CNC 8055 MCO/TCO

Operating manual

REF. 1010

SOFT: V01.0X

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 GENERAL CONCEPTS

1.1 M/T and CO work mode 5
1.2 Keyboard 6
1.3 Keyboard customizing 9

CHAPTER 2 OPERATING IN JOG MODE

2.1 Introduction 12
2.1.1 Standard screen of the CO mode 12
2.1.2 Auxiliary screen of the CO mode 14
2.2 Axis control 16
2.2.1 Work units 16
2.2.2 Coordinate preset 16
2.2.3 Managing the axis feedrate (F) 16
2.3 Home search 17
2.4 Jog movement 18
2.4.1 Movement to the programmed coordinate 18
2.4.2 Incremental movement 18
2.4.3 Continuous movement 19
2.4.4 Movement with an electronic handwheel 20
2.5 Tool control 24
2.5.1 Single tool change point 24
2.5.2 Variable tool change point 25
2.5.3 Live tool 26
2.6 Spindle control 27
2.6.1 Spindle in rpm 28
2.6.2 Constant surface speed 29
2.7 ISO management 31

CHAPTER 3 WORKING WITH OPERATIONS OR CYCLES

3.1 Associated programs 35
3.2 Associated subroutines 35
3.3 OEM cycles 36
3.3.1 Define the screen 36
3.3.2 Configuration file 37
3.3.3 Associated subroutine 41
3.3.4 Error log file (P999500) 43
3.4 Access to OEM cycles from the PLC 44
3.5 Cycle data entry 45
3.6 Simulation and execution 46
3.7 Background cycle editing 46
3.8 Positioning cycle 47
3.8.1 Definition of data 48
3.8.2 Definition of spindle conditions 49
3.8.3 Definition of machining conditions 49

CHAPTER 4 SAVING PROGRAMS

4.1 List of saved programs 52
4.2 See the contents of a program 53
4.2.1 See one of the cycles in detail 53
4.3 Edit a new part-program 54
4.3.1 Saving an ISO block or a cycle 54
4.4 Delete a new part program 55
4.5 Copying a part-program into another one 55
4.6 Modify a part-program 56
4.6.1 Delete an operation 56
4.6.2 Move an operation to another position 56
4.6.3 Add or insert a new operation 57
4.6.4 Modify an existing operation 57

1.

FAGOR

·MCO/TCO·
Option

Soft: V01.0x

CHAPTER 5 EXECUTION AND SIMULATION

5.1 Simulating or executing an operation or cycle 60
5.2 Simulating or executing a part-program 61
5.2.1 Simulating or executing a portion of a part-program 61
5.3 Simulating or executing an operation that has been saved 61
5.4 Execution mode 62
5.4.1 Tool inspection 63
5.5 Graphic representation 64

1.

FAGOR

·MCO/TCO·
Option

Soft: V01.0x

GENERAL CONCEPTS

1

1.1 M/T and CO work mode

The CNC offers all the features of the M/T model plus those specific of the CO mode.

There are 2 work modes: M/T and CO work mode. This manual refers to the CO mode.

The M or T work mode described in the installation, Operation and Programming manuals will be used for:

- Setup.
- Editing the PLC program.
- Screen customizing.
- Canned cycle customizing.
- Eliminating certain CNC errors.
- Etc.

The CO mode lets the machine manufacturer:

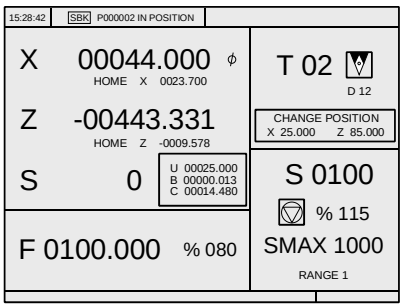
- Customize the keyboard.
- Change the CNC screens.
- Design the machining operations or canned cycles.
- Create screens for diagnosis, setup, CNC-PLC status information, etc.

This manual describes how to operate with the basic or standard Fagor features and it is up to the OEM to document all the changes and new cycles defined by him.

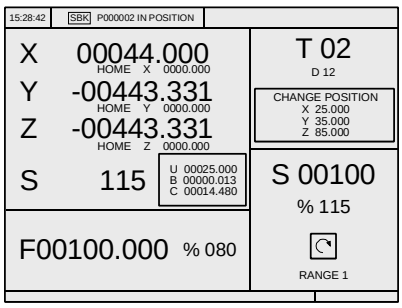
On CNC power-up and after the keystroke sequence [SHIFT] [RESET], the CNC displays the standard screen for the selected work mode.

With the keystroke sequence [SHIFT] [ESC] , it is possible to switch from CO mode to M/T mode and vice versa.

The standard screen of the CO mode depends on the model:



TCO model



MCO model

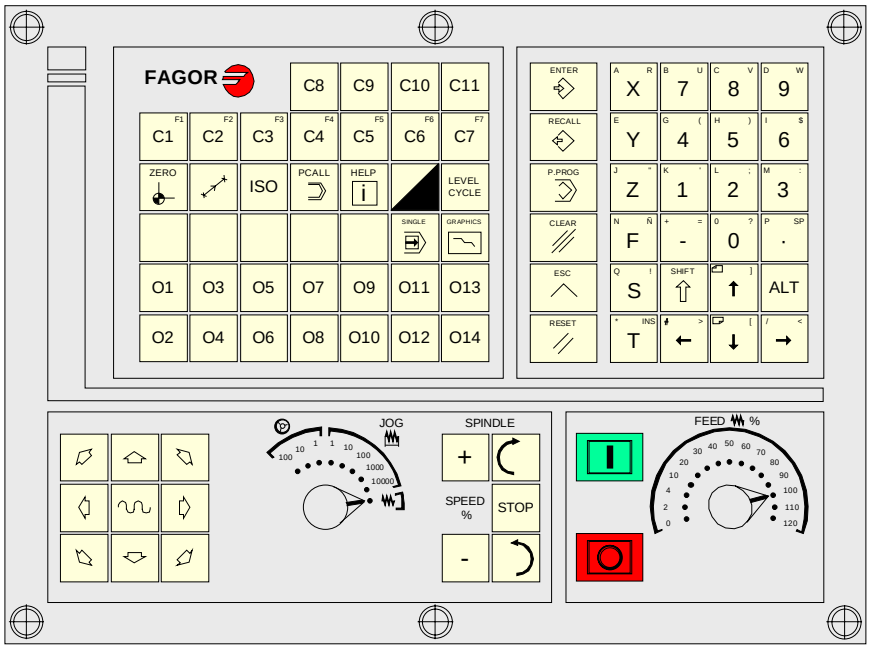
To customize (modify or replace) this screen, refer to the "Screen customizing" chapter of the installation manual.

FAGOR

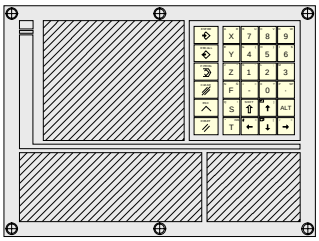
·MCO/TCO·
Option

Soft: V01.0x

1.2 Keyboard

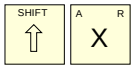


Alphanumeric keyboard and command keys

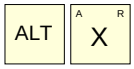




Select the X character.

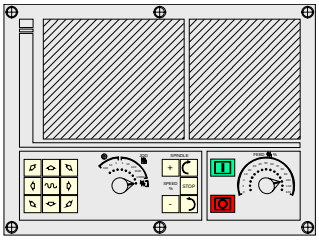


Select the A character.



Select the R character.

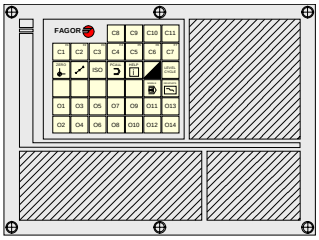
JOG keys



These keys may be used for:

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

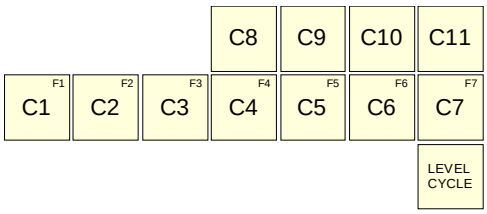
Specific keys of the CO model



These keys may be used for:

- Selecting and defining the machining operations.
- Govern the external devices.
- Select the graphic representation.
- Selecting the single block or automatic execution Mode.

Keys related to operations or canned cycles:



[C1] gives access to cycle 1, [C2] to cycle 2 and so on.

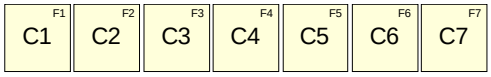
Each key may have up to 10 levels or variants that are selected with the [LEVEL CYCLE] key. For example:

C1 => Cycle 1, +Level => Cycle 21, +Level => Cycle 41, ...

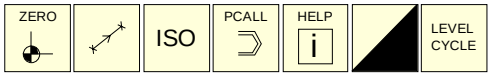
When pressing [C1] through [C11], the CNC recovers the last cycle (level) used in each group.

The OEM may set keys [C1] through [C11] by inserting printed strips behind the keyboard.

Keys that act like softkeys F1 through F7 in M/T mode:



Specific keys for the CO mode:



From left to right, they are used for:

- [ZERO]: Doing a home search.
- [POS.]: Executing the positioning cycle.
- [ISO]: ISO and MDI management.
- [PCALL]: To see the screens or cycles defined by the OEM.
- [HELP]: (For the M/T mode).
- [TWO-COLORED]: Selecting various states in particular cycles or screens.
- [LEVEL CYCLE]: Selecting various levels in each cycle.

Free keys:

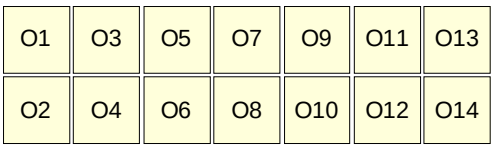


The OEM may set the free blank keys by inserting printed strips behind the keyboard. This chapter describes how to manage those keys.

The [SINGLE] key lets execute or simulate step by step.

The [GRAPHICS] key lets simulate the cycle or see the tool movement graphics duration execution.

Free keys, usually to control external devices:



The OEM may set them by inserting printed strips behind the keyboard.

This chapter describes how to manage those keys.

Command keys:



For the CNC to assume the edited value.



To extract saved data, cycles or blocks from memory.



To access program saving.

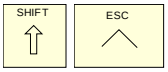


Depending on the data, it lets delete the last character inserted or the whole program.

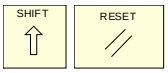


To quit the selected operation, cycle or mode.

Special keystroke sequences:



To toggle between CO and M/T modes.



To initialize the CNC. It is the same as turning the CNC off and back on.



It clears (blanks out) the CRT screen. Press any key to restore it.

FAGOR

·MCO/TCO·
Option

Soft: V01.0x

1.3 Keyboard customizing

The machine manufacturer must use the PLC program to activate and deactivate the external devices.

The CNC will inform the PLC about the status of each key. The corresponding register bit will be set to 1 when the key is pressed and 0 when it is not pressed.

The register bit for each key is:

O1	O3	O5	O7	O9	O11	O13
O2	O4	O6	O8	O10	O12	O14

B26 R561	B27 R561	B0 R562	B1 R562	B31 R560		
B28 R561	B29 R561	B2 R562	B3 R562	B7 R561	B6 R561	B5 R561
B30 R561	B31 R561	B4 R562	B5 R562	B15 R561	B14 R561	B13 R561

Keys [O1] through [O14] have a lamp. The status of the lamp of each key must be controlled by the machine manufacturer through the PLC program using the input variables TCLED* indicated in the figure.

TCLED1	TCLED3	TCLED5	TCLED7	TCLED9	TCLED11	TCLED13
O1	O3	O5	O7	O9	O11	O13
O2	O4	O6	O8	O10	O12	O14
TCLED2	TCLED4	TCLED6	TCLED8	TCLED10	TCLED12	TCLED14

Examples:

The [O1] key is used to control the coolant.

DFU B28R561 = CPL TCLED1
= CPL O33

A number of conditions must be met in order to activate the device that may be activated and deactivated with the [O2] key.

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

1.

