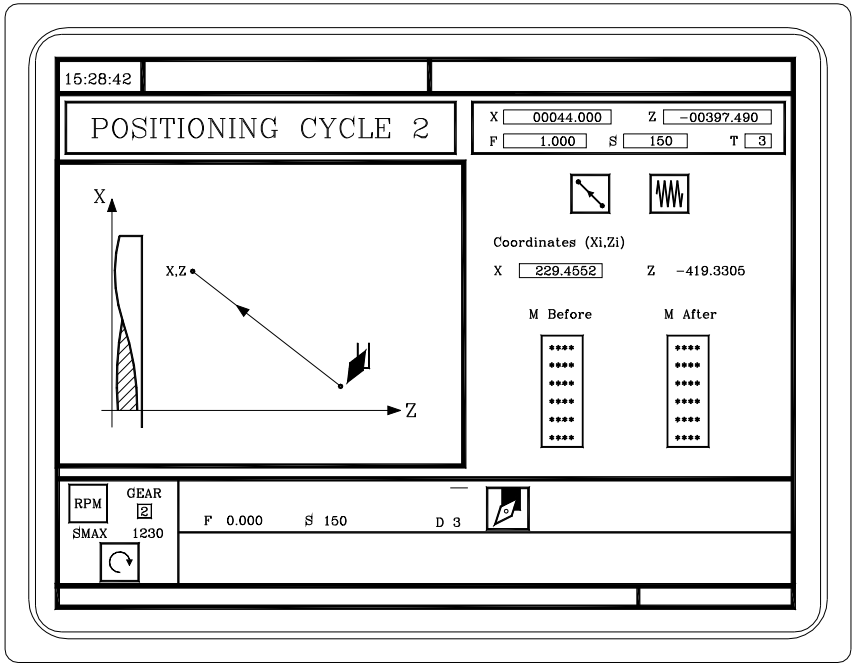
5 SUMMARY OF WORK CYCLES.

5.1 Positioning cycle.



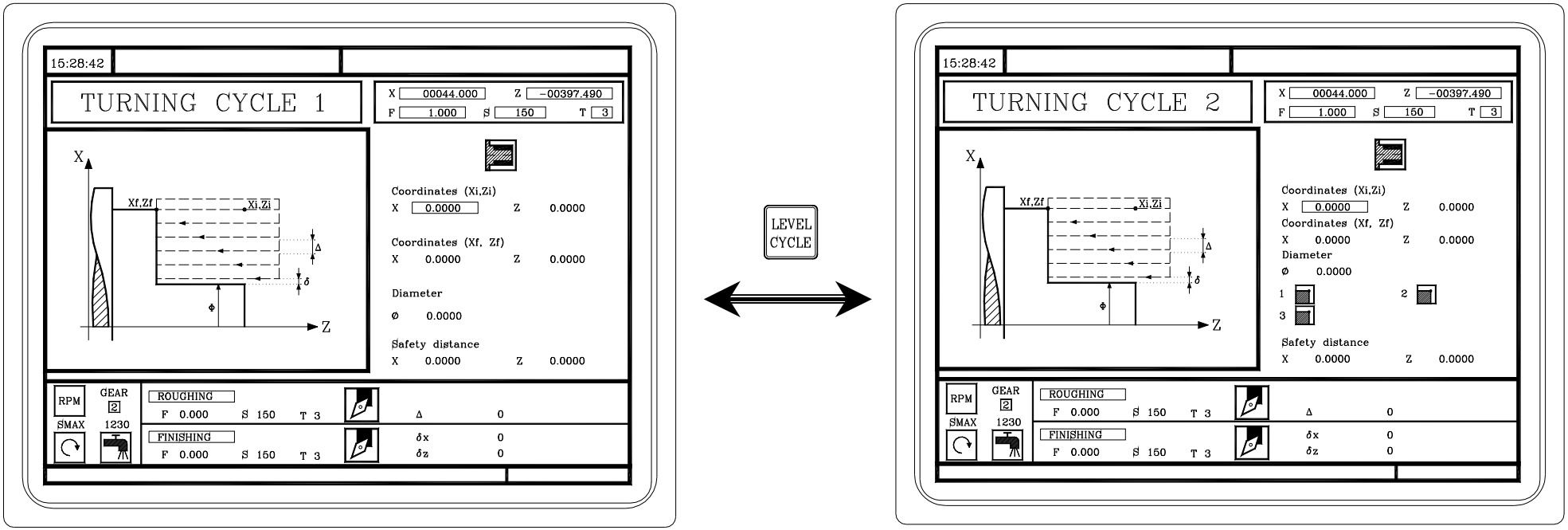


LEVEL CYCLE

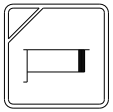


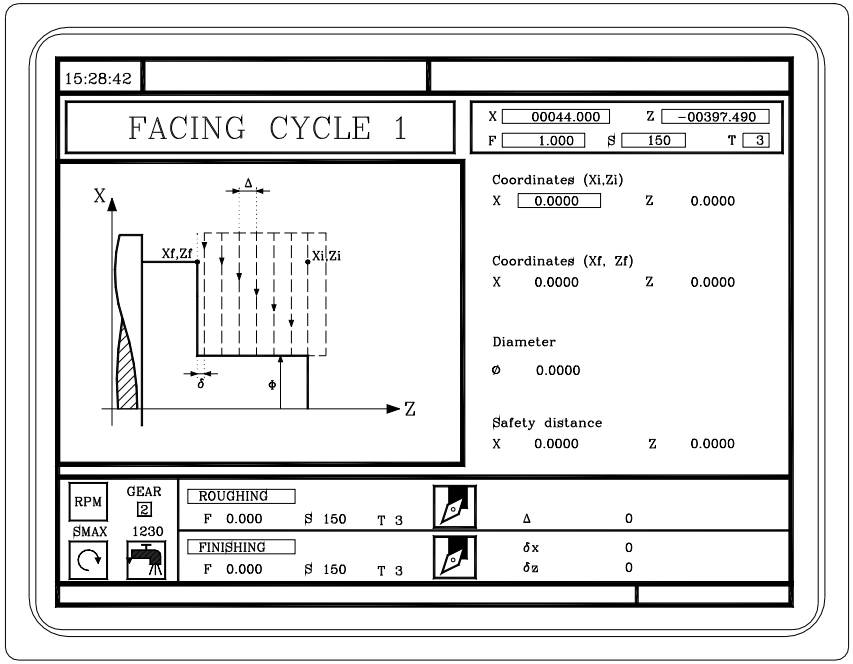
At this cycle level, it is possible to define the auxiliary functions that will be executed before or after the movement.

5.2 Turning cycle.

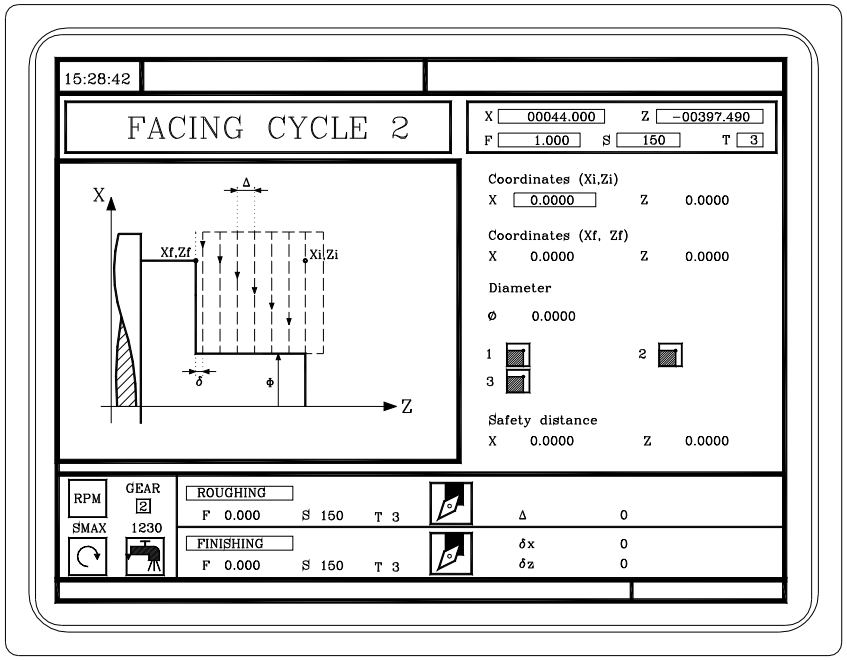


5.3 Facing cycle.

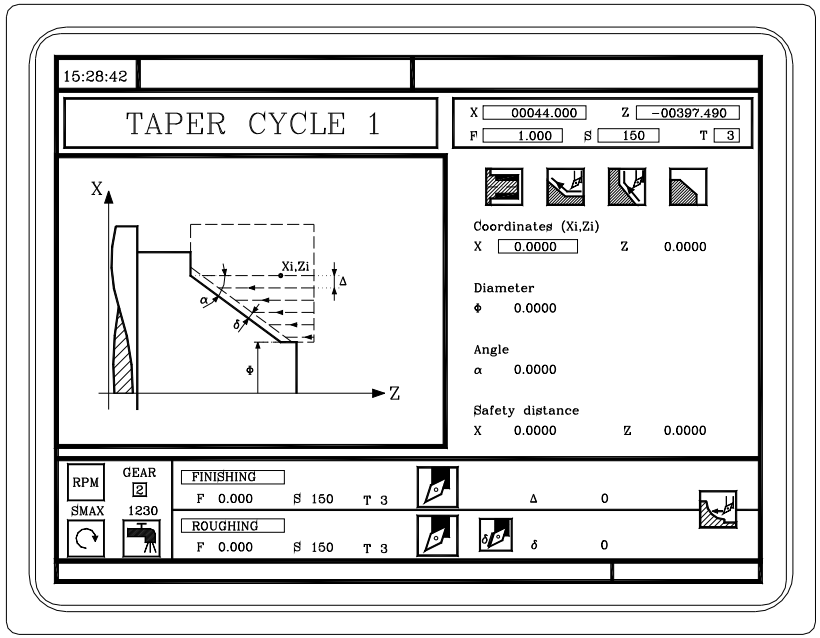




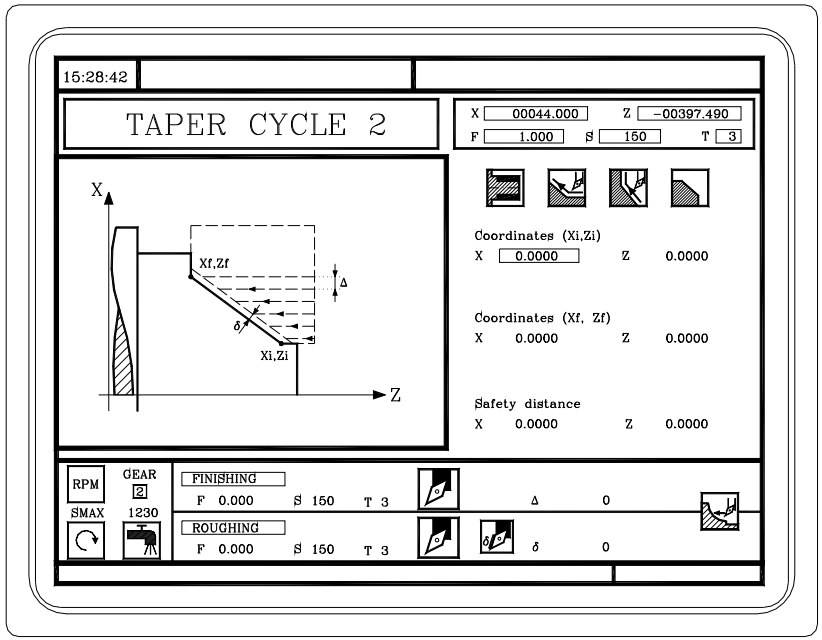
LEVEL CYCLE



5.4 Taper turning cycle.



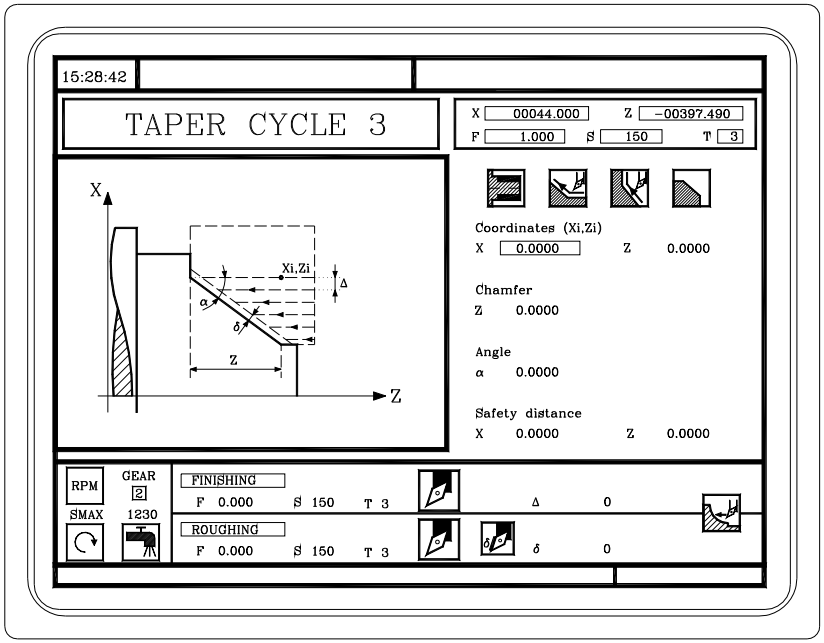
LEVEL CYCLE

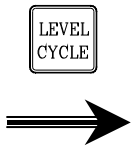


LEVEL CYCLE

In this level, it is necessary to define the coordinates of the theoretical corner, the angle of the taper and the final diameter.

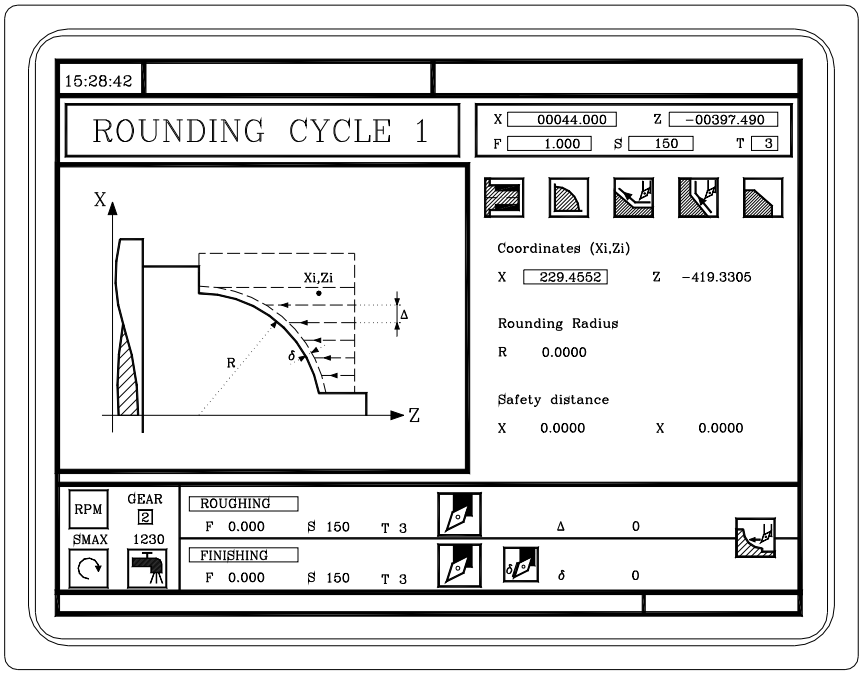
In this level, it is necessary to define the coordinates of the starting and final points.



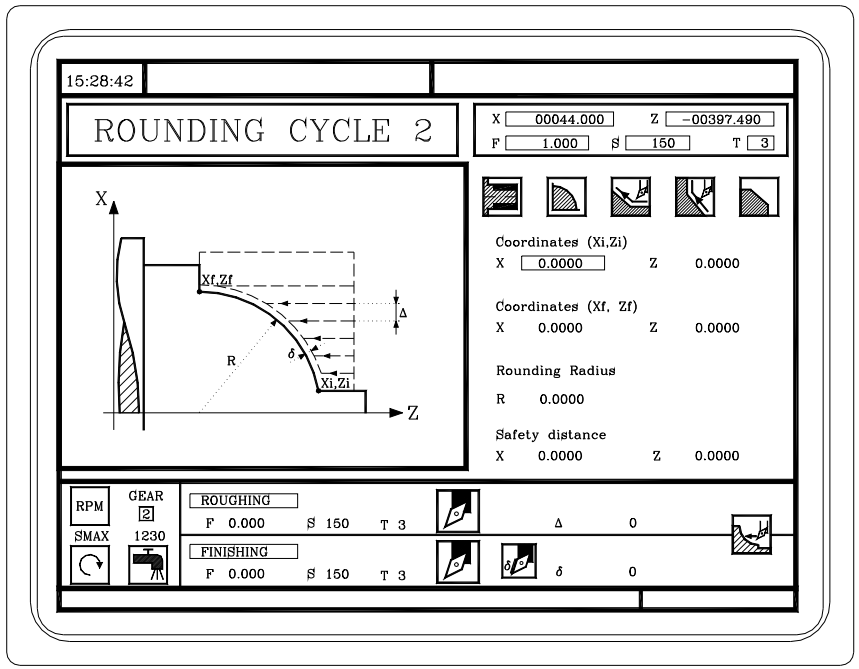


In this level, it is necessary to define the coordinates of the theoretical corner, the angle of the taper and the distance in Z.

5.5 Rounding cycle.

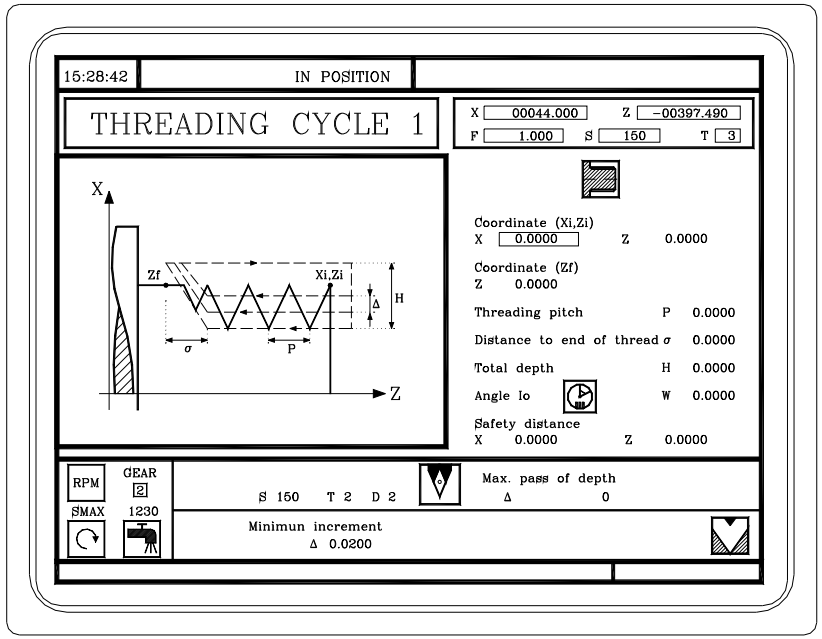


In this level, it is necessary to define the coordinates of the theoretical corner and the rounding radius.



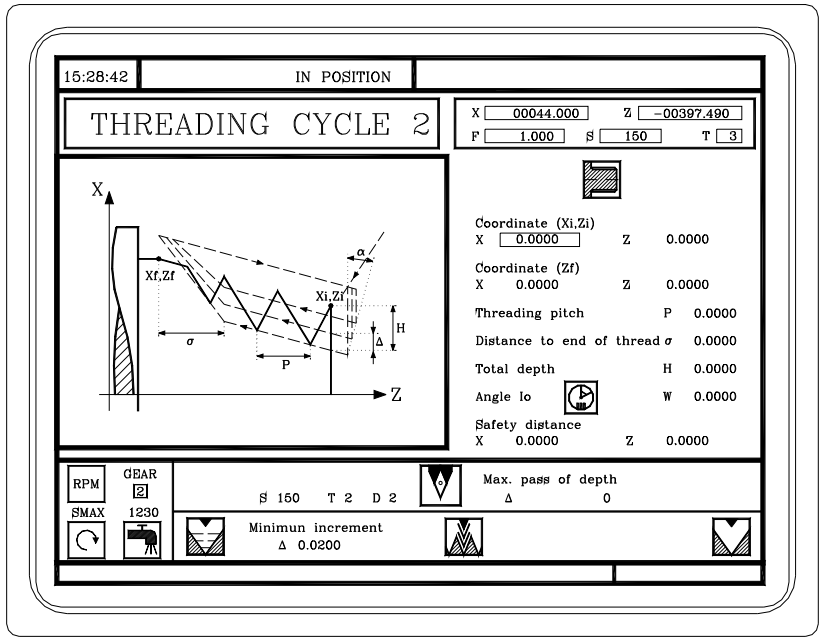
In this level, it is necessary to define the coordinates of the starting and final points and the rounding radius.

5.6 Threading cycle.



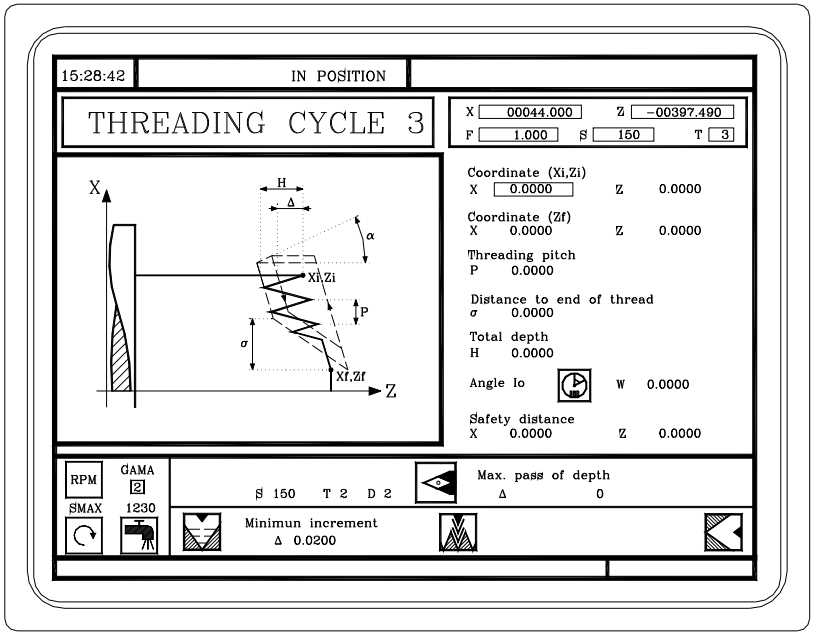
Longitudinal threadcutting.

LEVEL CYCLE



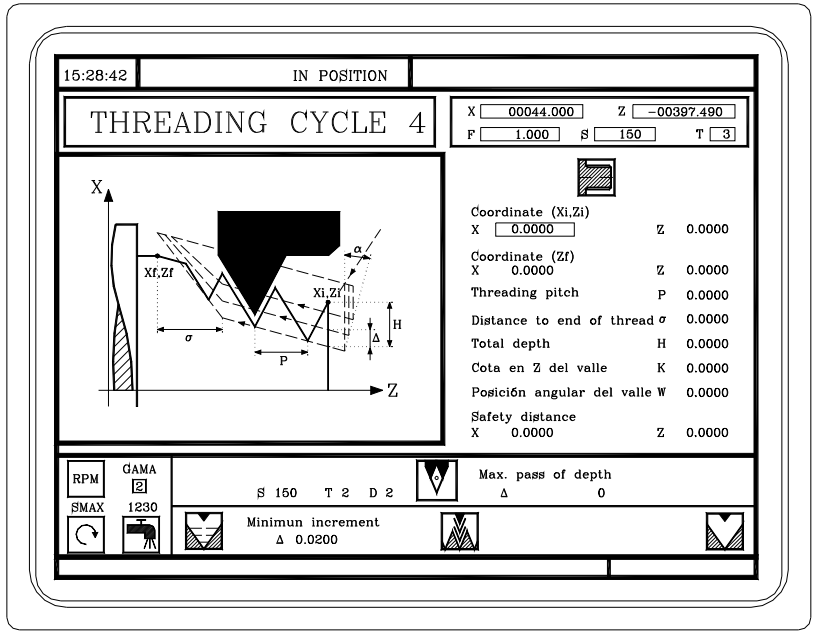
Taper thread.