GENERAL CONCEPTS
Keyboard customizing

FAGOR

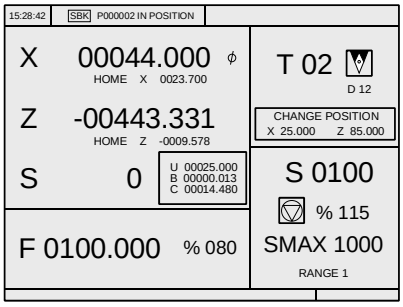
·MCO/TCO·
Option

Soft: V01.0x

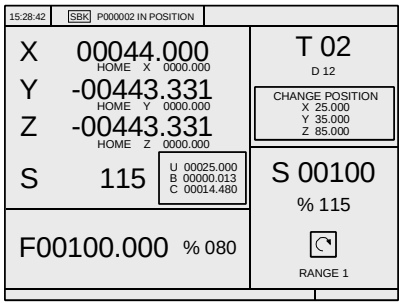
OPERATING IN JOG MODE

2

The standard screen of the CO mode depends on the model:



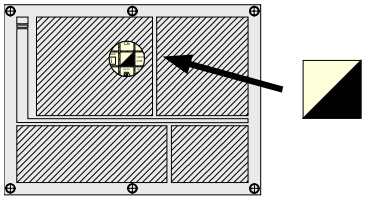
TCO model

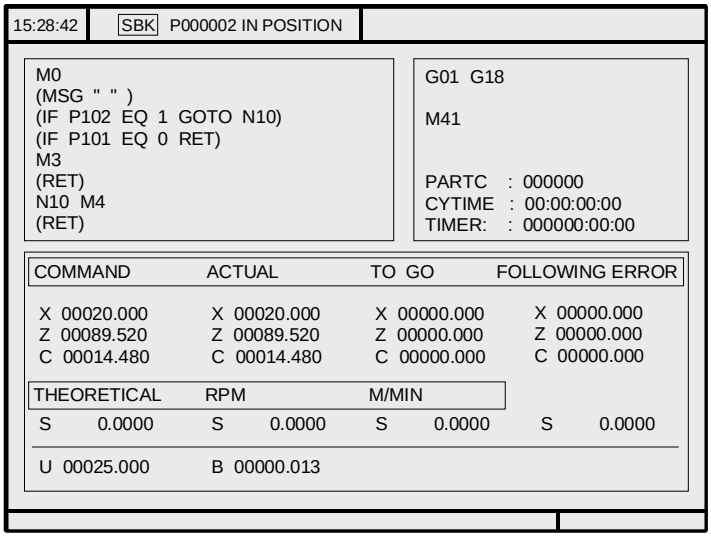


MCO model

The way to operate with either model is very similar. If any of the features described in this chapter is not common to both models, it will clearly indicate which model it corresponds to.

When pressing the [TWO-COLORED] key, the CNC shows the auxiliary screen of the CO mode.





FAGOR

·MCO/TCO·
Option

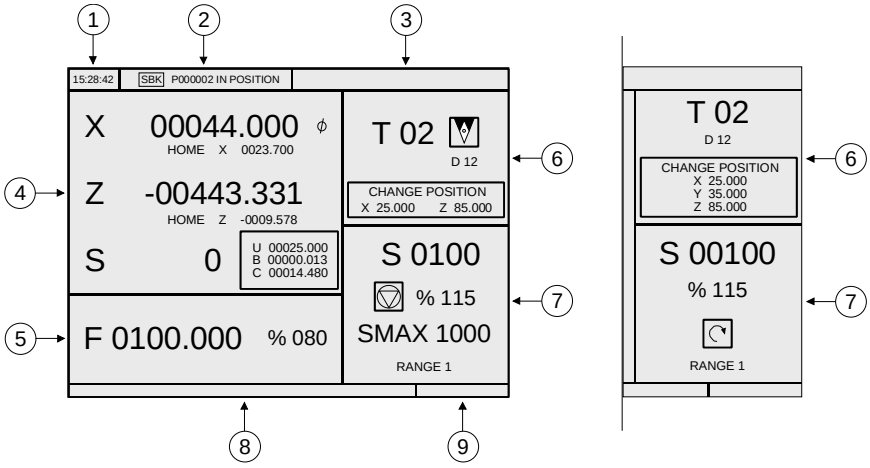
Soft: V01.0x

2.1 Introduction

2.1.1 Standard screen of the CO mode

The whole standard screen of the CO mode may be customized.

The one provided by Fagor contains the following information:



1. Clock.
2. This window may show the following data:

SBK	when “single block” execution mode is selected.
DNC	when the DNC mode is active.
P.....	number of the program currently selected.
Message	«In position» - «Execution» - «Interrupted» - «RESET».
PLC messages.	

3. This window shows the CNC messages.
4. This whole window may be customized.

   The one provided by Fagor may show the following data:

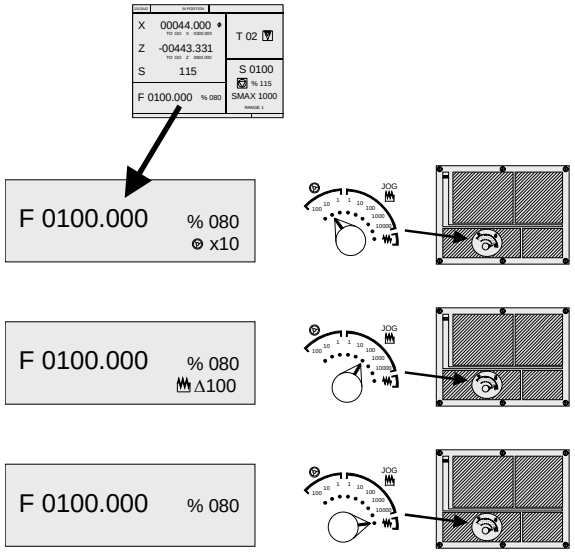
   - The coordinates of the axes. The Ø symbol indicates that the axis is working in diameter.
   - If text 33 of program 999997 is defined, the CNC shows, in small characters, the coordinates of the axes referred to machine reference zero.
   - The real spindle rpm (S).
   - In a frame, the coordinates of the auxiliary axes that have been defined.

5. This whole window may be customized.

The one provided by Fagor shows the axis feedrate «F» currently selected and the % of F being applied.

When feed-hold is active, the color of the feedrate value changes.

Also, dependingn on the position of the left switch, it can show the following information.



6. This whole window may be customized.

The one provided by Fagor shows everything about the tool:

- In large characters, the number of the currently selected tool "T".
- The graphic representation for the location code (shape) associated with the tool (TCO model).
- The "D" offset number associated with the tool. If the tool number and the offset number are the same, the CNC will not show the “D” value.
- The position values (coordinates) of the tool change point referred to machine reference zero. The CNC does not show this window when text 47 of program 999997 has not been defined.

7. This whole window may be customized.

The one provided by Fagor shows everything about the spindle:

- The selected theoretical turning speed. «S» value when working in rpm and «CSS» value when working at constant surface speed in the TCO model.
- The spindle status. It is represented with an icon and may be turning clockwise, counterclockwise or stopped.
- The % of spindle speed being applied.
- The maximum spindle rpm (TCO model).
- The active spindle speed gear (range). The CNC does not show this data when text 28 of program 999997 has not been defined.

8. When accessing a work cycle, this window shows the help text associated with the selected icon.

That help text must be defined in program P999994 and edited in the desired language.

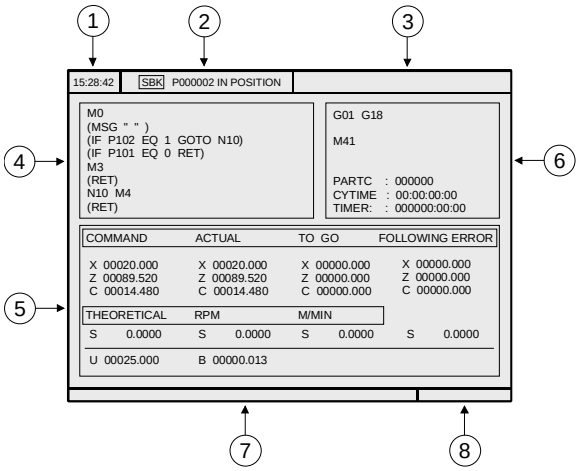
The format and considerations of program P999994 are described in detail in the chapter on "General concepts".

9. Reserved.

2.1.2 Auxiliary screen of the CO mode

The whole auxiliary screen of the CO mode may be customized.

The one provided by Fagor contains the following information:



1. Clock.
2. This window may show the following data:

   SBK when “single block” execution mode is selected.

   DNC when the DNC mode is active.

   P..... number of the program currently selected.

   Message «In position» - «Execution» - «Interrupted» - «RESET».

   PLC messages.
3. This window shows the CNC messages.
4. This window shows the lines of the program currently selected.
5. The axes have the following fields:

   COMMAND It indicates the programmed coordinate or position that the axis must reach.

   ACTUAL It indicates the actual (current) position of the axis.

   TO GO It indicates the distance which is left to run to the programmed coordinate.

   FOLLWOING ERRORDifference between the theoretical value and the real value of the position.

   The spindle (S) has the following fields:

   THEORETICALProgrammed theoretical S speed.

   RPM Speed in rpm.

   M/MIN Speed in meters per minute.

   FOLLOWING ERRORWhen working with spindle orientation (M19), it indicates the difference between the theoretical and the real speeds.

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

6. This window shows the status of the «G, F, T, D, M» functions and the value of the variables.

   PARTC — It indicates the number of consecutive parts executed with the same part-program.

   Every time a new program is selected, this variable is reset to "0".

   CYTIME — Indicates the time elapsed while executing the part. It is given in "hours : minutes : seconds : hundredths of a second".

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

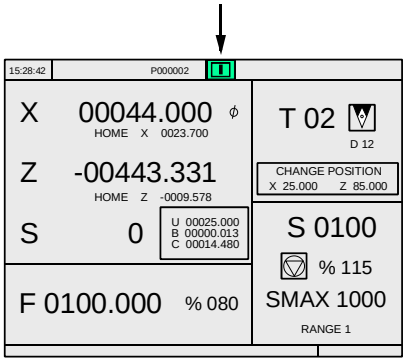
   TIMER — Indicates the count of the timer enabled by PLC. It is given in "hours : minutes : seconds".

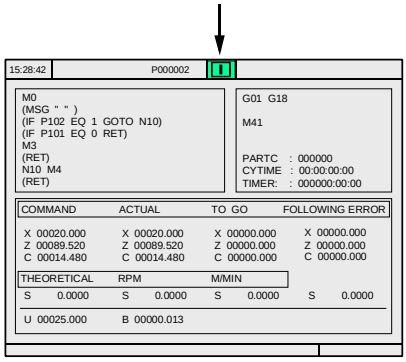
7. Reserved.
8. Reserved.

Selecting a program for simulation or execution



Symbol that appears in the middle of the top window when a part-program has been selected for simulation or execution.





When the top center window shows the part-program selected next to the green [START] symbol, the CNC acts as follows:

- If [START] is pressed, the CNC executes the part-program that is selected.
- If [CLEAR] is pressed, the CNC de-selects the part-program and removes it from the top center window.

2.2 Axis control

2.2.1 Work units

When accessing the CO mode, the CNC assumes the work units “mm or inches”, "radius or diameter", “mm/min. or mm/rev”, etc. selected by machine parameter.

To modify those values, access the M/T mode and change the corresponding machine parameter.

2.2.2 Coordinate preset

The coordinates must be preset on one axis at a time proceeding as follows:

[X] value [ENTER] [Z] value [ENTER]

The CNC requests confirmation of the command.

Pressing [ESC] quits the preset.

2.2.3 Managing the axis feedrate (F)

To set a particular axis feedrate value, proceed as follows:

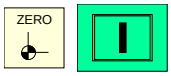
[F] value [START].

Pressing [ESC] quits the selection.

2.3 Home search

Home search may be done in 2 ways:

Homing all the axes

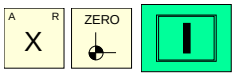


It executes the home search subroutine defined by the OEM in general machine parameters P34 (REFPSUB).

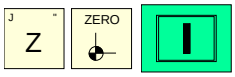
The CNC keeps the part zero or zero offset that is active.

If the home search subroutine has not been defined, the CNC will display the corresponding error message.

Homing a single axis



It homes the X axis.



It homes the Z axis.

The CNC will not maintain the part zero or zero offset active at the time and assumes the machine reference zero as the new part zero.

2.4 Jog movement

The axes of the machine can move as follows:

- Movement to the programmed coordinate.
- Incremental move.
- Continuous movement.
- Movement with an electronic handwheel.

2.4.1 Movement to the programmed coordinate

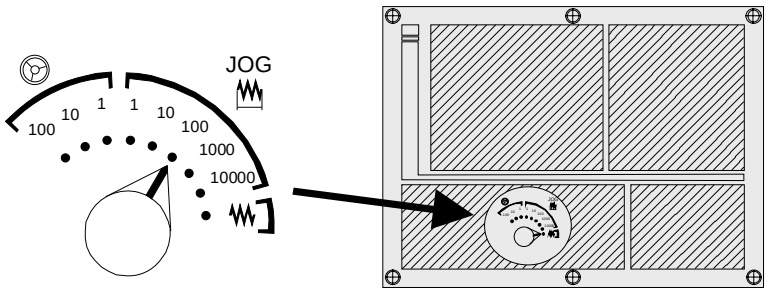
The axes move one by one. With the feedrate «F» and % currently selected.

[X] Coordinate [START] [Z] Coordinate [START]

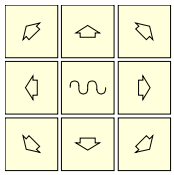
2.4.2 Incremental movement

JOG

Turn the JOG switch to one of the incremental jog positions.



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

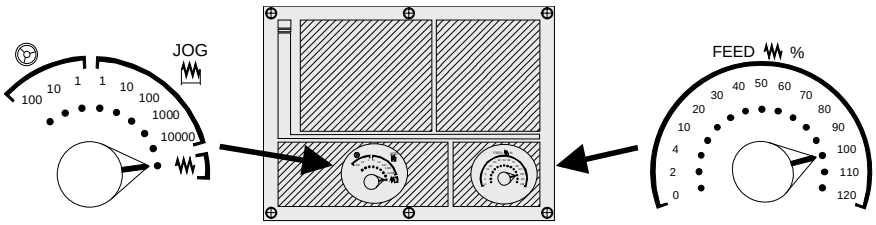


Every time a key is pressed, the corresponding axis moves the amount set by the switch. This movement is made at the selected feedrate «F».