LEVEL CYCLE



Face tapping.

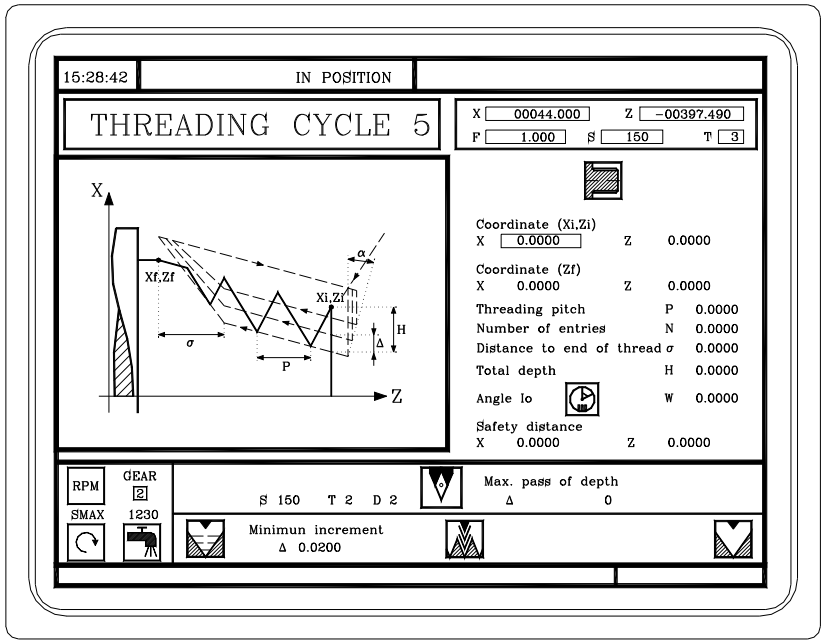
LEVEL CYCLE

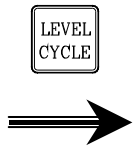


LEVEL CYCLE

Thread repair.

This level is only available when using spindle orientation.

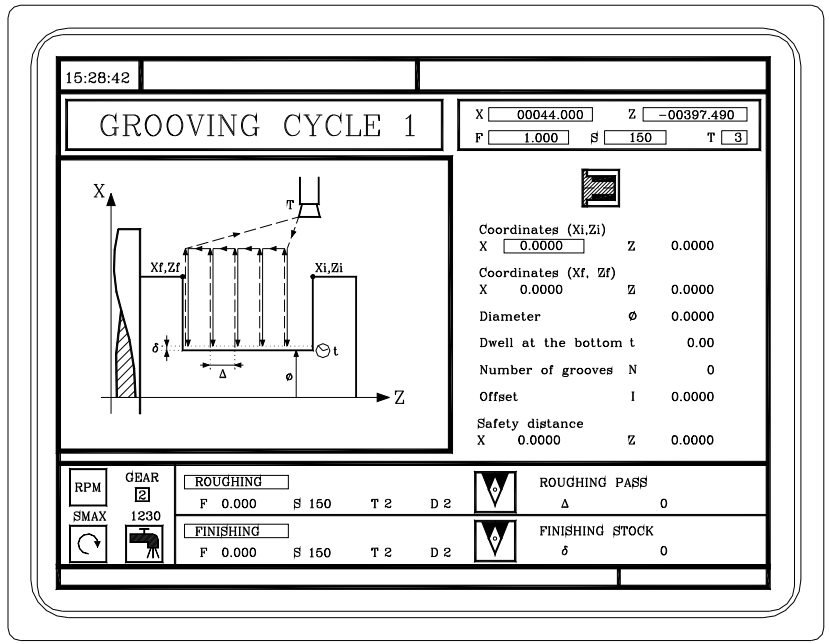




Threads with several entries.

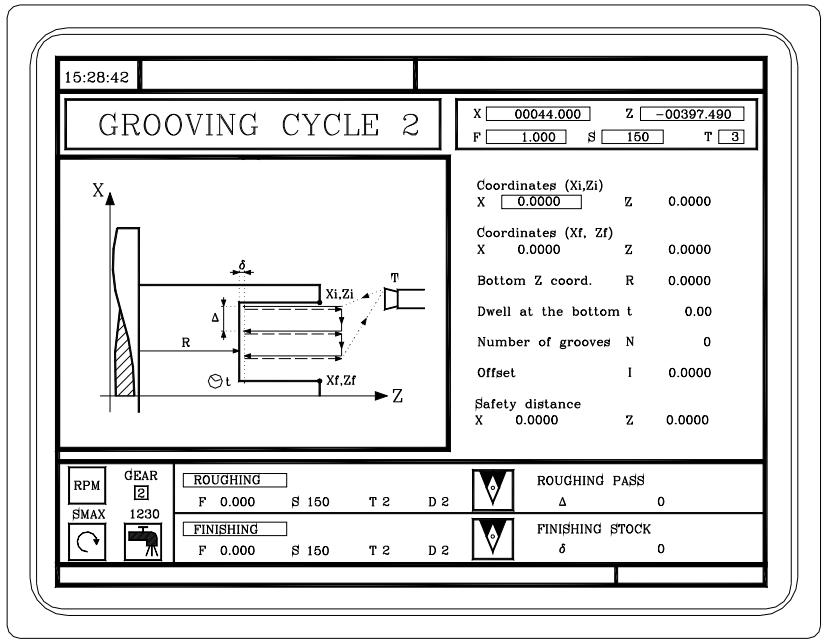
This level is only available when using spindle orientation.

5.7 Grooving cycle.



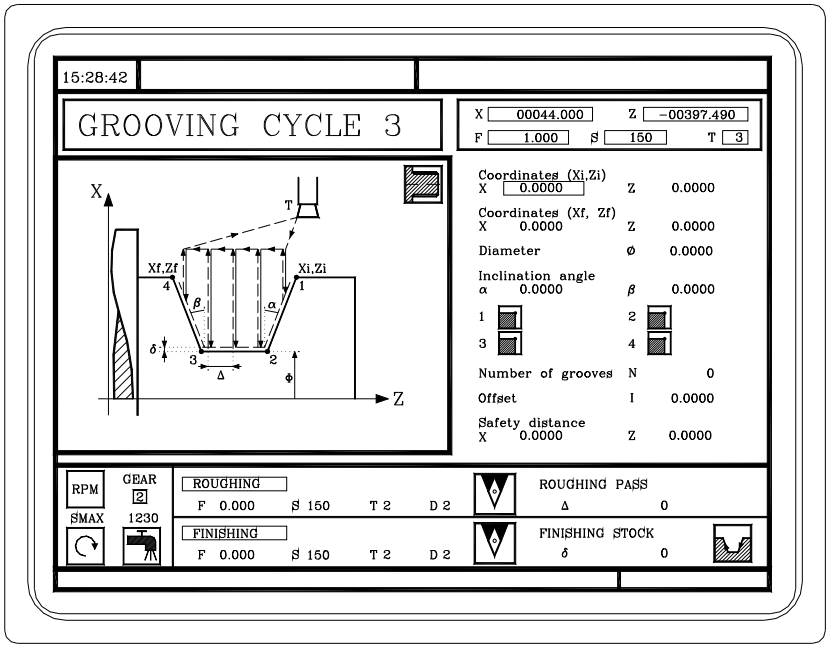
Grooving on the cylindrical side.

LEVEL CYCLE



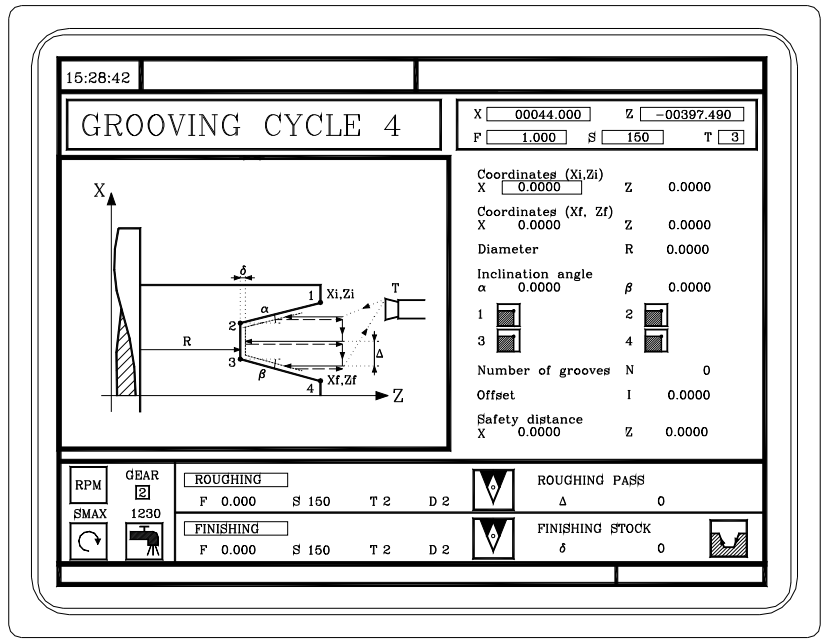
Face grooving.

LEVEL CYCLE



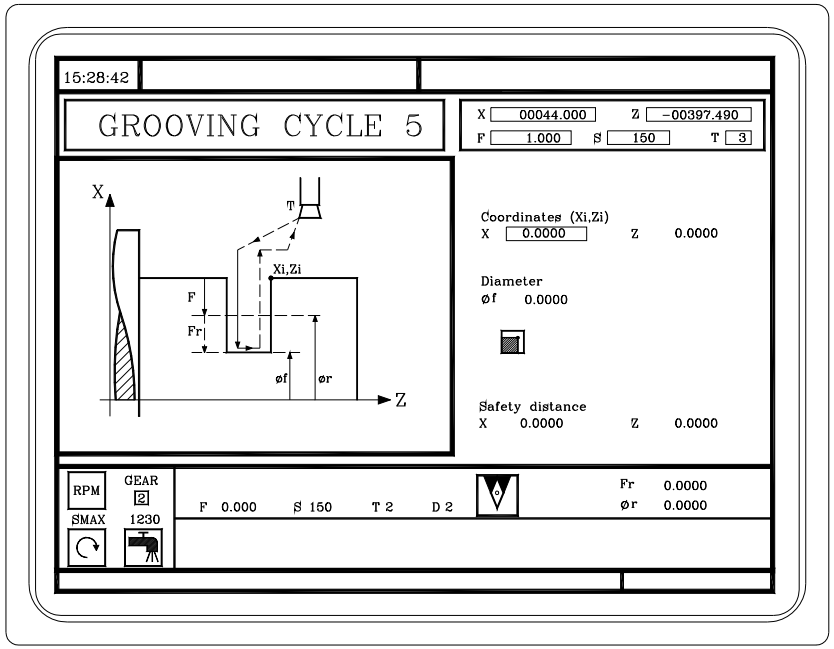
Grooving on the cylindrical side with incline walls.

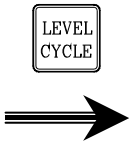
LEVEL CYCLE



Grooving on the face with incline walls.