Switch position	Distance
1	0.001 mm or 0.0001 inches
10	0.010 mm or 0.0010 inches
100	0.100 mm or 0.0100 inches
1000	1.000 mm or 0.1000 inches
10000	10.000 mm or 1.0000 inches

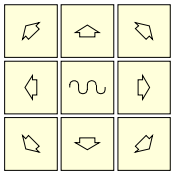
2.4.3 Continuous movement

Place the left switch in the continuous-jog position and select at the right switch the percentage (0% to 120%) of the feedrate to be applied.



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

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



If while jogging an axis, this key is pressed (rapid), the axis will move at the maximum feedrate possible, set by axis machine parameter "G00FEED". This feedrate will be applied while the key is kept pressed and the previous feedrate will be restored when that key is released.

Depending on the value assigned to general logic input "LATCHM", the movement will be carried out in one of the following ways:

- The axis will move only while the corresponding jog key is kept pressed.
- The axis will start moving from the moment the JOG key is pressed until the same is pressed again, or another JOG key is pressed. In this case, the movement will be transferred to that indicated by the new key.

The following cases are possible when working with "F" in mm/rev:

A. The spindle is turning clockwise or counterclockwise.

The CNC moves the axes at the programmed F.

B. The spindle is stopped, but a spindle speed S has been selected.

Feature available at the TCO model when working in G95.

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

For example if «F 2.000» and «S 500»:

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

The axis moves at a feedrate of 1000 mm/min.

C. The spindle is stopped and no spindle speed S has been selected.

Feature available at the TCO model when working in G95.

The machine does not move, only with rapid movements. When pressing the rapid key and the corresponding axis key.

2.4.4 Movement with an electronic handwheel

The various handwheel configurations are:

General handwheel: It may be used to jog any axis one by one.

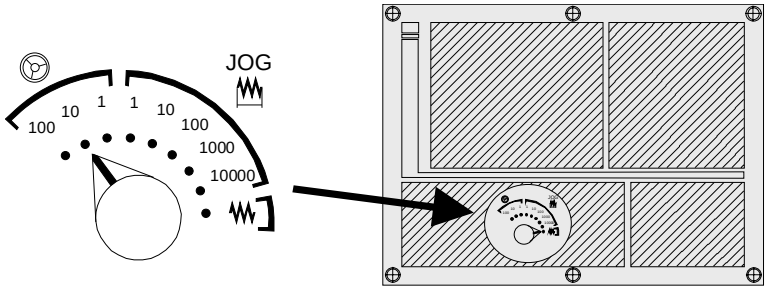
Select the axis and turn the handwheel to move it.

Individual handwheel: It replaces the mechanical handwheels.

One handwheel per axis may be used (up to 3).

It only moves the axis it is associated with.

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



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

For example, if the manufacturer has set a distance of 0.100 mm or 0.0100 inches per handwheel turn, thus:

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

When jogging with individual handwheels, it may happen that depending on the turning speed and the selector switch position, the CNC be demanded a faster feedrate than the maximum allowed. The CNC will move the axis the indicated distance but at the maximum feedrate allowed.

There are 3 operating modes with handwheels:

Standard handwheel:

With the general handwheel, select the axis to be moved and turn the handwheel.

With individual handwheels, turn the handwheel associated with the axis to be moved.

"Path handwheel" mode:

For chamfering and rounding corners.

2 axes are moved along a selected path (chamfer or rounding) by moving a single handwheel.

This feature must be managed from the PLC.

The general handwheel is assumed as the "path handwheel" or the individual handwheel associated with the X axis (MCO model) or Z (TCO model).

"Feed handwheel" mode:

To control the feedrate of the machine.

This feature must be managed from the PLC.

Standard handwheel

General handwheel

1. Select the axis to be jogged.

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

   When using a FAGOR handwheel with an axis selector button, the axis may be selected as follows:

   Push the button on the back of the handwheel. The CNC select the first axis and it highlights it.

   When pressing the button again, the CNC selects the next axis and so on in a rotary fashion.

   To deselect the axis, hold the button pressed for more than 2 seconds.

2. Jog the axis.

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

Individual handwheels

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

Simultaneous handwheels

The machine may have a general handwheel and up to 3 individual handwheels associated with each axis.

The individual handwheels have priority over the general handwheel. So, if an individual handwheel is moving, the general handwheel will be ignored.

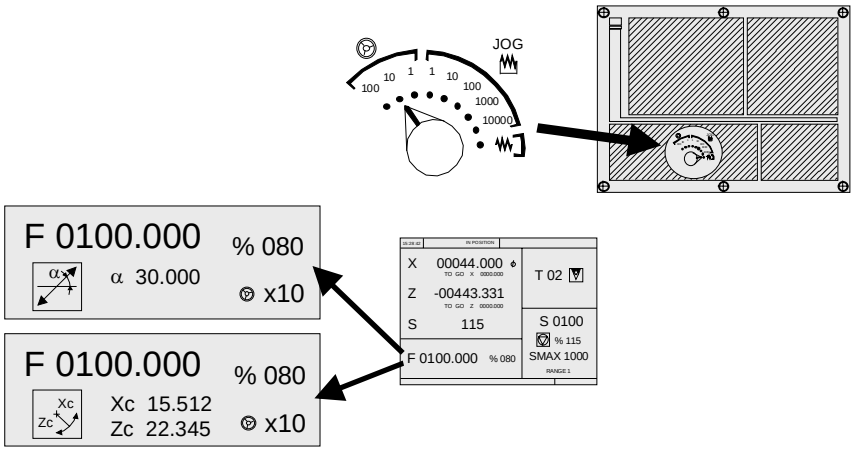
Path handwheel

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

The CNC assumes the general handwheel as the "path handwheel" or, when missing, the individual handwheel associated with the X axis (MCO model) or Z (TCO model).

It is up to the OEM to manage this feature.

While in handwheel mode and having selected path-handwheel from the PLC (MASTRHND=1), the CNC shows the following information:



The type of movement is set from the PLC: linear movement if HNLINARC = 0.

Arc movement if HNLINARC = 1.

For a linear movement (top figure), the path angle must be defined and for an arc (bottom figure), the center coordinates must be indicated. The center coordinates are defined in radius.

To define these variables, press the [F] key and then one of these keys: [⬅] [➡] [⬆] [⬇].

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

DFU B29 R561 = CPL MASTRHND

[O2] Activates and cancels the "Path Handwheel" mode.

DFU B31 R561 = CPL HNLINARC

[O3] selects the type of movement, straight section or arc section.

Simultaneous handwheels

When selecting the Path Handwheel mode, the CNC behaves as follows:

- If there is a General Handwheel, it will be the one working in Path handwheel mode. The individual handwheels, if any, will remain associated with the corresponding axes.
- If there is no General Handwheel, one of the individual handwheel starts working in Path handwheel mode. The one associated with the X axis at the MCO model or the one associated with the Z axis at the TCO if model.

Feed handwheel mode

Usually, when making a part for the first time, the machine feedrate is controlled by means of the feedrate override switch.

From this version on, it is also possible to use the machine handwheels to control that feedrate. This way, the machining feedrate will depend on how fast the handwheel is turned.

To do this, proceed as follows:

Inhibit all the feedrate override switch positions from the PLC.

Detect how far the handwheel is turned (reading of pulses received).

Set the corresponding feedrate override from the PLC depending on the pulses received from the handwheel.

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

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

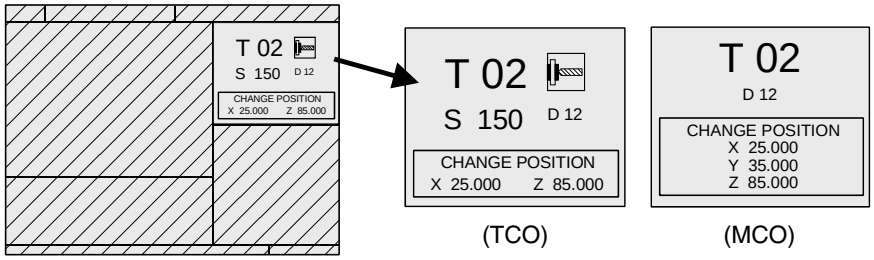
Example: The machine has a button to activate and deactivate this feature (feed handwheel) and the feedrate control is carried out with the second handwheel.