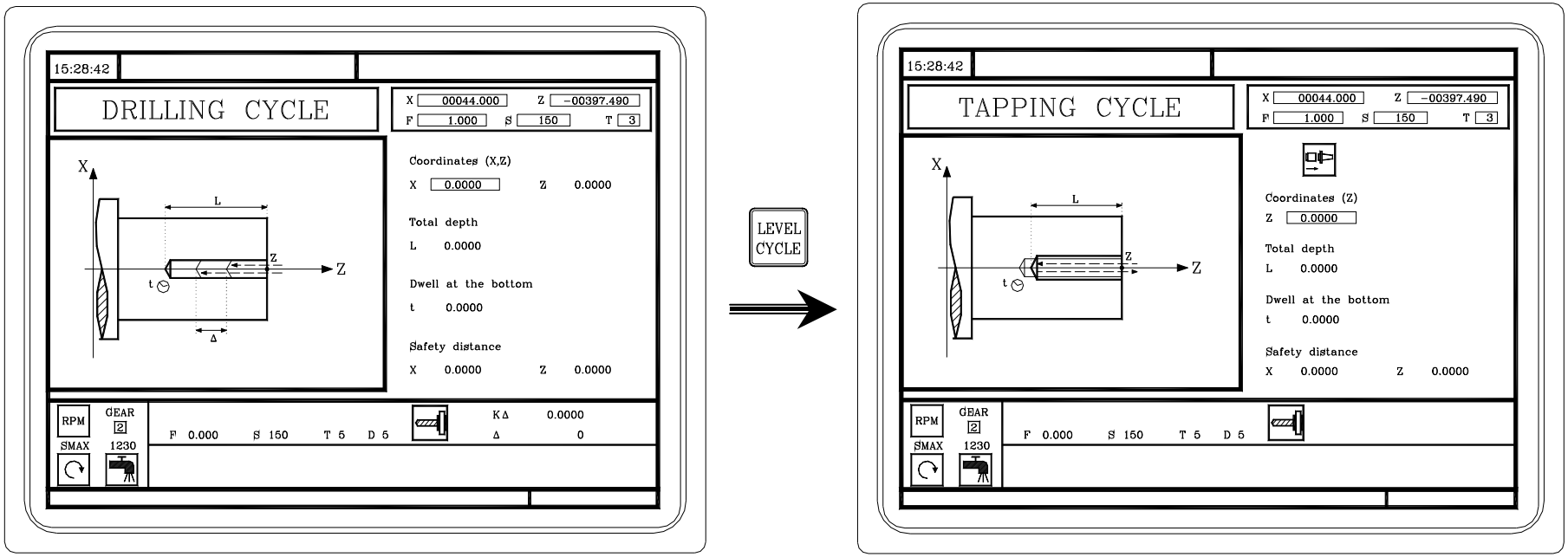
LEVEL CYCLE


15:28:42
GROOVING CYCLE 5
X 00044.000
Z -00397.490
F 1.000
S 150
T 3
X
T
Xi,Zi
F
Fr
øf
ør
Z
Coordinates (Xi,Zi)
X 0.0000
Z 0.0000
Diameter
øf 0.0000
Safety distance
X 0.0000
Z 0.0000
RPM
GEAR
2
SMAX
1230
F 0.000
S 150
T 2
D 2
Fr 0.0000
ør 0.0000


LEVEL
CYCLE

Cut-off.

5.8 Drilling and tapping cycles.

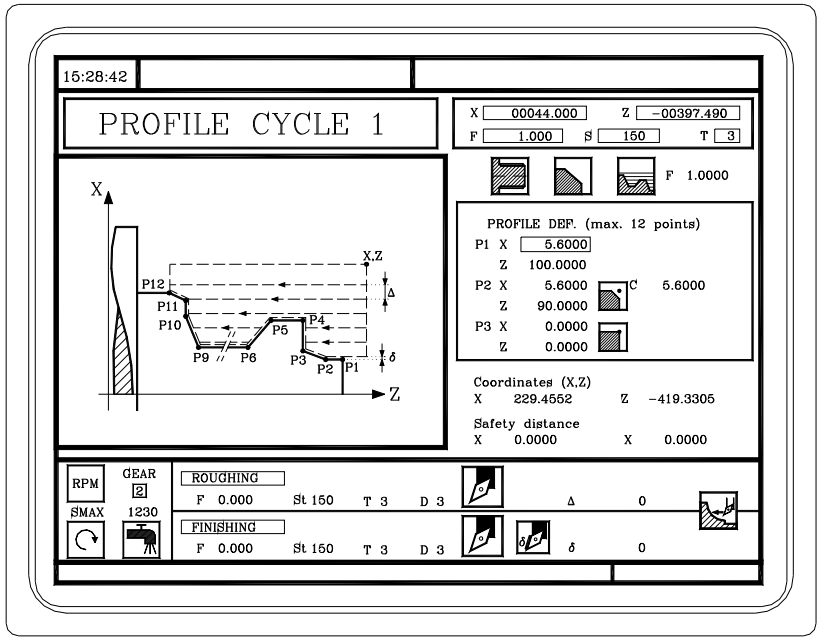


Drilling cycle.

Tapping cycle.

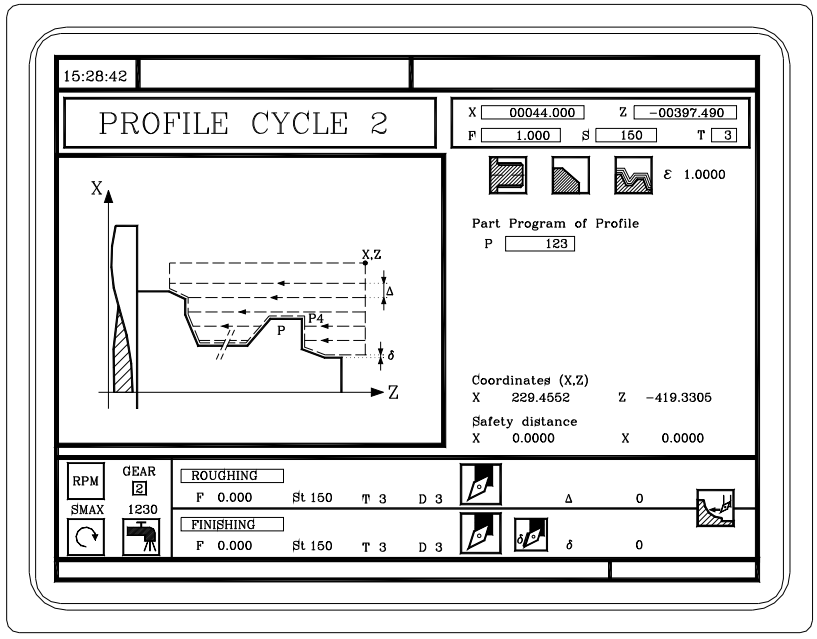
NOTE: When having an orienting spindle, it will also show the cycles described in appendix A.

5.9 Profiling cycle.



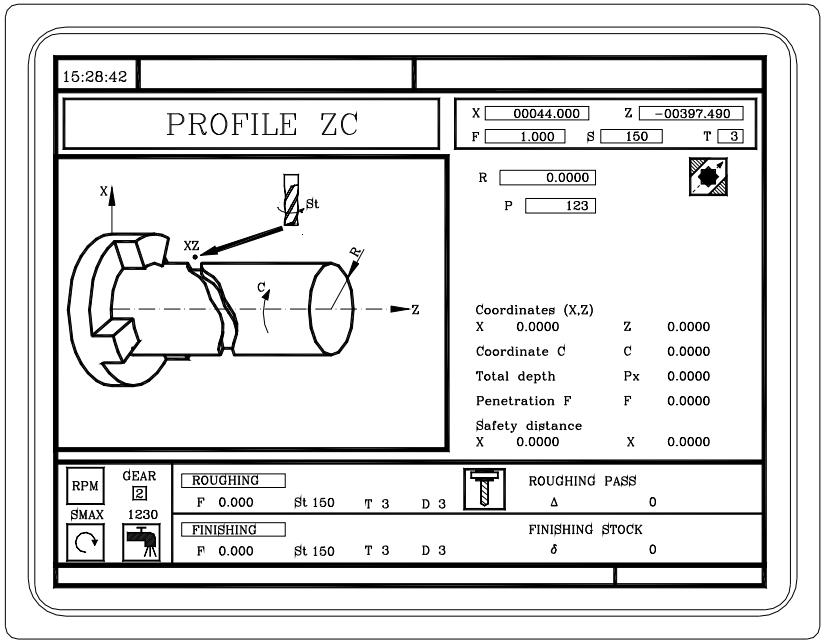
In this level, it is necessary to define all the points of the profile.

LEVEL CYCLE



In this level, the profile is defined in a part-program.

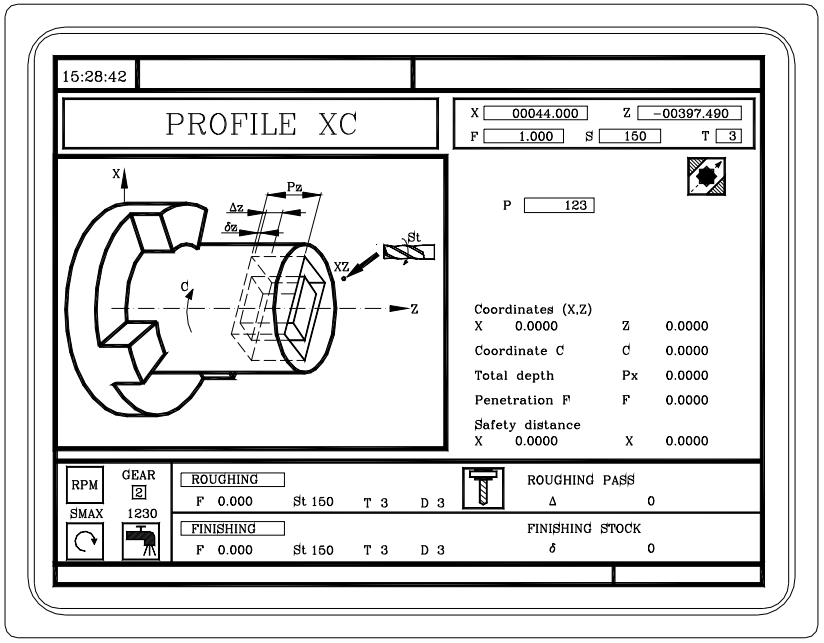
LEVEL CYCLE



ZC profile.

This level is only available when using the C axis.

LEVEL CYCLE



XC profile.

This level is only available when using the C axis.

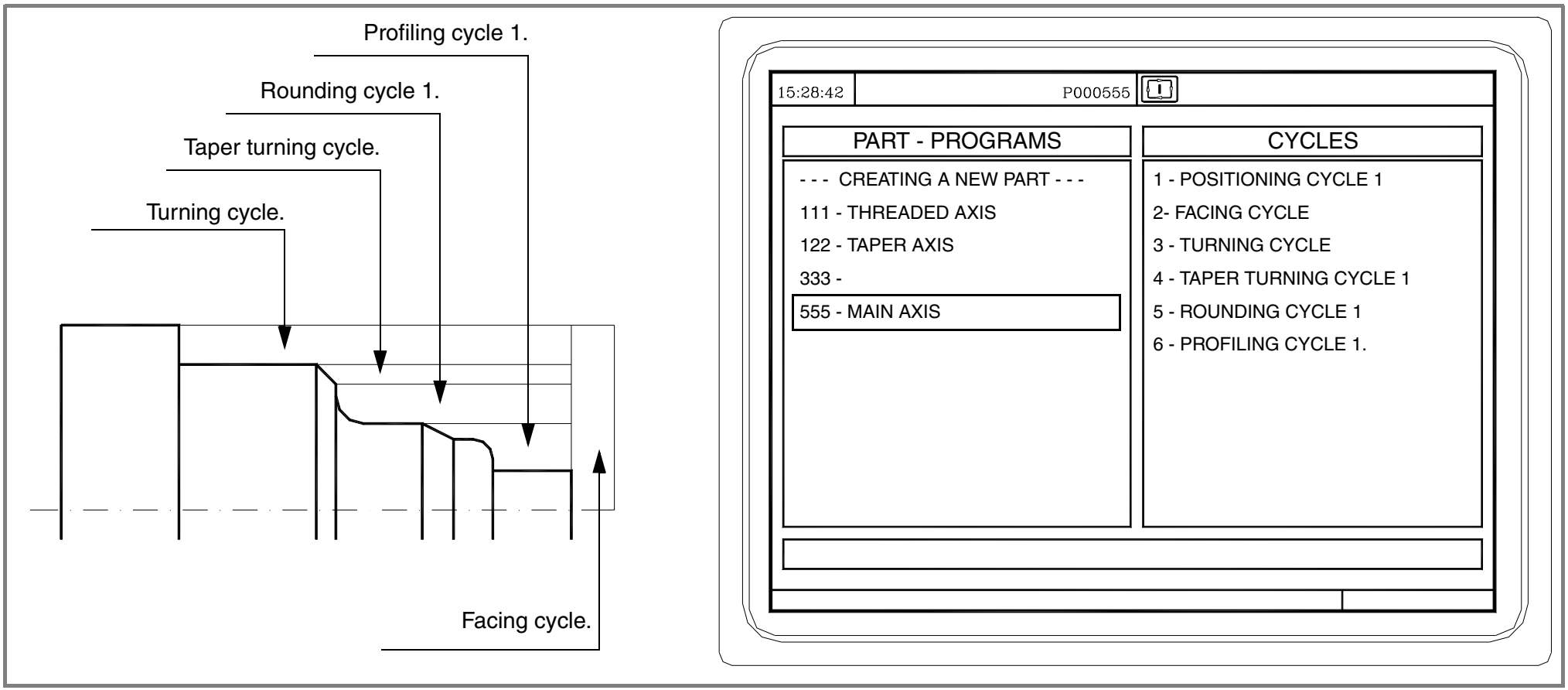
LEVEL CYCLE

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.