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

PRG
DFU I71 = CPL M1000                     Every time the button is pressed, mark
                                        M1000 is inverted.
M1000 = MSG1                            If the feature is active, a message is
                                        displayed.
NOT M1000                               If the feature is not active
= AND KEYDIS4 $FF800000 KEYDIS4         Inhibits all the feedrate override switch
                                        positions from the PLC.
= JMP L101                              and resumes program execution.
                                        If the feature is active
DFU M2009                               and there is an up flank at the clock mark
                                        M2009
= CNCRD(HANPS,R100,M1)                  it reads the handwheel pulses in R100
= SBS R101 R100 R102                    calculates the pulses received since the
                                        previous reading.
= MOV R100 R101                         updates R101 for the next reading
= MLS R102 3 R103                       calculates in R103 the proper feedrate
                                        override %
= OR KEYDIS4 $7FFFFF KEYDIS4            inhibits all the feedrate override switch
                                        positions.
CPS R103 LT 0 = SBS 0 R103 R103         ignores the handwheel turning direction.
CPS R103 GT 120 = MOV 120 R103          limits the maximum feedrate override to
                                        120%.
DFU M2009                               With the up flank at the clock mark
                                        M2009
= CNCWR(R103,PLCFRO,M1)                 sets the calculated feedrate override
                                        value (PLCFRO=R103).
L101
END
```

2.5 Tool control

The standard screen of the CO mode offers the following tool data.



(TCO) (MCO)

This whole window may be customized.

Depending on the model, the one provided by Fagor can show the following information:

- In large characters, the tool "T" number currently selected and a graphic representation of its tip.
- The "D" offset number associated with the tool.
- The "S" rpm that are selected for the live tool.

  This value is only shown when a live tool has been selected.
- The position values of the tool change point.
- The CNC does not show this window when text 47 of program 999997 has not been defined.

To select another tool, press: [T] Tool number [START].

The CNC will manage the tool change and update its graphic representation.

Pressing [ESC] quits the selection.

Another offset may be assigned to the tool temporarily without modifying the one associated with it.

To access the “D” field, press [T] and [⬇].

Key in the number of the desired tool offset and press [START].

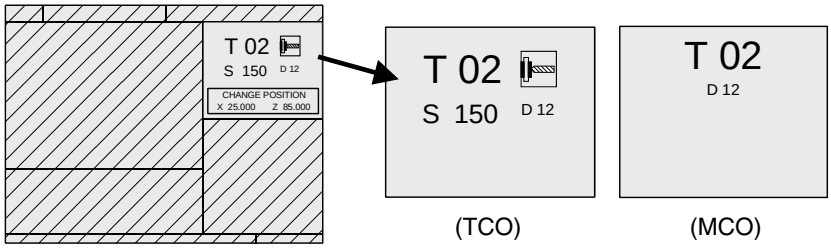
The CNC temporarily assumes the new offset for the current tool. The internal table is not modified, the tool’s associated offset is still the one assigned to it when it was calibrated.

2.5.1 Single tool change point

When the tool change always takes place at the same point, proceed as follows:

Set the text 47 of program 999997 as comment so the CNC does not show the window for the tool change point.

That is ;;47 $CHANGE POSITION.



(TCO) (MCO)

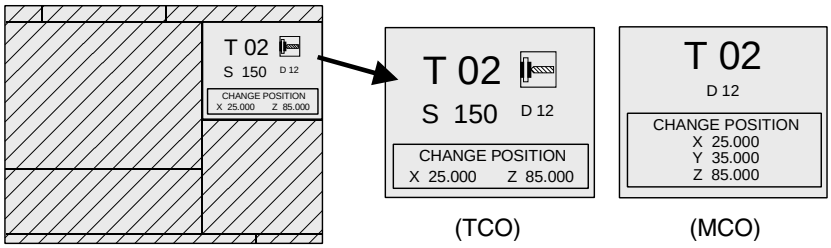
2.5.2 Variable tool change point

If the manufacturer so wishes, he can let the user define the tool change point every time. Obviously, this feature depends on the type of machine and type of tool changer.

This feature may be used to change the tool next to the part, thus avoiding movements to a tool change point located far away from it.

To do this:

Define the text 47 of program 999997 so the CNC requests the X and Z coordinates of the tool change point. For example: 47 $CHANGE POSITION.



These coordinates must be referred to machine zero point, so the zero offsets do not affect the tool change point.

Remember that the CNC can show, next to the X, Z coordinates and in small characters, the coordinates of the axes referred to machine reference zero (text 33 of program 999997).

Since the operator can change the tool change point at any time, the subroutine associated with the tools must consider those values.

Arithmetic parameters P290, P291 and P292 contain the values set by the operator as tool change position.

	MCO model	TCO model
Change position in X	P290	P290
Change position in Y	P291	
Change position in Z	P292	P291

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

Press the [T] key to select the «T» field. Then press the [X] or [Z] key of the desired axis or the [⬅] [➡] [⬆] [⬇] keys.

After placing the cursor on the coordinates of the axis to be defined, you can:

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

B. Assign the current machine position.

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

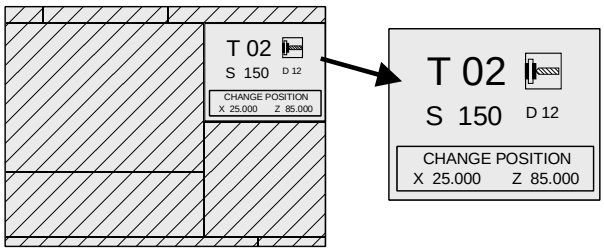
Press the [RECALL] key. The CNC assigns that coordinate to the selected field.

Press the [ENTER] key.

2.5.3 Live tool

Feature available only at the TCO model.

When a live tool has been selected, the standard screen of the CO mode offers the following information:



Proceed as follows to select the "S" rpm of the live tool:

1. Press the [T] key to select the «T» field.
2. Press the [S] key or the [⬇] key to select the "S" rpm of the live tool.
3. Entering the value manually. Key in the desired value and press [ENTER].

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

Set one of general parameters P0 to P9 with a value of 13.

The location code (shape) of the live tool must be 10, 20 or 30.

The PLC must manage the keys for the live tool.

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

The keys for the live tool are:

O1 Live tool counterclockwise.

O2 Stops the live tool.

O3 Live tool clockwise.

( ) = CNCRD (TOOL, R101, M1)

Assigns the number of the active tool to register R101.

= CNCRD (TOF R101, R102, M1)

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

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

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

CUSTOM AND (DFU B30R561 OR DFD M2) = CNCEX1 (M45 S0, M1)

If while the CO mode is selected (CUSTOM=1) ...
... the O2 "Stop live tool" key is pressed (DFU B30R561) or the live tool is de-selected (DFD M2)...
... the PLC "tells" the CNC to execute block M45 S0 (stops the live tool).

CUSTOM AND M2 AND DFU B29R561 = CNCRD (LIVRPM, R117, M1) = CNCWR (R117, GUP100, M1) = CNCEX1 (M45 SP100, M1)

If in CO mode (CUSTOM=1), a live tool is selected (M2) and the O3 key "live tool clockwise" is pressed (DFU B29R561) ...
... the PLC reads in R117 the rpm currently selected for the live tool (LIVRPM) and assigns them to general parameter P100 ...
... finally, the PLC "tells" the CNC to execute block M45 SP100 (live tool clockwise at the selected rpm).

CUSTOM AND M2 AND DFU B28R561 = CNCRD (LIVRPM, R117, M1) = CNCWR (R117, GUP100, M1) = CNCEX1 (M45 S-P100, M1)

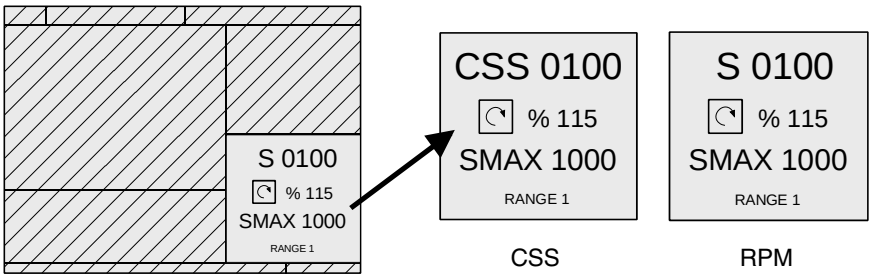
If in CO mode (CUSTOM=1), a live tool is selected (M2) and the O1 key "live tool counterclockwise" is pressed (DFU B28R561) ...
... the PLC reads in R117 the rpm currently selected for the live tool (LIVRPM) and assigns them to general parameter P100 ...
... finally, the PLC "tells" the CNC to execute block M45 S-P100 (live tool counterclockwise at the selected rpm).

2.6 Spindle control

The standard screen of the CO mode has a window that shows the following spindle related data.

This whole window may be customized.

Depending on the model, the one provided by Fagor can show the following information:



The data shown in that window depends on the selected work mode:

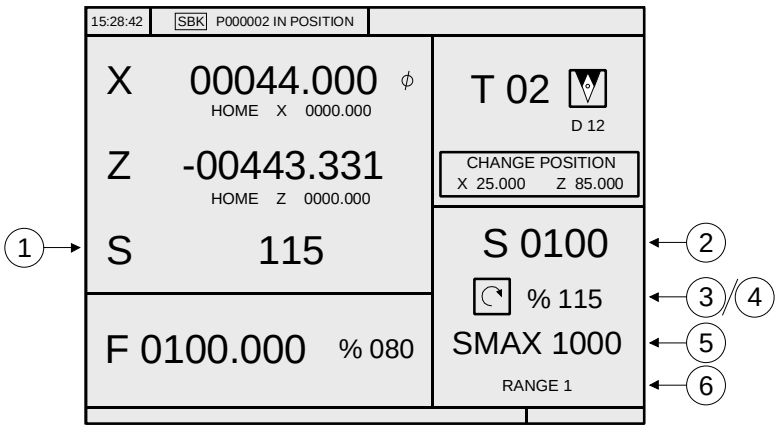
(RPM) Spindle in revolutions per minute.

(CSS) Constant surface speed. Available only at the TCO model.

On CNC power-up and after the keystroke sequence [SHIFT] [RESET], the CNC selects the rpm mode.

2.6.1 Spindle in rpm

The CNC displays the following information:



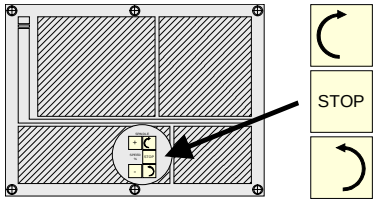
1. Real spindle speed in rpm.
2. Theoretical spindle speed in rpm.

   To select another speed, press [S] value [START].

   The CNC assumes that value and refreshes the real spindle speed.
3. Spindle status:

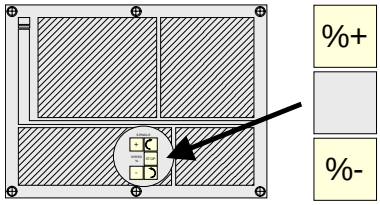
   turning clockwise, turning counterclockwise or stopped.

   To modify the spindle status, press the following keys:



4. % of the theoretical spindle speed being applied.

   To modify the percentage (%), press:



5. Maximum spindle speed in rpm.

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

   Enter the new value and press [ENTER]. The CNC assumes this value and does not allow the spindle to exceed these rpm.
6. Spindle gear currently selected.

   This value cannot be changed when using an automatic gear change.

   When not using an automatic tool changer, press the [S] key and then use the [⬇] key to highlight the current value.

   Enter the gear number to be selected and press [ENTER] or [START].

When the machine does not use spindle gears, this message makes no sense. That is why the CNC does not show this message when text 28 of program 999997 has not been defined.

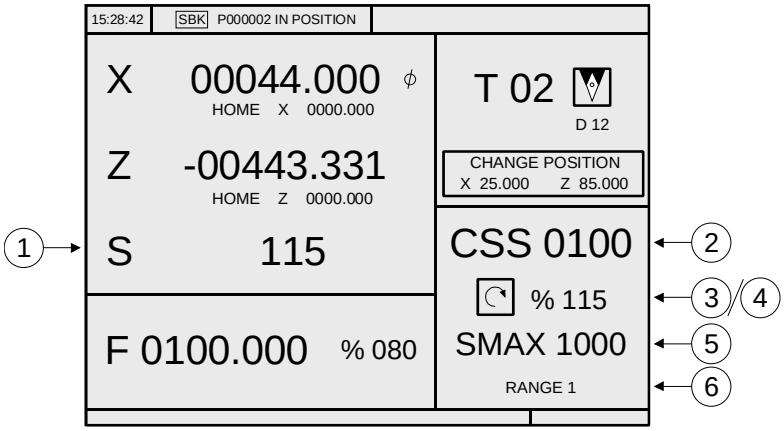
2.6.2 Constant surface speed

Available only at the TCO model.

In constant surface speed mode, the user sets the (linear) tangential speed that must always be kept between the tool tip and the part.

Therefore, the spindle rpm depend on the position of the tool tip with respect to the rotation axis. Thus, if the tool tip gets away from the rotation axis, the spindle slows down and if it gets closer, it speeds up.

When selecting the CSS mode, the CNC shows the following information:



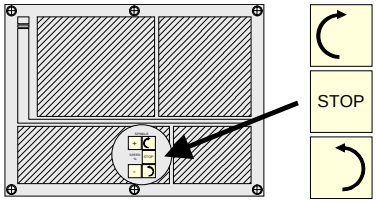
1. Real spindle speed in rpm.
2. Theoretical constant surface speed. It is set in mm/minute or feet/minute.

   To select another speed, press [S] value [START].

   The CNC assumes this value and if the spindle is turning, it updates the real spindle speed (in rpm)
3. Spindle status:

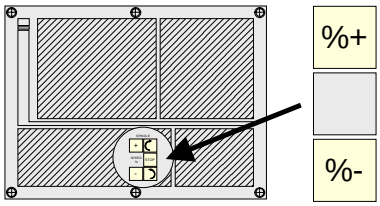
    turning clockwise,  turning counterclockwise or  stopped.

   To modify the spindle status, press the following keys:

4. % of the theoretical constant surface speed being applied.

   To modify the percentage (%), press:

5. Maximum spindle speed in rpm.

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

   Enter the new value and press [ENTER]. The CNC assumes that value.

6. Spindle gear currently selected.

   This value cannot be changed when using an automatic gear change.

   When not using an automatic tool changer, press the [S] key and then use the [⬇] key to highlight the current value.

   Enter the gear number to be selected and press [ENTER] or [START].



When the machine does not use spindle gears, this message makes no sense. That is why the CNC does not show this message when text 28 of program 999997 has not been defined.

Work at constant surface speed

When selecting the constant surface speed mode (CSS), the CNC assumes the spindle gear (range) that is currently selected.

The following cases may occur in this mode, when selecting a new constant surface speed value:

A. The spindle is stopped:

The CNC selects the new speed, but it does not apply it until the starts turning.

B. The spindle is running:

The CNC, depending on the position of the axis, calculates and turns the spindle at the corresponding rpm so the constant surface speed is the one defined.

The following cases may occur when moving the axes when working at constant surface speed:

A. The spindle is running.

The CNC moves the axes at the programmed F.

As the X axis moves, the CNC adapts the spindle speed (rpm) to maintain the selected constant speed.

Thus, if the tool tip gets away from the rotation axis, the spindle slows down and if it gets closer, it speeds up.

The CNC limits the spindle rpm to the maximum speed set by "SMAX".

B. The spindle is stopped, but a spindle speed S has been selected.

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

For example if «F 2.000» and «S 500»:

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

The axis moves at a feedrate of 1000 mm/min.

C. The spindle is stopped and no spindle speed S has been selected.

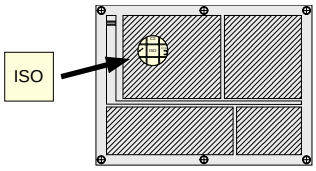
The machine does not move, only with rapid movements.

When pressing  and the corresponding axis axis.

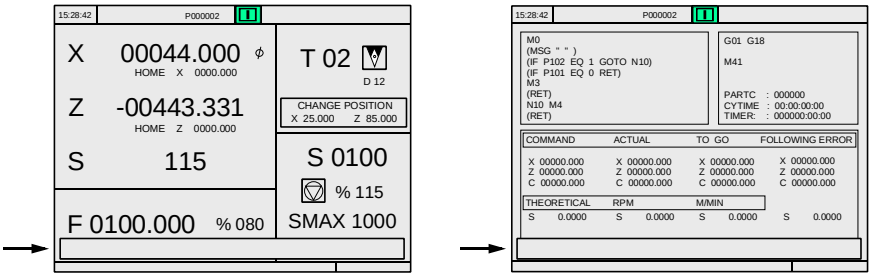
2.7 ISO management

Pressing the ISO key once gives access to the MDI mode. In this mode, a block may be edited and executed.

Pressing the ISO key twice gives access to the ISO mode. In this mode, up to 6 blocks may be edited and executed or simulated.



In MDI mode (pressing once), the CNC shows a window at the bottom of the screen.



In this window, a block may be edited and later executed.

In ISO mode (pressing twice), the CNC displays a special screen for editing up to 6 program blocks either in ISO code or in high level language.

Example: [ISO]

G95 G96 S120 M3 [ENTER]

G0 Z100 [ENTER]

G1 X30 F0.1 [ENTER]

Once the block or blocks have been edited, press [ESC]. The upper right side of the screen will show the [START] symbol.

From this moment one, the edited blocks may be simulated, executed or saved like any operation or cycle.

To simulate, press the [GRAPHICS] key. To execute, press the [START] key.

It is possible to combine ISO-coded blocks with standard and/or user cycles to create part-programs. The chapter on "Saving programs" describes in detail how to do it and how to use those programs.

To save the blocks edited in ISO code, press the [P.PROG] key.

2.

OPERATING IN JOG MODE
ISO management

FAGOR

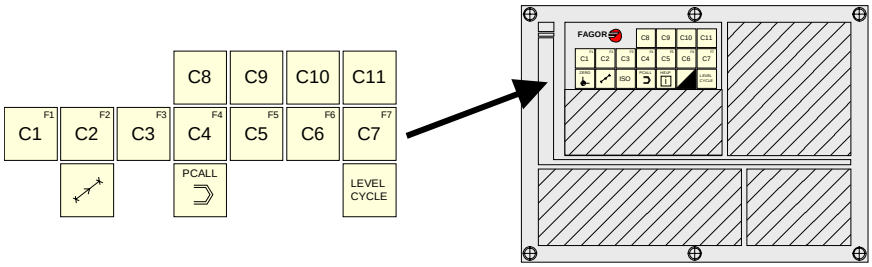
·MCO/TCO·
Option

Soft: V01.0x

WORKING WITH OPERATIONS OR CYCLES

3

Use the following keys to access the different machining cycles:



The standard version offers 2 positioning cycles.



Press the [POS.] key to access these cycles.



Press the [LEVEL CYCLE] to change the cycle.

The OEM can define up to 200 different cycles that will be grouped in 20 groups.

Keys [C1] through [C11] give access to the first 11 groups.

The OEM must set the access to the other groups in the PLC program as described later on. Doing that requires the use of external devices or free keys (blank keys or keys [O1] through [O14]).

To switch levels within each group, press the [LEVEL CYCLE] key.

The OEM cycles that may be accessed with each key are:

F1 C1	Cycles	1	21	41	61	81	101	121	141	161	181
F2 C2	Cycles	2	22	42	62	82	102	122	142	162	182
F3 C3	Cycles	3	23	43	63	83	103	123	143	163	183
F4 C4	Cycles	4	24	44	64	84	104	124	144	164	184
F5 C5	Cycles	5	25	45	65	85	105	125	145	165	185
F6 C6	Cycles	6	26	46	66	86	106	126	146	166	186
F7 C7	Cycles	7	27	47	67	87	107	127	147	167	187
C8	Cycles	8	28	48	68	88	108	128	148	168	188
C9	Cycles	9	29	49	69	89	109	129	149	169	189
C10	Cycles	10	30	50	70	90	110	130	150	170	190
C11	Cycles	11	31	51	71	91	111	131	151	171	191

The OEM cycles that may be accessed from the PLC, by simulating the key whose code is shown here, are:

$F10C	*Cycles*	*12*	*32*	*52*	*72*	*92*	*112*	*132*	*152*	*172*	*192*
$F10D	*Cycles*	*13*	*33*	*53*	*73*	*93*	*113*	*133*	*153*	*173*	*193*
$F10E	*Cycles*	*14*	*34*	*54*	*74*	*94*	*114*	*134*	*154*	*174*	*194*
$F10F	*Cycles*	*15*	*35*	*55*	*75*	*95*	*115*	*135*	*155*	*175*	*195*
$F110	*Cycles*	*16*	*36*	*56*	*76*	*96*	*116*	*136*	*156*	*176*	*196*
$F111	*Cycles*	*17*	*37*	*57*	*77*	*97*	*117*	*137*	*157*	*177*	*197*
$F112	*Cycles*	*18*	*38*	*58*	*78*	*98*	*118*	*138*	*158*	*178*	*198*
$F113	*Cycles*	*19*	*39*	*59*	*79*	*99*	*119*	*139*	*159*	*179*	*199*
$F114	*Cycles*	*20*	*40*	*60*	*80*	*100*	*120*	*140*	*160*	*180*	*200*

Each OEM cycle has the following items associated with it:

- The OEM screen of the same number, created with the WINDRAW55 application.
- The configuration file P999xxx.
- The subroutine that contains the executable 9xxx.
- Program P999995 that contains the texts used by all the screens.
- Program P999994 that contains the help texts of the cycles.

Thus, the configuration file P999004 and subroutine 9004 are associated with cycle 4.

When pressing [PCALL] , the CNC shows all the screens defined by the OEM with the WINDRAW55 application.

- Screens 001 through 200 correspond to the canned cycles.
- Screens 201 through 255 are customizing screens.

To delete any of them, position the cursor on it and press [CLEAR].

The CNC requests confirmation and the access code for customizing, if the OEM has previously defined it.

To execute a screen, position the cursor on it and press [ENTER].

Screens 201 through 255 cannot be executed because they are customizing screens.

Screens 001 through 200 can also be used for diagnosis, setup, CNC, PLC and drive status information, etc.

They are defined the same way as the canned cycles, they have a configuration file and a subroutine associated with them and the text they use must be defined in program P999995.



In the WINDRAW55 application, the text for to the screen title should be defined with the same number as the screen.

This way, when pressing, the key [PCALL], CNC will show the number and title (text of program P999995 with the same number) of the available screens.

3.1 Associated programs

In CO mode, programs P900000 through P999999 are reserved for the CNC itself; in other words, the user cannot use them as part-programs.

Some have a special meaning and must be used by the machine manufacturer (OEM).

P999001 through P999200 Configuration files defined by the OEM.

P999001 corresponds to cycle 1, P999002 to cycle 2 and so on up to P999200 that corresponds to cycle 200.

P999500 Program that logs (stores) the errors occurred when interpreting a configuration file.

P999994 Help texts of the OEM cycles (see WINDRAW55 manual).

P999995 Texts and titles used by all the OEM screens (see WINDRAW55 manual).

P994xxx When the OEM cycles allow to define profiles, the user defines them with 3 digits (from 0 to 999) and the CNC saves them internally as P994xxx.

Profile 1 in P994001, 2 in P994002 and so on.

The rest of the reserved programs are used internally by the CNC and cannot be erased.

3.2 Associated subroutines

In CO mode, subroutines 0000 through 8999 are free to use and subroutines 9000 through 9999 are reserved for customizing the CNC.

It is recommended to define all the OEM subroutines in a program with a high number to prevent the user from modifying them. Program P999999 is free and may be used for this purpose.

The following subroutines must be defined by the OEM.

9001 through 9200 Subroutines that contain the executable sources associated with the cycles defined by the OEM.

9001 corresponds to cycle 1, 9002 to cycle 2 and so on up to 9200 that corresponds to cycle 200.

9998 Subroutine that the CNC will execute at the beginning of each part-program.

9999 Subroutine that the CNC will execute at the end of each part-program.

Every time a new part-program (set of operations or cycles) is edited, the CNC inserts a call to the relevant subroutine at the beginning and at the end of the program.

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

Example of how to define subroutines 9998 and 9999.