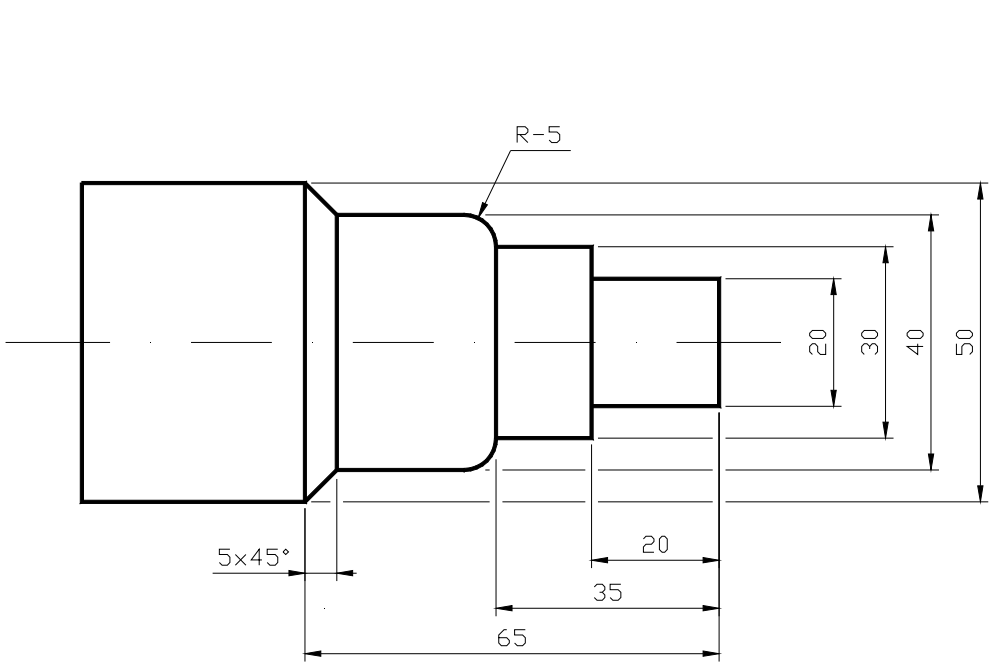
6.1 What is a conversational part-program?

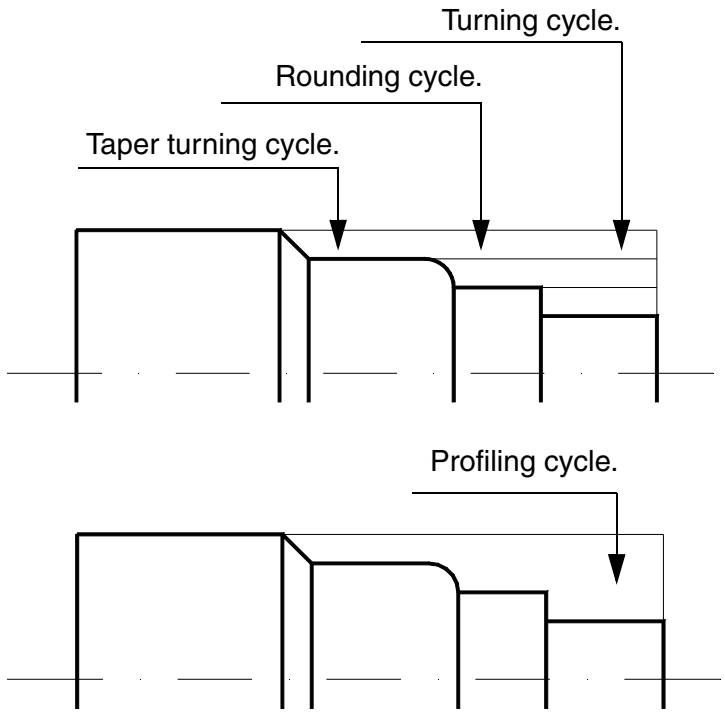
A conversational part-program consists in a set of operations ordered sequentially. Each operation is defined separately and they are then saved one after another in a program. The name of the part-program may be a number between 1 and 899999.



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.



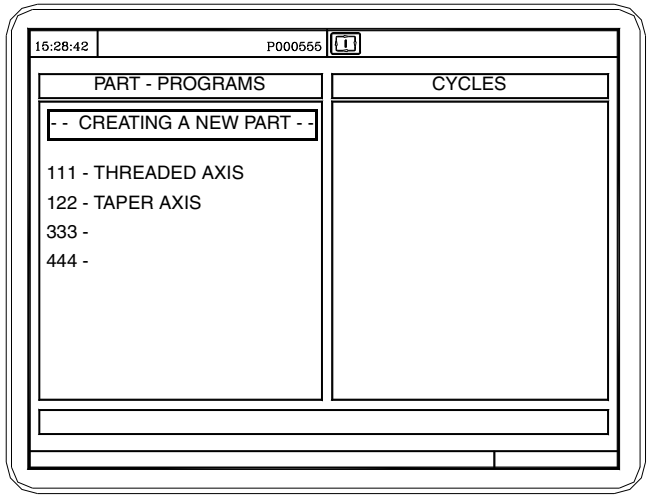


Different solutions for the same part.

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

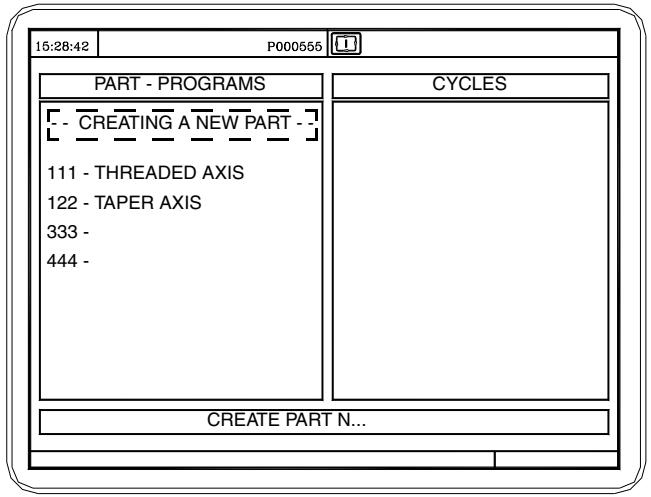
STANDARD SCREEN

P.PROG



Select "Creating new part" with the focus.

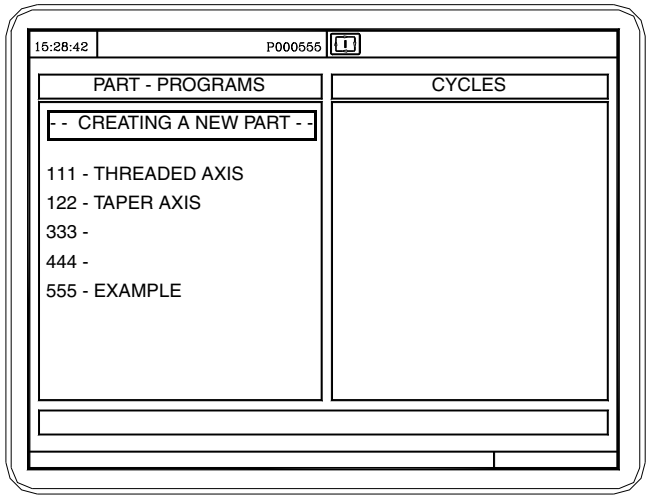
P.PROG



Enter the part-program number.

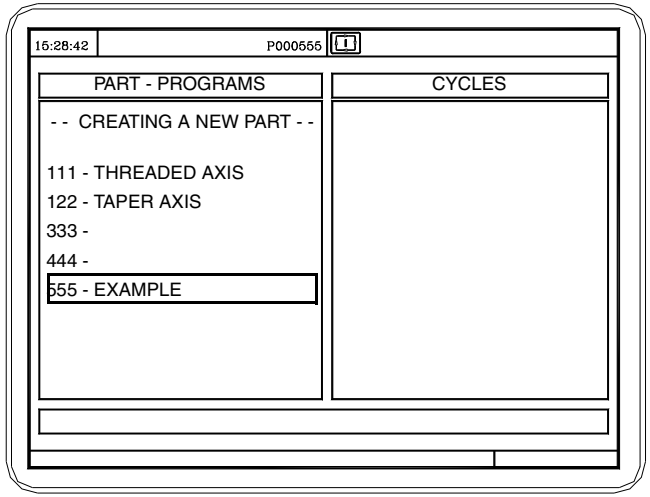
(Type number) + [ENTER] + (comment) + [ENTER]

555 + [ENTER] + EXAMPLE + [ENTER]

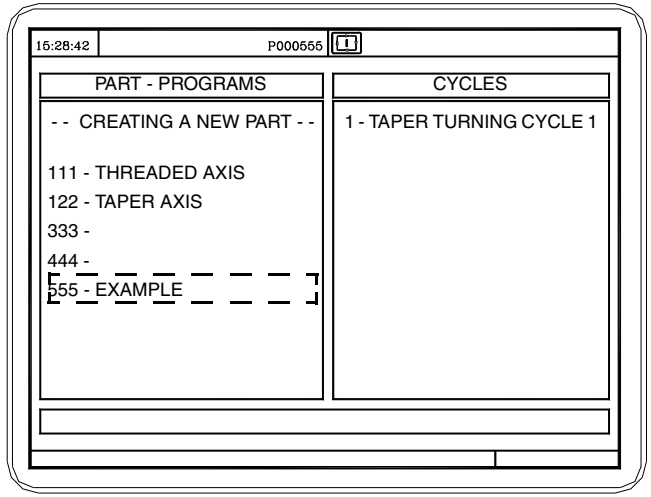


Select an operation and define its parameters. When the operation is defined, press [P.PROG].

P.PROG

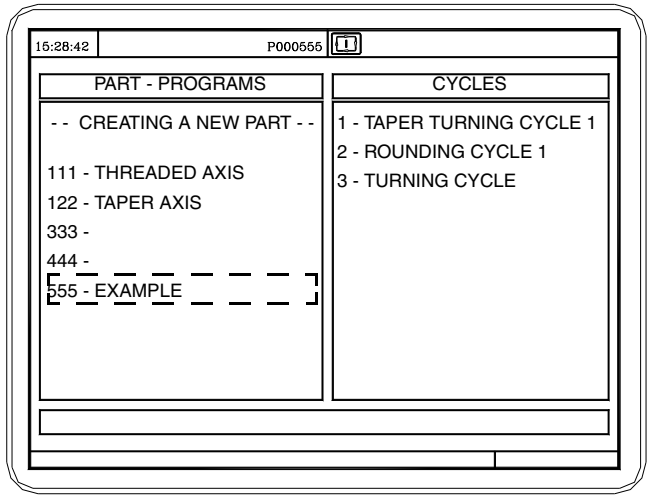


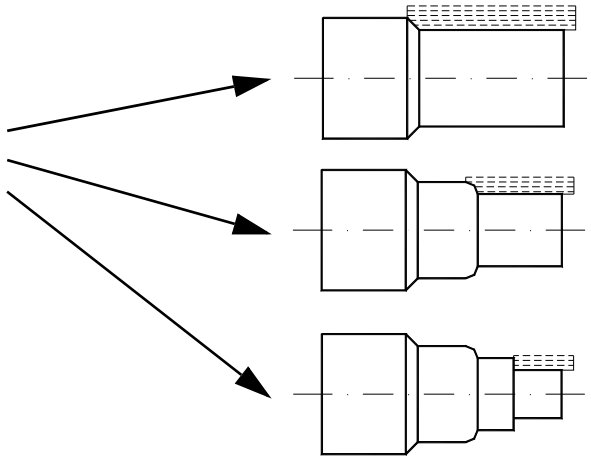
ENTER



Select the program with the focus.