```
(SUB 9998)      ; Definition of subroutine 9998
                ; Program blocks defined by the OEM.
(RET)           ; End of subroutine
(SUB 9999)      ; Definition of subroutine 9999
                ; Program blocks defined by the OEM.
(RET)           ; End of subroutine
```

If subroutines 9998 and 9999 are not defined, the CNC will issue an error message when trying to execute a part-program.

3.3 OEM cycles

As mentioned at the beginning of this chapter, up to 200 OEM cycles may be defined.

Each OEM cycle has the following items associated with it:

- The OEM screen of the same number, created with the WINDRAW55 application.
- The configuration file P999xxx.
- The subroutine that contains the executable 9xxx.
- Program P999995 that contains the texts used by all the screens.
- Program P999994 that contains the help texts of the cycles.

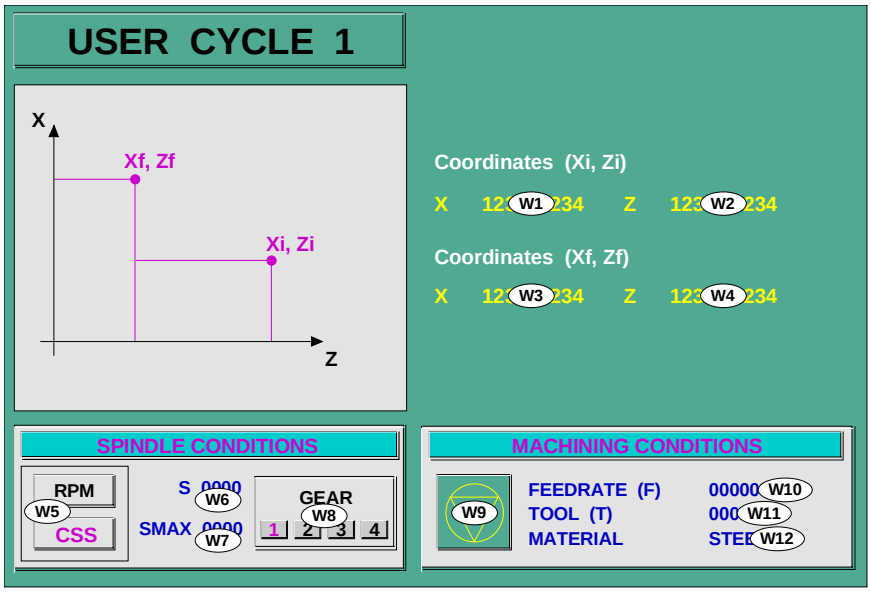
This section describes the steps to create OEM cycle 001 and how the user must set the data for it. In other words:

- How to create screen 001 using the WINDRAW55 application (OEM).
- How to define the configuration file P999001 (OEM).
- How to define the associated subroutine 9001 (OEM).
- How to edit cycle 001 (user).

3.3.1 Define the screen

We will use screen 001 of the example in the manual for the WINDRAW55 application.

The data that the user may edit in this cycle have the W identifier, the number associated with each one indicates the order in which they are selected (W1, W2, etc.). W11, W12).



When creating a screen using WINDRAW55, we recommend to divide the screen into several areas identifying parameter groups. We recommend to associate the widget numbers sequentially, but leaving unassigned numbers between two groups so other widgets can be added later without messing up the sequence. We also recommend to reserve numbers 1 through 5 so new fields can be added before them if necessary.

Each widget can have an associated text (program P999994) that is displayed in the text line at the bottom of the screen.

3.3.2 Configuration file

The configuration file is a CNC file written in high-level language (configuration language) that describes the operating characteristics of the various screen elements.

There is a configuration file for each cycle. P994001 corresponds to cycle 001, P994002 to 002 and so on up to P999200 that corresponds to cycle 200.

The general characteristics of the configuration language are:

- All instructions are preceded by ";" and enclosed in parenthesis.
- The comments must be alone and preceded by ";;".
- The configuration file must begin with the line ;(PRGSCRIPT 1).

  It indicates that it is a configuration file corresponding to the version being used (in this case "1")
- The configuration file should end with the line ;(END).
- It is recommended to use the ;(DEBUG) instruction for the CNC to check the configuration file. If an error occurs, it will log it in program 999500.

The configuration language has the following key words:

;(PRGSCRIPT 1)

Header of the configuration file and version used to edit it (in this case "1"). It must always be defined.

;(DEBUG)

It is optional. It indicates on which line program 999500 starts giving out information of the errors that have come up when debugging the configuration file.

The debugging of the configuration file begins on the first line ;(PRGSCRIPT 1) and ends on the ;(END) line.

;(DISABLE 1)

It indicates that the blue frame at the top right must not be displayed; it shows the current status of the machine (coordinates and machining conditions).

;(HOTKEY W4,88)

It permits associating a key to the (W) data.

In this case, pressing the [X] key (ASCII value 88) selects the W4 data.

;(TEACHIN W5=POSX)

Permits assigning the value of a CNC variable to the data.

In this case, While the W5 data is selected, when pressing the [RECALL] key, W5 will show the variable of POSX (X axis position). If [ENTER] is then pressed, the cycle assumes this value.

;(FORMAT W7,INCH)

The value is shown in the work units (mm / inches) set by machine parameter.

;(PROFILE W12)

Permits associating a profile with the (W) field. The field must be an unsigned 3-digit integer.

Select the (W) field, enter the number of the profile to be edited, press [RECALL] and it will access the profile editor.

The edited profile is stored as program P994xxx.

P994001 corresponds to profile 001, P994002 to 002 and so on up to P994999 that corresponds to profile 999.

;(P100=W13)

Permits assigning the value of the (W) data to a local or global parameter only when calling a subroutine. Two formats may be used:

(PCALL 9001, A=2, B=5) with local parameters (P0-P25).

(PCALL 9001, P100=2, P101=5) with global parameters (P100-P299).

With local parameters:

When executing a cycle, the CNC calls the associated subroutine indicating with local parameters which values have been defined in each field.

For example: (PCALL 9001, A=10, B=12, C=5, D=8)

Using local parameters with widgets between W1 through W25, the local parameter number is identified as follows:

W1 with A -> P0, W2 with B -> P1, etc.

With global parameters:

When using global parameters, the CNC uses another PCALL instruction to transfer the global parameters. USER or OEM parameters cannot be used.

For example: (PCALL 9301, P100=22, P101=32, P102=48)

(PCALL 9001, A10, B12, C5, ... Y8, Z100)

The subroutine associated with the cycle is 9000 + cycle number.

The auxiliary subroutine associated with the cycle is 9300 + cycle number.

When using a (P100=W13) type instruction, the auxiliary subroutine must also be defined even if it only contains the SUB and RET instructions.

;(W1=GUP100) To associate the value of a global parameter with the (W) data.

;(W2=PLCFRO) To associate the value of a variable with the (W) data.

;(W3=PLCR127) To associate the value of a PLC resource with the (W) data.

that of a Register ;(W6=PLCR127)

that of a Mark ;(W6=PLCM1000,1) first one and how many

that of a input group ;(W6=PLCI8,4) first one and how many

that of a olutput group;(W6=PLCO10,3) first one and how many

Associate only resources that are defined in the PLC program.

For marks, inputs and outputs, one must indicate how many of them, if none is indicated, 32 are assigned.

;(W6=PLCO11,4) Assigns the value of O11, O12, O13, O14

;(W6=PLCO11) Assigns the value of O11, O12 ... O41, O42

If a field (W) has a parameter, variable or resource associated with it, it acts as follows:

- It assumes the value that its associate has when accessing the page.

  To continuously update the field value, use the (AUTOREFRESH) instruction as described later on.
- If its associate is a read-only type, the user will not be able to change the field value.
- It its associate is read/write type, the user may change the field value; when the value of the (W) field is changed, the value of its associate is also changed. On the other hand, when using the (AUTOREFRESH) instruction and the CNC or PLC changes the value of the associate, the value of the field is also changed.
- If an error occurs because the variable does not exist (PLC register) or the communication fails (drive variable), the field is momentarily disabled and it displays a grey window. The field is enabled again after 10 seconds.

;(AUTOREFRESH W6=FLWEX)

Refreshes (updates periodically) the value of graphic element W6 assigning it the X axis following error value.

;(FORMAT W8,LEDBARDEC)

It must be used with Ledbar type (W) data that have a decimal variable associated with them. (For example: X axis following error).

The values assigned, at the PLC, to the end and intermediate values of a LEDBAR element must be integer values and must be related to the variable associated at the CNC.

When the associated variable has a decimal format the following instruction must be used:

;(FORMAT W8,LEDBARDEC)

This instruction is used to convert coordinate values (decimal) to integer values by multiplying them by 10000.

Examples:

To represent the % of axis feedrate, the FRO variable is used.

The FRO values are integers (between 0 and 120) and, therefore, do not require LEDBARDEC

;(AUTOREFRESH W9=FRO)

On the other hand, to represent the amount of following error on the X axis, the FLWEX variable is used.

The values of FLWEX are not integers and, therefore, require LEDBARDEC (multiplying it by 10000) in order to make them integers.

;(FORMAT W11,LEDBARDEC)

;(AUTOREFRESH W11=FLWEX)

;(MODALCYCLE)

It indicates that cycle is modal (refer to the programming manual).

The subroutine call is of the (MCALL 9001, A10, B12, C5,....) type).

If after executing the cycle, several movements are carried out, the cycle will be executed again after each movement calling the (MCALL 9001, A10, B12, C5,...) subroutine again.

When using global parameters, the CNC only transfers the global parameters the first time.

First time: (PCALL 9301, P100=22, P101=32, P102=48)

(MCALL 9001, A10, B12, C5, ... Y8, Z100)

Rest of the times: (MCALL 9001, A10, B12, C5, ... Y8, Z100)

To cancel this mode, execute the (MDOFF) instruction.

;(END)

It indicates that it has finished debugging the configuration file.

It ignores the instructions programmed next.

Configuration file (P999001) for cycle 001

%CFGFILE,MX,

;(PRGSCRIPT 1)

Header and version.

The debugging of the configuration file begins.

;(DEBUG)

Starting at this line, program 999500 keeps a log of the errors originated when debugging the configuration file.

;(HOTKEY W1,88)

;(TEACHIN W1=POSX)

The W1 field has the X hotkey (88) and the X axis coordinate associated with it. In other words:

Pressing the [X] key selects this field.

While this field is selected, if the [RECALL] key is pressed, it will show the X axis position. If [ENTER] is then pressed, the cycle assumes that value.

;(HOTKEY W2,90)

;(TEACHIN W2=POSZ)

The W2 field has the Z hotkey (90) and the Z axis coordinate associated with it. In other words:

Pressing the [Z] key selects this field.

While this field is selected, if the [RECALL] key is pressed, it will show the Z axis position. If [ENTER] is then pressed, the cycle assumes that value.

;(HOTKEY W3,88)

;(TEACHIN W3=POSX)

The W3 field has the X hotkey (88) and the X axis coordinate associated with it. In other words:

Pressing the [X] key selects the W1 field and pressing it again selects this field.

While this field is selected, if the [RECALL] key is pressed, it will show the X axis position. If [ENTER] is then pressed, the cycle assumes that value.

;(HOTKEY W4,90)

;(TEACHIN W4=POSZ)

The W4 field has the Z hotkey (90) and the Z axis coordinate associated with it. In other words:

Pressing the [Z] key selects the W2 field and pressing it again selects this field.

While this field is selected, if the [RECALL] key is pressed, it will show the Z axis position. If [ENTER] is then pressed, the cycle assumes that value.

;(HOTKEY W6,83)

;(HOTKEY W7,83)

The W6 and W7 fields have the S (83) hotkey associated with them. In other words:

Pressing the [S] key selects the W6 field and pressing it again selects the W7 field.

;(HOTKEY W10,70)

The W10 field has hotkey F (70) associated with it. In other words:

Pressing the [F] key selects this field.

;(HOTKEY W11,84)

The W11 field has hotkey T (84) associated with it. In other words:

Pressing the [T] key selects this field.

;(END)

End of the debugging of the configuration file and end of program.

3.3.3 Associated subroutine

The OEM must define the subroutine associated with the cycle (refer to the programming manual). It must contain all the necessary instructions to run the canned cycle.

There is one subroutine associated with each cycle. Subroutine 9001 corresponds to cycle 001, 9002 to 002 and so on up to 9200 that corresponds to cycle 200.

When the configuration file of the cycle uses an instruction like (P100=W13), the auxiliary subroutine must also be defined.

There is one auxiliary subroutine associated with each cycle. Subroutine 9301 corresponds to cycle 001, 9302 to 002 and so on up to 9500 that corresponds to cycle 200.

Therefore, the 2 subroutines associated with the cycle are:

The basic subroutine associated with the cycle9000 + cycle number

The auxiliary subroutine associated with the cycle9300 + cycle number.

When executing the cycle, its associated subroutine is called upon indicating in local parameters A - Z (P0 - P25) the value used to define each field.

Parameter A (P0) indicates the value of the W1 field, B (P1) that of the W2 field and so on up to Z (P25) that indicates the value of the W26 field. If there are more fields, use global parameters.

The data shown by each parameter depends on the type of associated data.

If it is a numeric data (coordinates, feedrate, etc.) .....The value assigned

If it is the number of a profile .....The value assigned

(the call to the profile must be done in the subroutine)

It is one of several available options (group of buttons)..... Values 0, 1, 2, etc.

If it is an icon with several representations (multiple in WINDRAW55)Values 0, 1, 2, etc.

The next table shows the values assumed by local parameters when setting cycle 001 defined earlier with the indicated values.

Field		Value	Parameter		
Xi	W1	11	A	(P0)	=11
Zi	W2	22	B	(P1)	=22
Xf	W3	33	C	(P2)	=33
Zf	W4	44	D	(P3)	=44
RPM / CSS	W5	RPM	E	(P4)	=0
S	W6	1234	F	(P5)	=1234
SMAX	W7	9999	G	(P6)	=9999
GEAR	W8	2	H	(P7)	=1
Spindle rotation	W9	Clockwise	I	(P8)	=1
Axis feedrate (F)	W10	1234.567	J	(P9)	=1234.567
Tool (T)	W11	12	K	(P10)	=12
Material type	W12	Aluminum	L	(P11)	=1

The fields that are not data fields return the following values:

W5 (RPM / CSS)	E = 0 if RPM	E = 1 if CSS		
W8 (GEAR)	H=0 if gear 1	H=1 if gear 2	H=2 if gear 3	H=3 if gear 4
W9 (spindle rotation)	I=0 if stopped	I=1 if clockwise	I=2 if counterclockwise	
W12 (material type)	L=0 if steel	L=1 if aluminum	L=2 if brass	L=3 if iron
	L=4 if plastic	L=5 if wood		

When executing the cycle with the indicated values, it calls subroutine 9001 like this:

(PCALL 9001, A=11, B=22, C=33, D=44, E=0, F=1234, G=9999, H=1, I=1, J=1234.567, K=12, L=1)

The subroutine (9001) associated with cycle 001