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

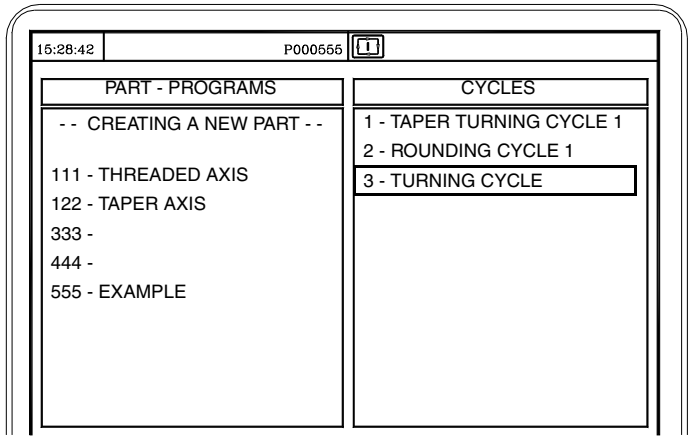




6.3 Modifying a part-program.

Replacing an operation.

Select an operation and press [RECALL].

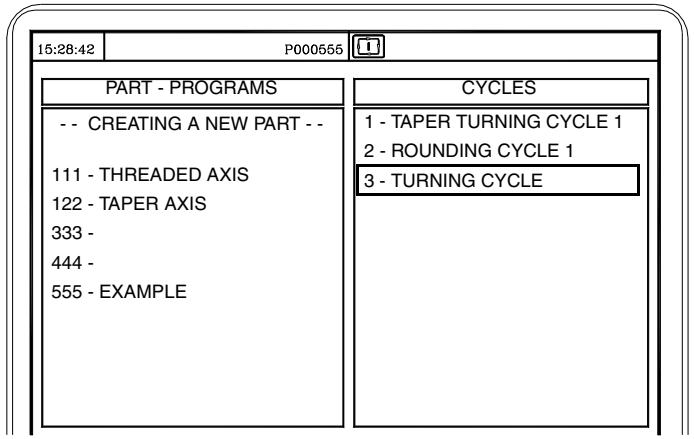


RECALL

The CNC shows the operation with all the data. Modify the parameters like in editing mode.

After modifying the data, press [P.PROG].

P.PROG



ENTER

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

ENTER

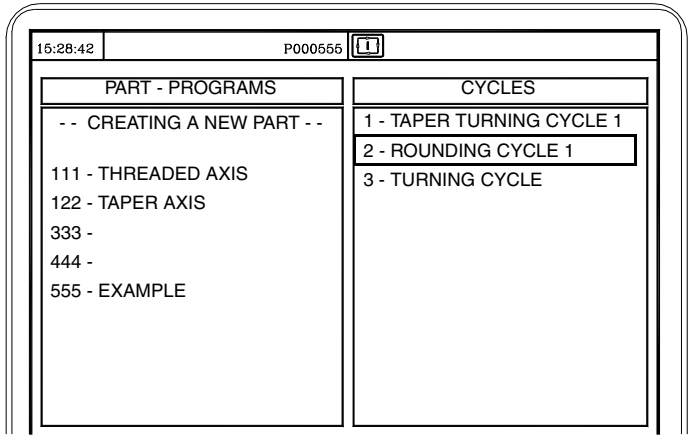
The new operation replaces the previous one.

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

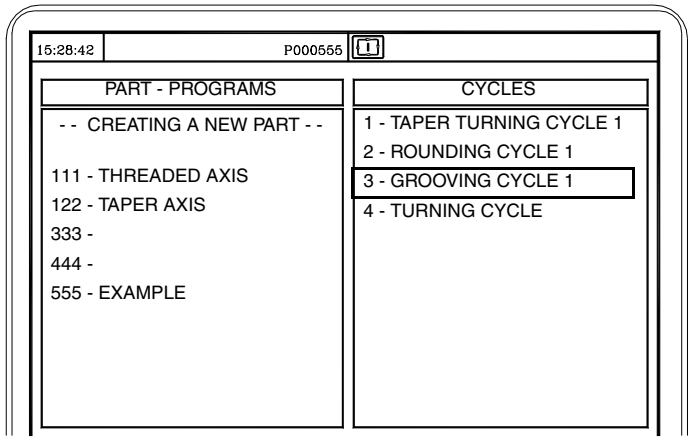
Inserting an operation.

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].

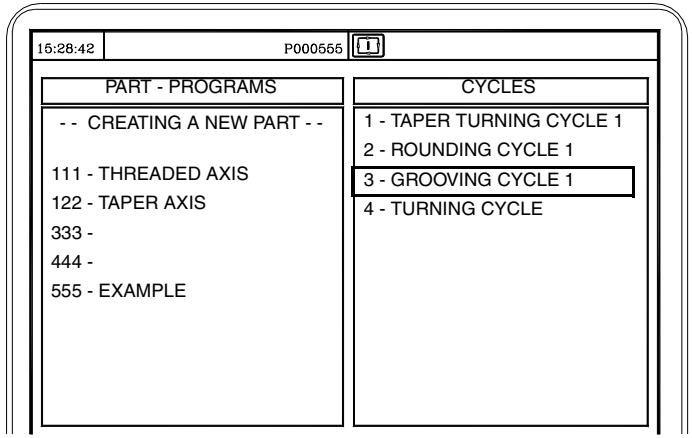


The new operation is inserted after the selected position.

Operations can also be deleted from a part-program.

To delete an operation.

Select the operation and press [CLEAR].



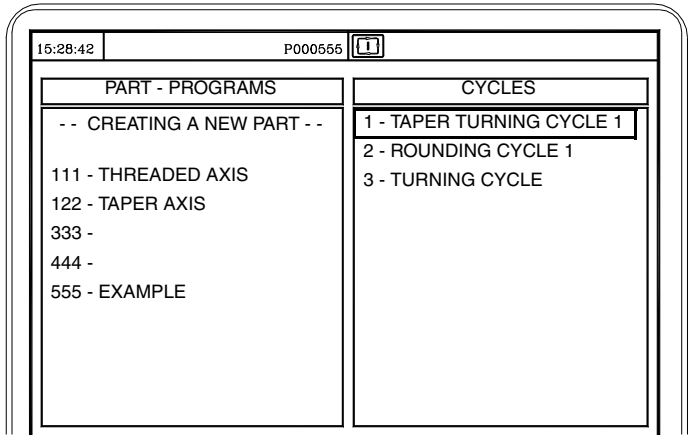
CLEAR

The CNC will request confirmation. Press [ENTER].

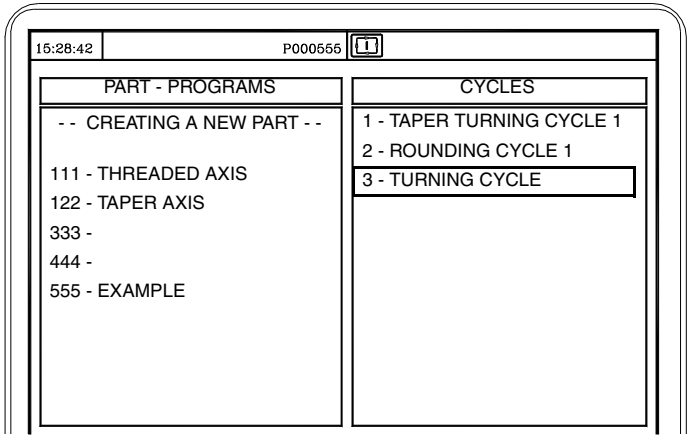
The position of an operation can also be changed.

Changing the position of an operation.

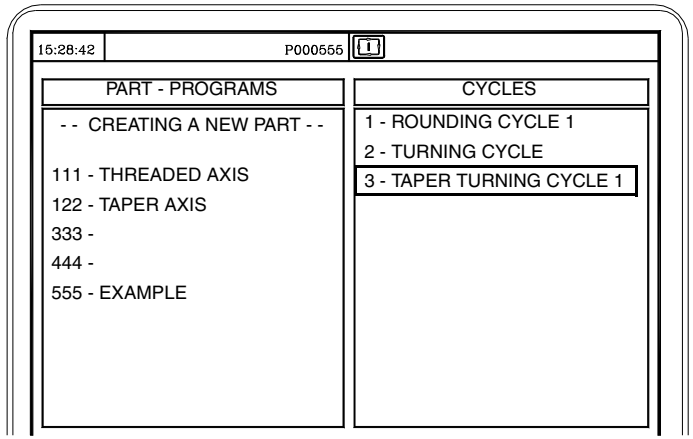
Select the operation to be relocated and press the two-color key.



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



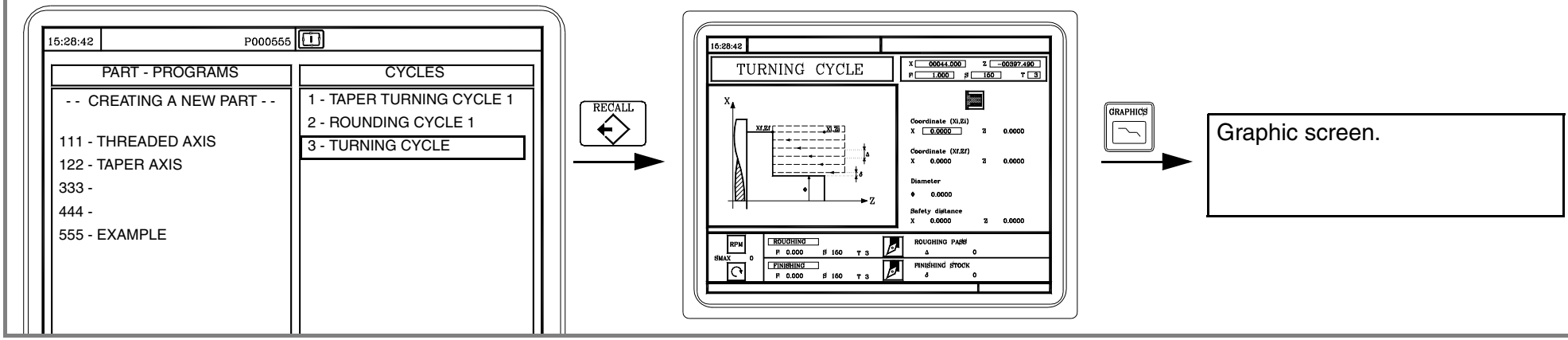
ENTER



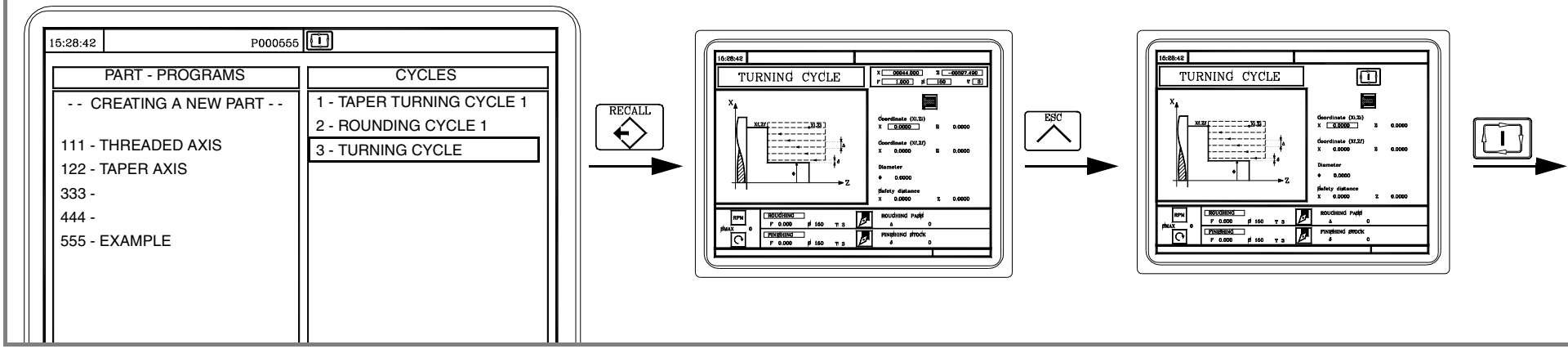
The new operation is inserted after the selected position.