```
%SUBROUTINES,MX,
( SUB 9001)
```
Subroutine definition

Analyzing the machining conditions

```
(IF P4 EQ 0 P100=97 ELSE P100=96)
```
It checks RPM/CSS and prepares G97/G96 respectively in P100

```
(IF P5 LT P6 P101=P5 ELSE P101=P6)
```
If S>Smax, it assumes Smax. Leaves the value in P101

```
(P102=41+P7)
```
Prepares the gear (M41..M44), leaves the value in P102

```
(IF P8 EQ 0 P103=5 ELSE P103=2+P8)
```
Prepares spindle rotation in P103 (M3, M4, M5)

```
(IF P11 LT 4 P104=8 ELSE P104=9)
```
Coolant according to type of material. Coolant ON (M8) and coolant OFF (M9).
Coolant ON if steel, aluminum, brass or iron and coolant OFF if plastic or wood.

```
(P105=POSX, P106=POSZ)
```
Takes the coordinates of the cycle calling point

Machining of the cycle

```
G90 GP100 FP9 SP101 TP10 MP102 MP103 MP104
```
Sets the machining conditions

```
G92 SP6
```
Limits the spindle speed to the maximum indicated.

```
G0 XP0 ZP1
```
Rapid move to point Xi Zi

```
G1 XP2
ZP3
```
Move at working feedrate. First to Zf and then to Xf

```
G0 XP105 ZP106
```
Rapid return to the cycle calling point.