6.4 Simulating/executing an operation.

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

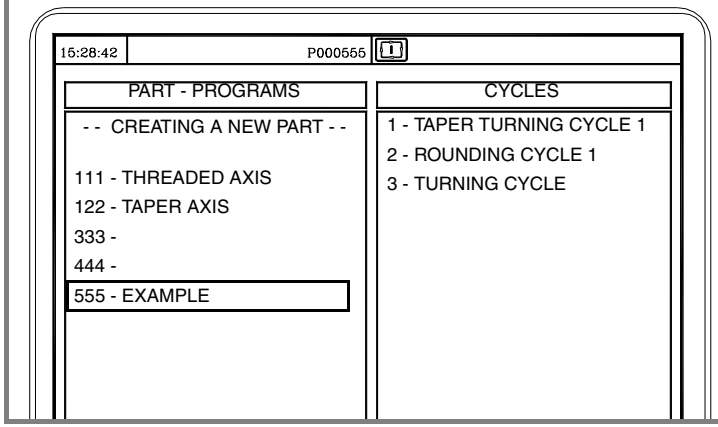


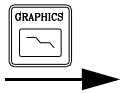
Select the operation to be simulated and press [ENTER].



6.5 Simulating/executing a part-program.

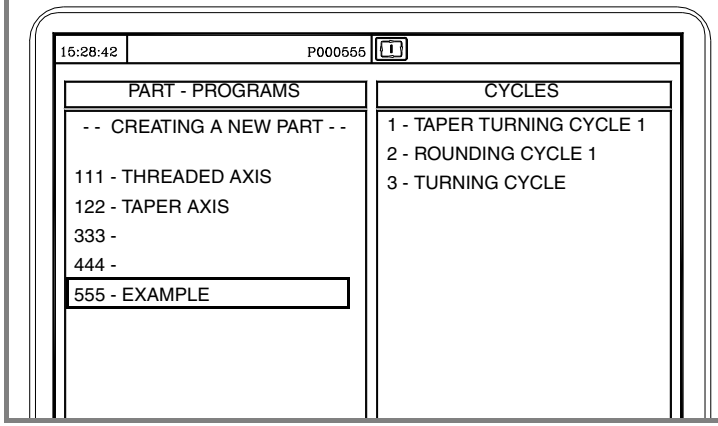
Select the part-program to be simulated and press [GRAPHICS].

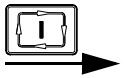




Graphic screen.

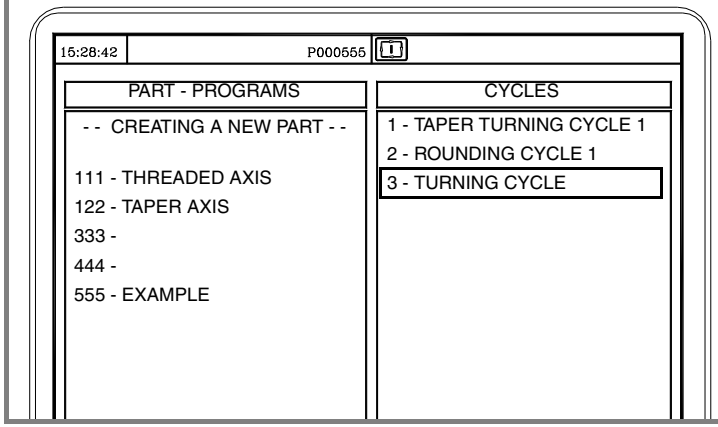
Select the part-program to be executed and press [START].

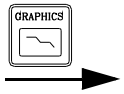




6.6 Simulating/executing a part-program starting with an operation.

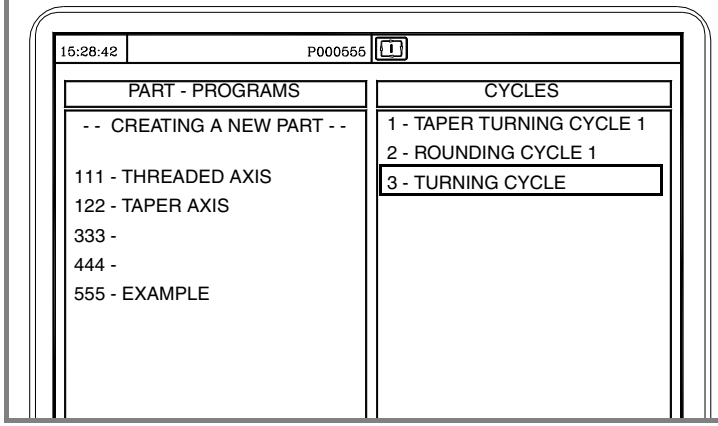
Select the first operation to start simulating and press [GRAPHICS].

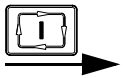




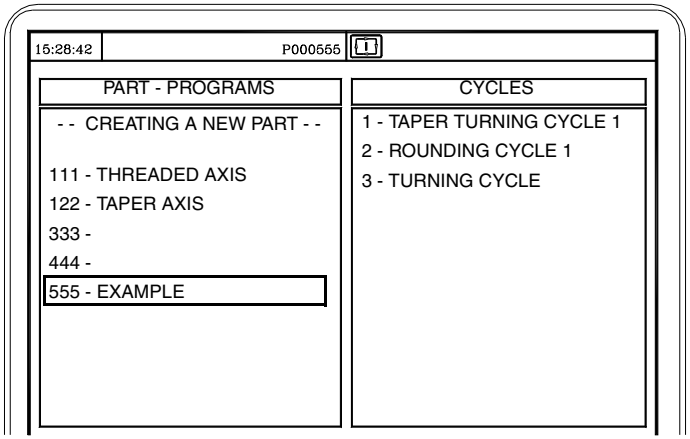
Graphic screen.

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

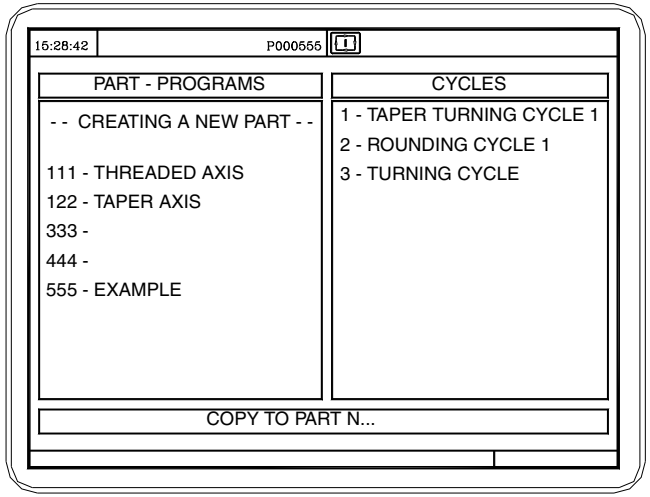




6.7 Copying a part-program into another one.

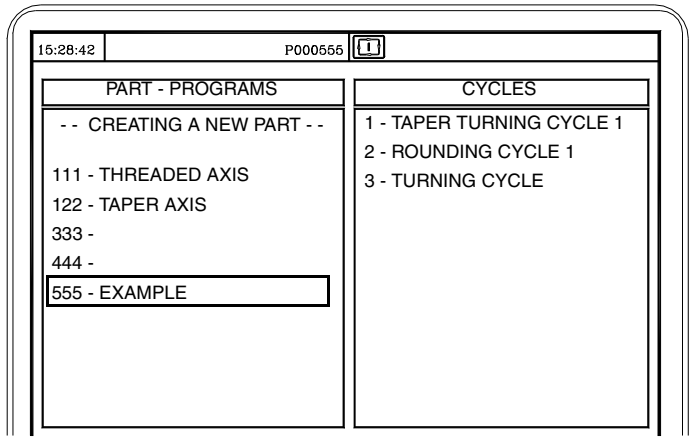


Select the program to be copied and press [P.PROG].

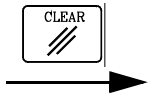


Enter the number of the new program.

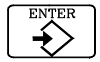
6.8 Deleting a new part program.

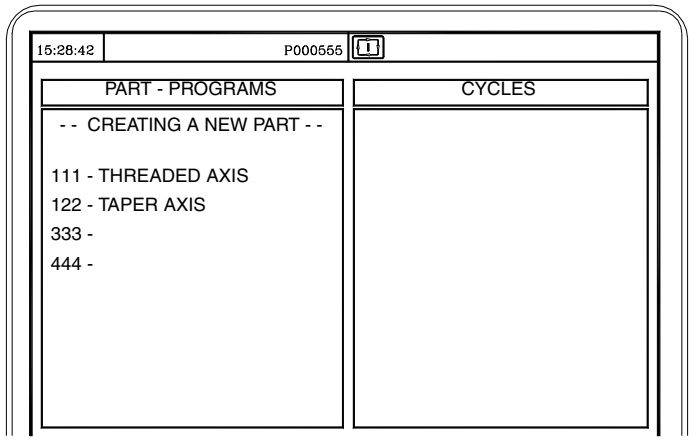


Select the program to be deleted and press [CLEAR].



The CNC will request confirmation. Press [ENTER].



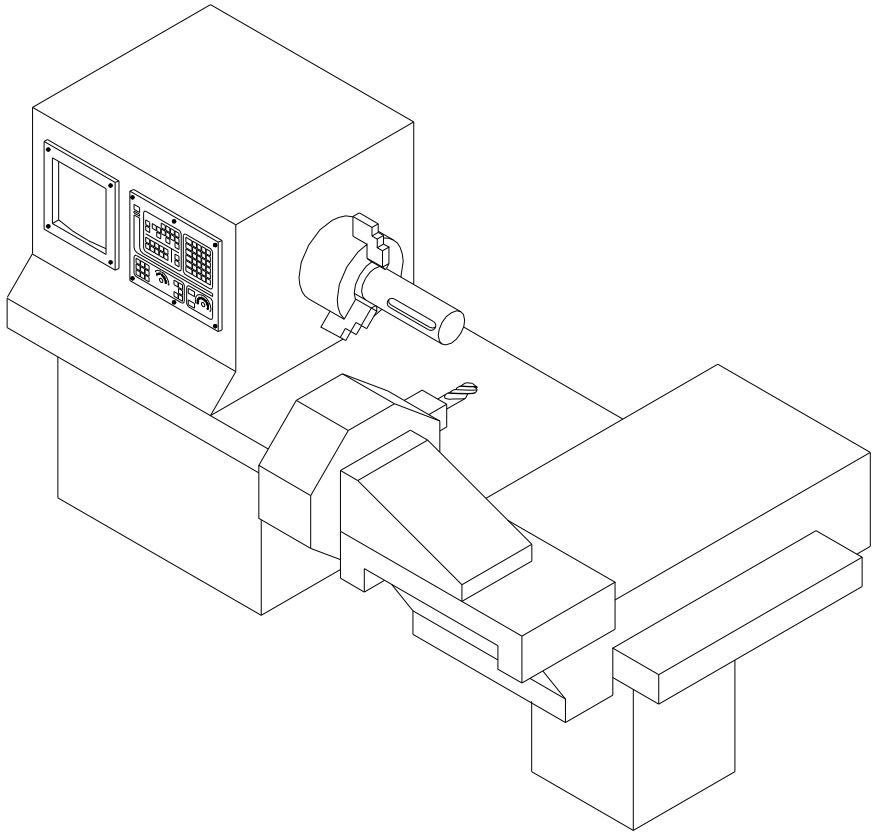


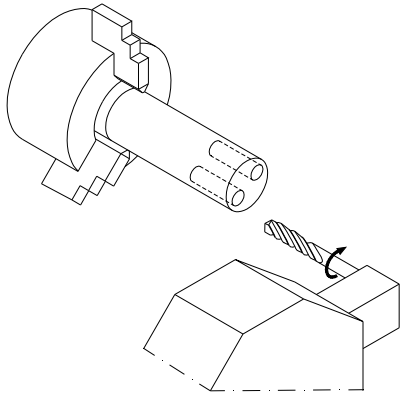
A OTHER LATHE TYPE MACHINING OPERATIONS.