```
(RET)
```
End of subroutine

3.3.4 Error log file (P999500)

There is a configuration file for each cycle. P994001 corresponds to cycle 001, P994002 to 002 and so on up to P999200 that corresponds to cycle 200.

The CNC checks these programs when accessing for the first time. When detecting an error, it displays a message window.

In all of them, if the error has been detected in a section defined after the DEBUG instruction, they generate, in program P999500 several lines indicating the detected error or errors.

The error log file (P999500) contains all the errors detected since the CNC was turned on. When the CNC is turned off, this error log file (P999500) is deleted.

Examples of detected errors:

Error due to a nonexistent variable. It must be FLWEX

;(AUTOREFRESH W2=FLWEXX)

; Syntax error ...

; Unknown CNC variable name

; Error on line: 12

; Error on character: $^{L}_{F}$

Error caused for referring to a nonexistent graphic element (W33).

;(AUTOREFRESH W33=PLCR124)

; Warning ...

; Programmed Widget does not exist.

; Warning in line: 15.



After modifying the configuration file, reset the CNC so it can check it again.

3.4 Access to OEM cycles from the PLC

The OEM cycles that may be accessed from the PLC, by simulating the key whose code is shown here, are:

$F10C	Cycles	12	32	52	72	92	112	132	152	172	192
$F10D	Cycles	13	33	53	73	93	113	133	153	173	193
$F10E	Cycles	14	34	54	74	94	114	134	154	174	194
$F10F	Cycles	15	35	55	75	95	115	135	155	175	195
$F110	Cycles	16	36	56	76	96	116	136	156	176	196
$F111	Cycles	17	37	57	77	97	117	137	157	177	197
$F112	Cycles	18	38	58	78	98	118	138	158	178	198
$F113	Cycles	19	39	59	79	99	119	139	159	179	199
$F114	Cycles	20	40	60	80	100	120	140	160	180	200

The following example shows how to select 2 groups of cycles:

The blank key under the [ZERO] key selects and unselects the group 12, 32, 52, etc.

The external push button connected to input I27 that selects and unselects the group 13, 33, 53, etc.

( ) = MOV 1 R111 = MOV 0 R110 = MOV $F10C R101 = MOV $F10D R102

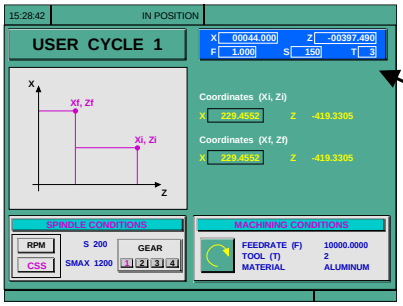
Initializations

DFU B26 R561	Every time the blank key is pressed ...
= CNCWR(R111, KEYSRC, M1)	... it indicates to the CNC that the keys come from the PLC
= CNCWR(R101, KEY, M1)	... it sends the key code for the group 12, 32, 52, etc.
= CNCWR(R110, KEYSRC, M1)	... and it indicates to the CNC that the keys come from the CNC

DFU I27	Every time the external key is pressed ...
= CNCWR(R111, KEYSRC, M1)	... it indicates to the CNC that the keys come from the PLC
= CNCWR(R102, KEY, M1)	... it sends the key code for the group 13, 33, 53, etc.
= CNCWR(R110, KEYSRC, M1)	... and it indicates to the CNC that the keys come from the CNC

3.5 Cycle data entry

Once the cycle has been selected, the CNC shows the relevant screen. It may have a blue frame at the top right indicating the current status of the machine. Coordinates and machining conditions.





One of the data that define the cycle will be highlighted indicating that it is selected.

To select another data, use the [⬆] [⬇] keys.

There are different types of data that are define as follows:

Numeric data that is used for coordinates, feedrate, spindle speed, tool number, etc. (value in WINDRAW55).

X 00044.000 Key in the desired value and press [ENTER].
In certain cases, the value of an internal variable (coordinate, tool number, etc.) may be assigned to this field. In those cases, press [RECALL] and [ENTER].

Numeric data for selecting a profile (3-character unsigned integer value in WINDRAW55).

PROFILE 001 Key in the number of the desired profile and press [RECALL]. The CNC accesses the profile editor.

Select an option among those available (button in a group of buttons of WINDRAW55).

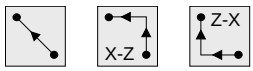


Use the [⬅] [➡] keys.

Select one of the representations in an icon (multiple in WINDRAW55).



Press the [TWO-COLOR] button until the desired icon appears:

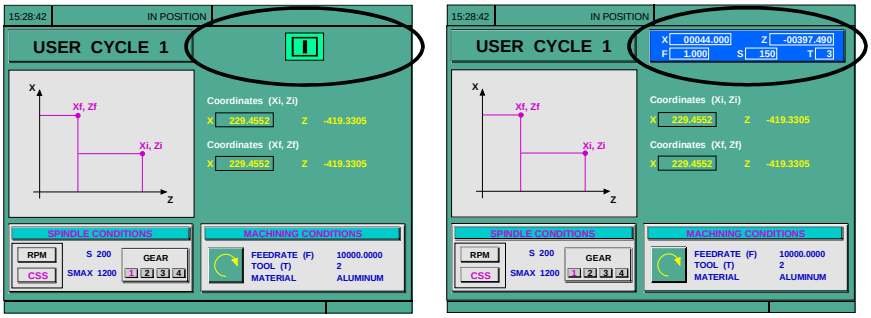


To de-select a cycle and return to the standard screen, press the key for the selected cycle or the [ESC] key twice.

The CNC lets combine, in a single part-program, several cycles and blocks edited in ISO code. Chapter *"4 Saving programs"* describes in detail how to do it and how to use those programs.

3.6 Simulation and execution

There are two ways (modes) to work with operations or cycles: Edit mode and Execution mode.



Execution mode

Edit mode.

Press [ESC] to switch from edit mode to execution mode.

To switch from execution mode to edit mode, press one of these keys: To define these variables, press the [F] key and then one of these keys: [⬅] [➡] [⬆] [⬇].

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

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

For further information, see chapter *"5 Execution and simulation"*

3.7 Background cycle editing

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

The new operation edited may be saved as part of a part-program other than the one being executed.

The operation being edited in background cannot be executed or simulated, and the current position of the machine cannot be assigned to a coordinate.

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

Press the [STOP] key=> Interrupts the execution and goes on editing in background.

Press the [ESC] key=> To quit background editing.

Then, carry out the tool inspection as described in chapter *"5 Execution and simulation"*

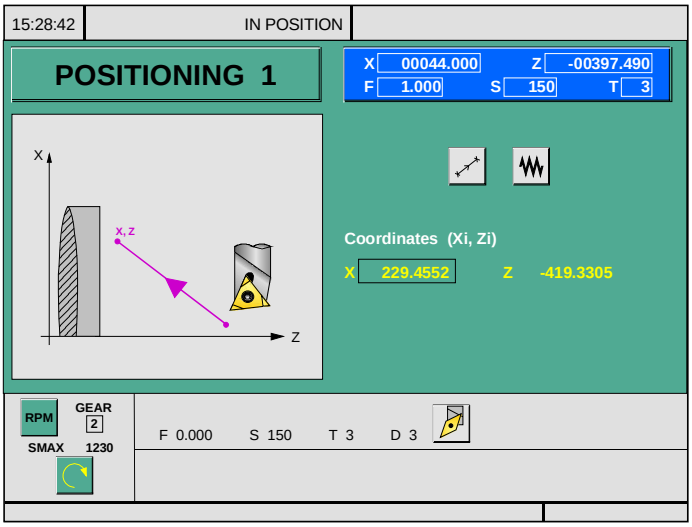
3.8 Positioning cycle



It selects the positioning cycle.

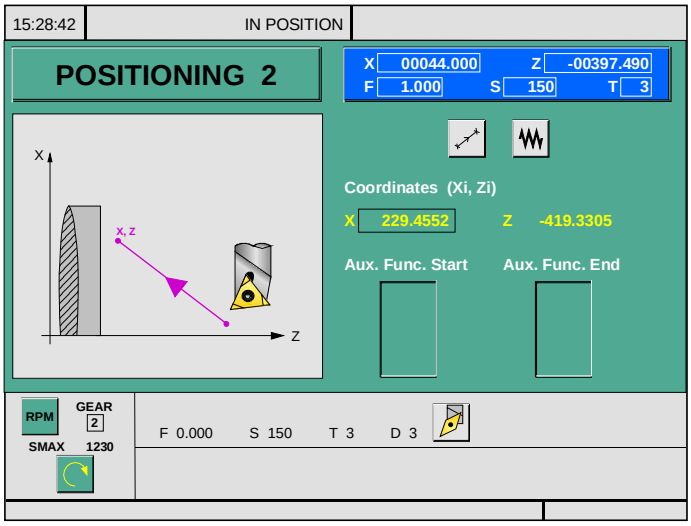
This cycle may be defined in two ways:

Level 1:



You must define	The coordinates of the target point.
	How the movement is to be carried out.
	Type of feedrate, rapid or at the selected F.

Level 2:

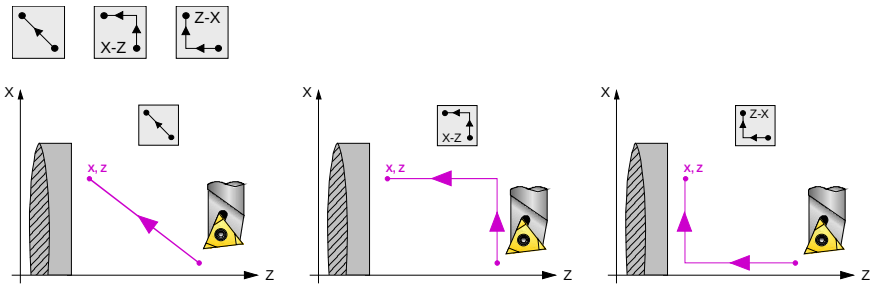


You must define	The coordinates of the target point.
	How the movement is to be carried out.
	Type of feedrate, rapid or at the selected F.
	The auxiliary functions that will be executed before and after the movement.

To switch levels within each group, press the [LEVEL CYCLE] key.

3.8.1 Definition of data

Type of positioning:



To select the type of movement, place the cursor over this icon and press the [TWO-COLOR] key. The icon changes and the help graphics are refreshed.

Type of feedrate:

 Feedrate at the indicated F.  Rapid feedrate.

To select the type of feedrate, place the cursor over this icon and press the [TWO-COLOR] key.

Coordinates of the target point (X, Z):

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

A. Entering the value manually.

Key in the desired value and press [ENTER].

B. Assign the current machine position.

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

Press [RECALL] so the selected data assumes the value shown in the top right window.

Press the [ENTER] key.

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

Auxiliary "M" functions are those set by the manufacturer for governing various devices of the machine.

Some auxiliary "M" functions are used to interrupt the program, to select the spindle turning direction, to control the coolant, the spindle gear box, etc.

The programming manual describes how to program these functions and the installation manual describes how to set up the system to use them.

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

A. Place the cursor in the corresponding window with the [⬅] [➡] keys

Use the [⬆] [⬇] keys to move around the window.

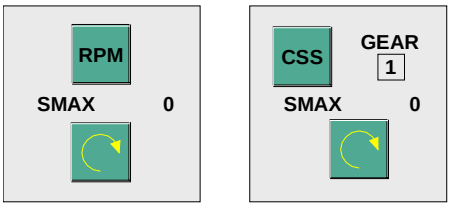
B. Define the auxiliary functions required.

The functions will be executed in the same order they are inserted on the list.

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

3.8.2 Definition of spindle conditions

Work mode (RPM) or (CSS):



Move onto the "RPM" or "CSS" icon and press the [TWO-COLOR] key to change the work mode.

Spindle gear (range):

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

Maximum spindle speed (S) in rpm:

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

Spindle turning direction:

There are 2 ways to select the spindle turning direction:

A. Place the cursor on this data and press the [TWO-COLOR] key to change the icon.

B. Start the spindle in the direction required by means of the JOG keys:



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

3.8.3 Definition of machining conditions

Axis feedrate (F):

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

Spindle turning speed (S):

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

Tool for machining (T):

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

The CNC updates the associated offset (D) and refreshes the associated icon showing the graphics for the location code (shape) of the new tool.

Tool offset number (D):

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

3.

WORKING WITH OPERATIONS OR CYCLES

Positioning cycle

FAGOR

·MCO/TCO·
Option

Soft: V01.0x

SAVING PROGRAMS

4

A part-program is the set of operations that must be executed to make a particular part (workpiece).

To create a part-program, just define each basic operation and save them in the proper order.

The possibility to use ISO-coded blocks as part of the program makes it possible to define prior machining conditions, activate particular devices or even interrupt the machining of the part.

This chapter indicates how to operate with those part-programs through the following sections and sub-sections.

List of saved programs.

See the contents of a program ..... See one of the operations in detail.

Edit a new part-program ..... Save an operation or cycle.

Deleting a new part program.

Copying a part-program into another one.

Modify a part-program ..... Delete an operation.

Move an operation to another position.

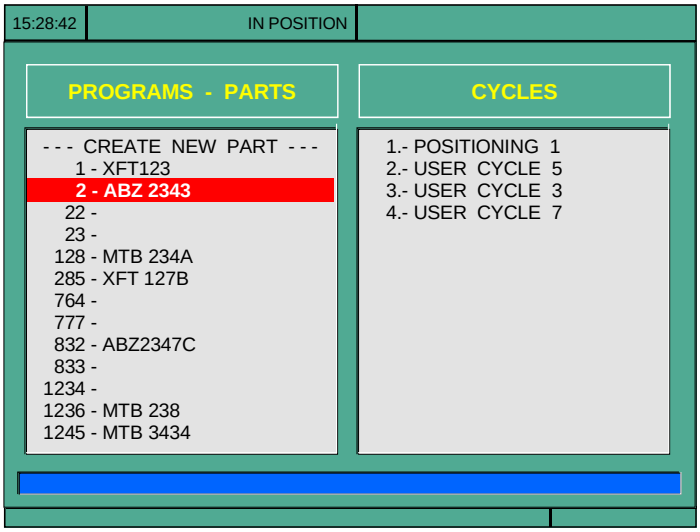
Add or insert a new operation.

Modify an existing operation.

Chapter *"5 Execution and simulation"* describes how to simulate and execute the part-programs.

4.1 List of saved programs

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



The left side shows the list of part-programs that are saved in the CNC's memory.

When there are more programs than those appearing in the window, use the [⬆] and [⬇] keys to move the pointer over the list of programs. Use the [SHIFT] [⬆] and [SHIFT] [⬇] keys to scroll a page at a time.

The right column shows the cycles and blocks in ISO code that make up the part.

Once the list of programs has been selected, the CNC lets:

- Creating a new part-program.
- Seeing the contents of a part-program.
- Deleting a new part program.
- Copying a part-program into another one.
- Modifying a part-program.

To exit the directory or list of part-programs, press:

The [ESC] key.

The key for an operation or cycle.

The [ISO] key.

In the WINDRAW55 application, the text for the title should be defined with the same number as the cycle. Since the CNC shows, next to the number, the title of the cycle (text of the program P999995 of the same number)

4.2 See the contents of a program

To see the contents of a part-program, select it on the left column using the pointer. To do that, use the [⬆] and [⬇] keys.

The right column shows the cycles that make up the part.

When pressing [ENTER] or one of the [⬅] or [➡] keys, the pointer switches to the right column.

Now, the [⬆] and [⬇] keys may be used to move the pointer over the blocks or cycles that make up the part.

Briefly, use the keys:

[⬆] and [⬇]to move up and down in each column.

[⬅] and [➡] to change columns.

Once an operation has been selected, the CNC lets:

- To see the operation in detail.
- To delete the operation.
- To move the operation to another position.
- To modify the operation.

4.2.1 See one of the cycles in detail

Once the desired operation has been selected using the pointer, press [RECALL].

The CNC will extract the data saved and will display them on the screen of the corresponding cycle and, once there, it will be possible to:

- Modify data.
- Simulate the operation. See chapter *"5 Execution and simulation"*.