- Orienting spindle and live tool.
- Multiple drilling.
- Multiple tapping.
- Slot milling.

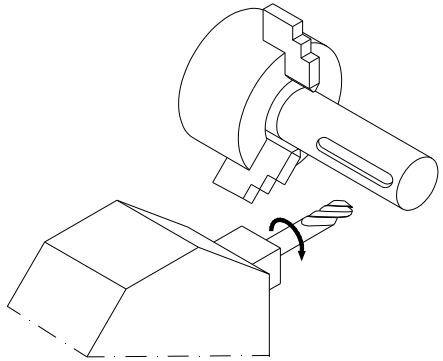
A.1 Introduction

For these machining operations, the machine must have an orienting spindle and a live tool. If the machine has these features, the CNC will show the options for "multiple drilling" and "slot milling" when accessing the drilling cycle.





Multiple drilling.



Slot milling.

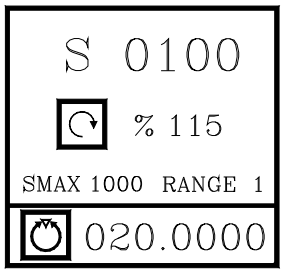
A.2 Orienting spindle and live tool.

With this feature, it is possible to orient the spindle at the desired angular position to make holes and slots on the cylindrical side of the part or on its face.

Orienting spindle.

This key accesses the spindle orientation mode. The spindle stops (if it was turning) and orients at the indicated angle. Every time this key is pressed, the spindle position increments in that angle.

To enter the angular increment value, press the [S] key three times.



Selected turning speed.

Turning direction and percentage applied.

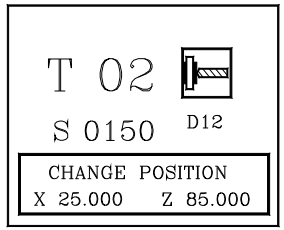
Maximum turning speed and active gear.

Angular increment of the spindle.

To exit the spindle orientation mode, press one of the keys for the spindle.

Live tool.

Being the live tool active, the CNC shows the following information.



Tool number.

Tool turning speed.

Position of the tool change point.

To set the speed of the live tool, first press [T] to select the window for tools. Then press:

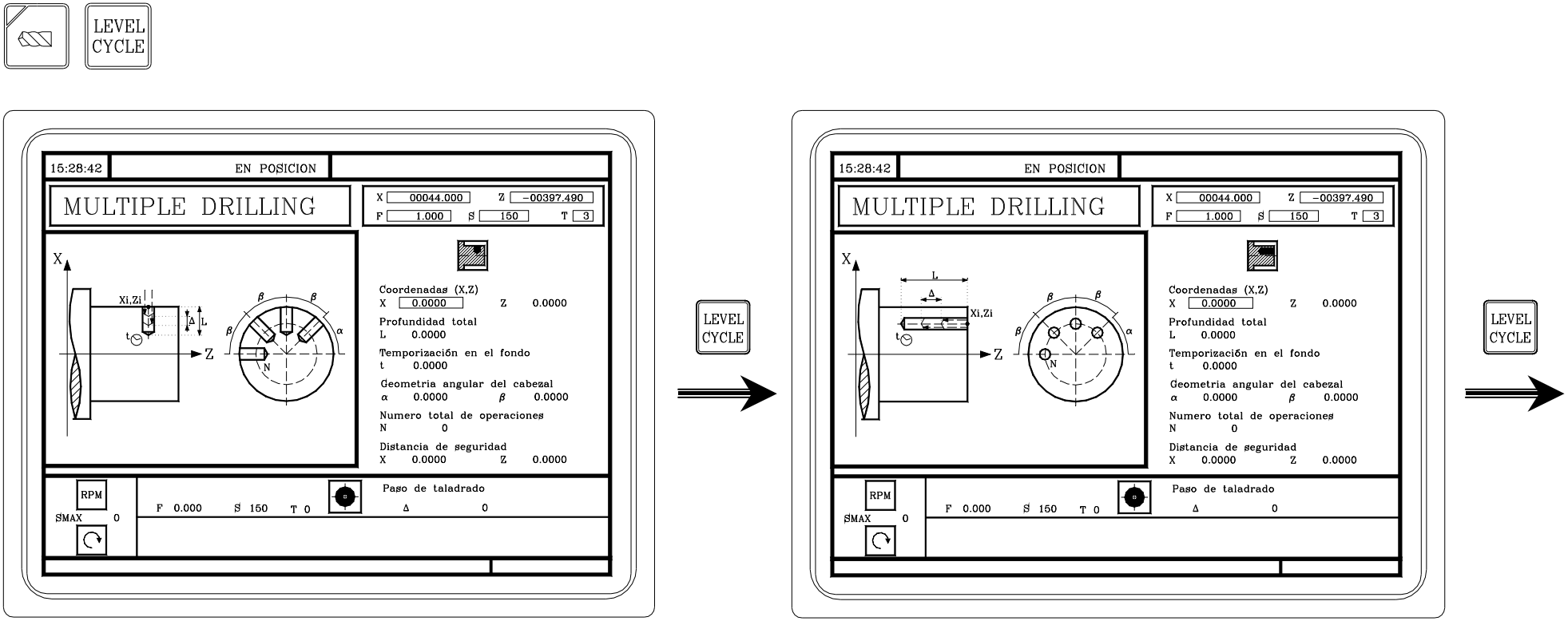
S + Turning speed. + ENTER

To monitor the live tool:

Starts the live tool.

Stops the live tool.

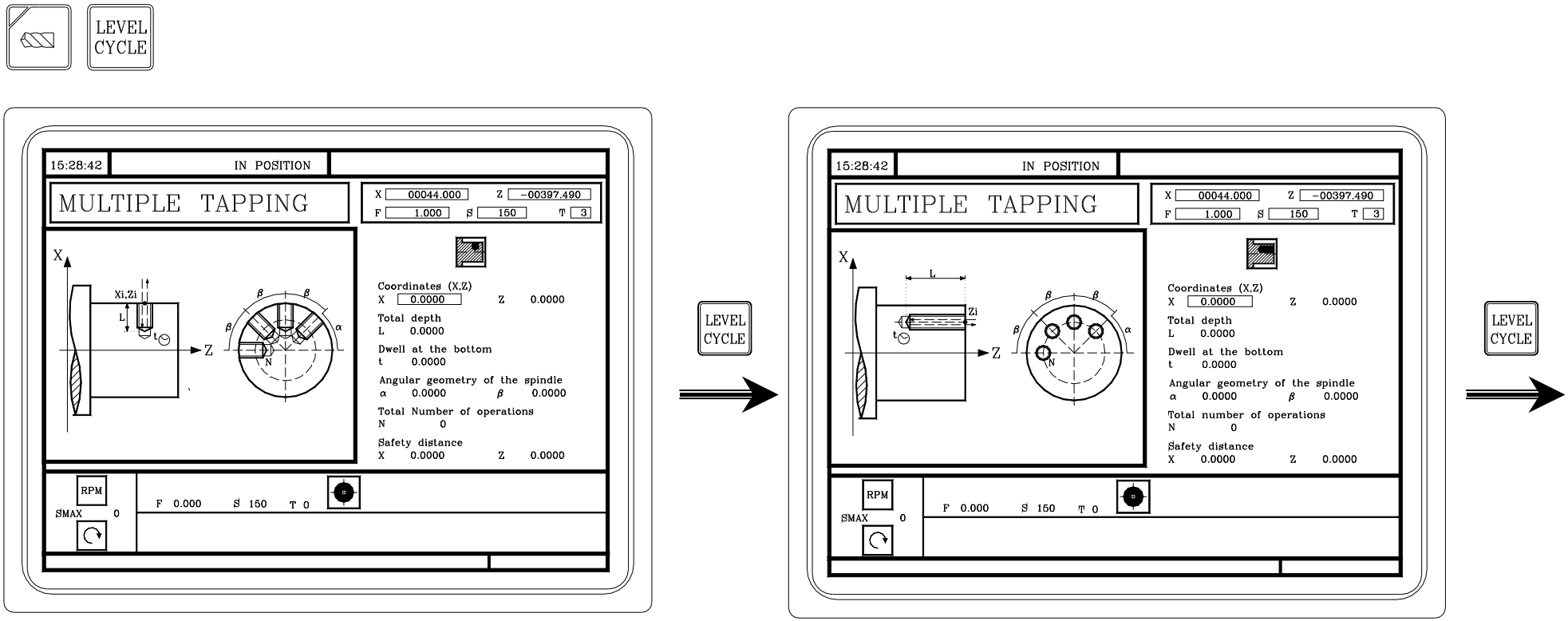
A.3 Multiple drilling.



Multiple drilling on the cylindrical side of the part.

Multiple drilling on the face of the part.

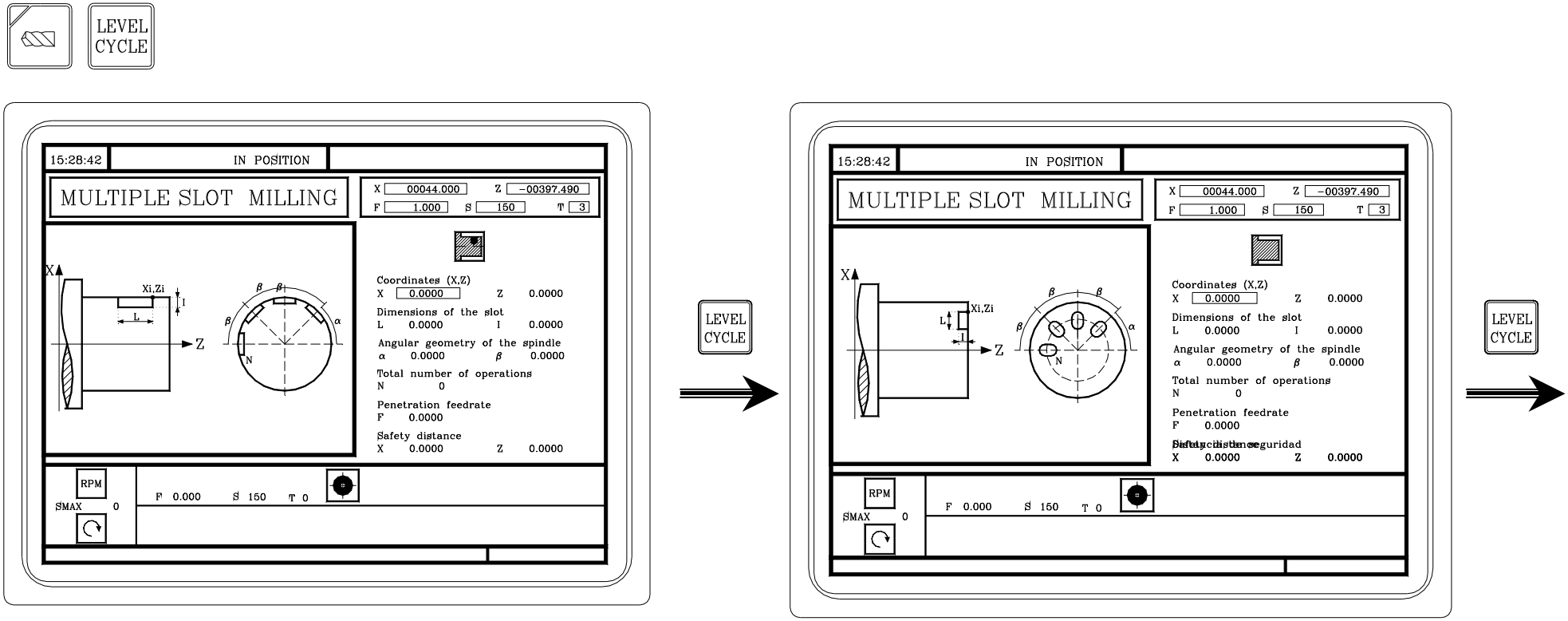
A.4 Multiple tapping.



Multiple tapping on the cylindrical side of the part.

Multiple tapping on the face of the part.

A.5 Slots.



Slots on the cylindrical side of the part.

Slots on the face of the part.