- Execute the operation. See chapter *"5 Execution and simulation"*.
- Save the operation again. Replacing the previous one or inserting it as a new one.

4.3 Edit a new part-program

To edit a new program, proceed as follows:

- Press [P.PROG] to access the list of part-programs stored.
- Select, with the pointer, the option "Creating new part" on the left column.
- Press the [P.PROG] key. The bottom of the screen will request the number to be assigned to the new part-program and it will offer the first one available.
- Enter the number of the desired program and press [ENTER].

  It must be a number between 1 and 899999, both included.
- The CNC requests the comment to be associated with the part-program.

  It is not necessary to associate a comment.
- Press the [ENTER] or [ESC].

  The CNC includes the new part-program in the list (left column).

From this moment on, all the desired operations may be saved in the desired order.

4.3.1 Saving an ISO block or a cycle

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

To save a block or cycle, proceed as follows:

- Define the desired block or cycle assigning the relevant data to it.
- Press [P.PROG] to access the list of part-programs stored.
- Select, with the pointer, the desired program number on the left column and switch over to the right column.
- Place the pointer on the operation after which the block or cycle is to be saved and press [ENTER].

Example:	**You get**	**You want**
	1.- User cycle 2	1.- User cycle 2
	2.- User cycle 5	2.- User cycle 5
	3.- User cycle 11	3.- User cycle 11
		4.- User cycle 3
	4.- User cycle 1	5.- User cycle 1
	5.- User cycle 8	6.- User cycle 8
		7.- User cycle 9

4.- User cycle 3	Once the cycle has been defined, place the cursor on the operation "3.- User cycle 11" and press [ENTER].
7.- User cycle 9	Once the cycle has been defined, place the cursor on the last operation "User cycle 8" and press [ENTER].

4.4 Delete a new part program

To delete a part-program, proceed as follows:

- Press [P.PROG] to access the list of part-programs stored.
- Select, with the pointer, the part-program to be deleted from the left column.
- Press [CLEAR].

  The bottom of the screen will show a message requesting confirmation of the deleting operation.

  If [ENTER] is pressed, the CNC will delete the selected program and will update the list of saved programs.

  If [ESC] is pressed, the program will not be deleted and it will quit deleting operation.

4.5 Copying a part-program into another one

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

- Press [P.PROG] to access the list of part-programs stored.
- Select, with the pointer, the part-program to be copied from the left column.
- Press the [P.PROG] key.

  The bottom of the screen will show a message requesting the number to be assigned to the copy.
- Enter the number of the desired program and press [ENTER].

  It must be a number between 1 and 899999, both included.
- If there is already a part-program with that number, the bottom of the screen will show a message asking to either replace it or cancel the operation?

  If [ENTER] is pressed, the CNC will request a new program number.

  If [ESC] is pressed, the CNC will not delete the current program and will copy the program.
- The CNC requests the comment to be associated with the new part-program (the copy).

  It is not necessary to associate a comment.
- Press the [ENTER] or [ESC].

  The CNC updates the list of saved part-programs.

4.6 Modify a part-program

To modify a part-program, proceed as follows:

- Press [P.PROG] to access the list of part-programs stored.
- Select, with the pointer, the part-program to be modified from the left column.

Once the program has been selected, the CNC lets:

- To delete an operation.
- To move an operation to another position.
- To add or insert a new operation.
- To modify an existing operation.

4.6.1 Delete an operation

To delete an operation, proceed as follows:

- Select, with the pointer, the operation to be deleted from the right column.
- Press [CLEAR].

  The bottom of the screen will show a message requesting confirmation of the deleting operation.

  If [ENTER] is pressed, the CNC will delete the selected operation and will update the right column.

  If [ESC] is pressed, the operation will not be deleted and it will quit deleting operation.

4.6.2 Move an operation to another position

To move an operation to another position, proceed as follows:

- Select, with the pointer, the operation to be moved from the right column.
- Press the [TWO-COLOR] key.

  The CNC will highlight that operation.
- Place the cursor on the operation after which the operation is to be placed and press [ENTER].

Example:

You get	You want
1.- User cycle 2	1.- User cycle 2
2.- User cycle 5	2.- User cycle 11
3.- User cycle 11	3.- User cycle 3
4.- User cycle 3	4.- User cycle 1
5.- User cycle 1	5.- User cycle 8
6.- User cycle 8	6.- User cycle 5
7.- User cycle 9	7.- User cycle 9

Select "User cycle 5" and press the [TWO-COLOR] key.

Place the cursor on "User cycle 8" and press [ENTER].

4.6.3 Add or insert a new operation

To add or insert an operation follow the same steps as for saving an operation.

- Define the desired block or cycle assigning the relevant data to it.
- Press [P.PROG] to access the list of part-programs stored.
- Place the pointer on the operation after which the block or cycle is to be saved and press [ENTER].

4.6.4 Modify an existing operation

To modify an operation, proceed as follows:

- Select, with the pointer, the block or cycle to be modified from the right column.
- Press the [RECALL] key.

  The CNC will show the editing page for that operation.
- Modify all the desired data.

To save the modified operation, proceed as follows:

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

  The CNC shows the pointer on the same operation.

  To select another position, use the [⬆] and [⬇] keys. The new operation will be inserted after it.
- Press the [ENTER] key.

  To place the modified operation in its previous location, the CNC will show a message asking to either replace the previous operation or keep it inserting the new one after it.

  The following example modifies the "user cycle" operation.

You get	Replace option	Insert option
1.- User cycle 5	**1.- User cycle 5**	**1.- User cycle 5**
2.- User cycle 2	2.- User cycle 2	**2.- User cycle 5**
		3.- User cycle 2

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

4.

SAVING PROGRAMS
Modify a part-program

FAGOR

·MCO/TCO·
Option

Soft: V01.0x

EXECUTION AND SIMULATION

5

The simulation may be used to graphically show a part-program or an operation with the data used to define it.

This way, the simulation may be used to check the part-program or the operation before executing it or saving it and, therefore, correct or modify its data.

It is possible to execute or simulate a part-program or any operation. The simulation or execution may be done from beginning to end or step by step by pressing the [SINGLE] key.

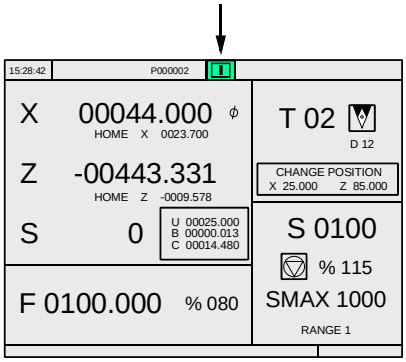
It is possible to simulate or execute:

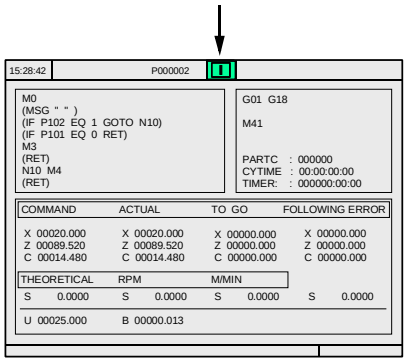
- An operation or Cycle.
- A part-program.
- An operation saved as part of a part-program.

Selecting a program for simulation or execution



Symbol that appears in the middle of the top window when a part-program has been selected for simulation or execution.





When the top center window shows the part-program selected next to the green [START] symbol, the CNC acts as follows:

- If [START] is pressed, the CNC executes the part-program that is selected.
- If [CLEAR] is pressed, the CNC de-selects the part-program and removes it from the top center window.

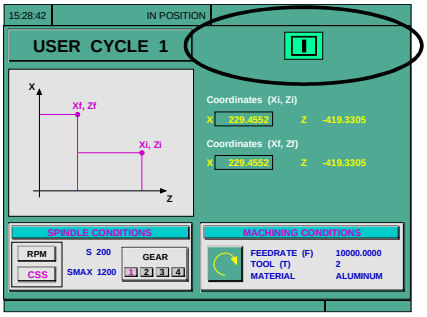
FAGOR

·MCO/TCO·
Option

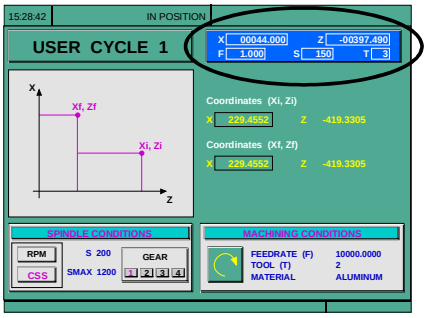
Soft: V01.0x

5.1 Simulating or executing an operation or cycle

All operations or cycles have 2 work modes: Execution and edit mode.



Execution mode



Edit mode.

Simulation:

The operation or cycle may be simulated in both work modes. To do that, press the [GRAPHICS] key. The CNC shows the graphic screen.

Execution:

An operation or cycle can only be executed in the cycle execution mode.

Press [ESC] to switch from edit mode to execution mode.

Press [START] to execute an operation or cycle.

5.2 Simulating or executing a part-program

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

- Press [P.PROG] to access the list of part-programs stored.
- Select on the left column the program to be simulated or executed.

Press [GRAPHICS] to simulate the part-program and [START] to execute it.

5.2.1 Simulating or executing a portion of a part-program

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

- Press [P.PROG] to access the list of part-programs stored.
- Select the program from the left column and the operation (on the right column) from which to begin executing or simulating the part program.

Press [GRAPHICS] to simulate the part-program and [START] to execute it.



When executing a portion of a program, the CNC does not execute the initial subroutine 9998 that is associated with all part-programs.

5.3 Simulating or executing an operation that has been saved

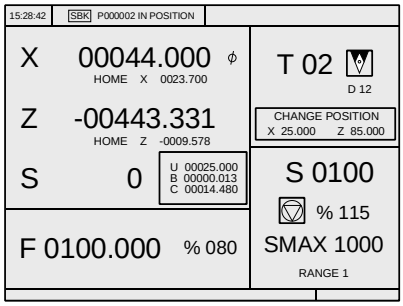
Proceed as follows to simulate or execute an operation that has been saved as a part of program:

- Press [P.PROG] to access the list of part-programs stored.
- Select the program that contains it from the left column and the operation to be simulated or executed from the right column.
- Press the [RECALL] key.

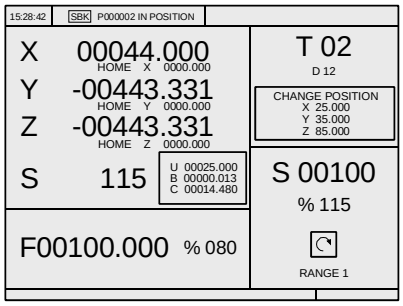
Press [GRAPHICS] to simulate the operation and [START] to execute it.

5.4 Execution mode

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

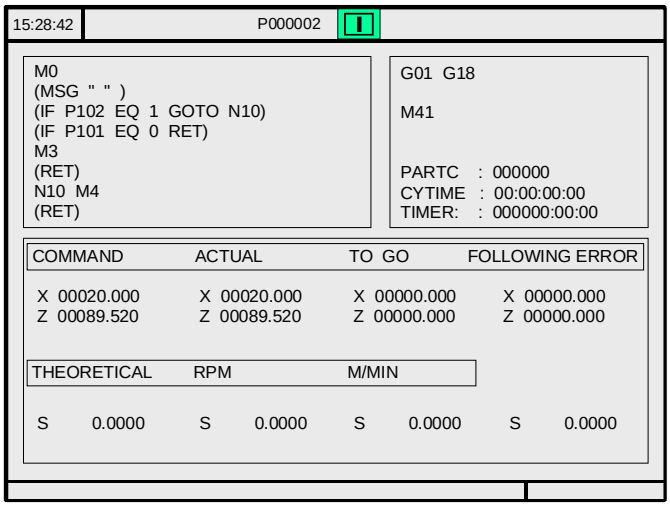


TCO model



MCO model

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



Once the operation or part has been selected, it may be executed as often as desired; to do that, once the execution has finished, pressed the [START] again.

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

To interrupt the execution, press [STOP].

Once the execution has been interrupted, a tool change may be done. See next section.

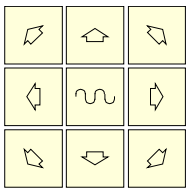
5.4.1 Tool inspection

Depending on how the PLC mark TOOLINSP has been set, the tool inspection mode will be available after interrupting the execution or it will be necessary to interrupt the execution and press the [T] key.

Once tool inspection has been selected, it is possible:

Move the axes to the tool change point:

Use the keys or the handwheels to move the tool to the position where the tool change will be carried out.



Select another tool:

The standard screen of the CO mode must be selected in order to be able to make a tool change.

Press the [T] key. The CNC highlights the tool number.

Type in the number of the tool to be selected and press [START] so the CNC selects the new tool.

The CNC will manage the tool change.

Resuming program execution:

Press the [START] key to resume program execution.

The CNC will reposition the tool by moving it to the point where tool inspection began. There may be 2 cases:

1. Only one axis moved.

   The CNC repositions it and resumes the execution.

2. Both axes moved.

   The CNC shows a window with the possible options. Choose the desired repositioning order.

5.5 Graphic representation

When pressing the [GRAPHICS] key, the CNC shows the graphics screen.

To quit the graphics mode, press either [GRAPHICS] or [ESC].

The operation manual, models M-MC or T-TC, "Graphics" section of the "Execution / Simulation" chapter describes how to use the graphics display. However, here is a brief description of the softkeys.

Type of graphics:

It depends on the selected model, MCO / TCO. Refer to the relevant operation manual.

Display area:

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

Use the [⬆] and [⬇] keys to select the maximum and minimum position values.

Once all the data has been defined, press [ENTER].

When selecting a new display area, the CNC clears the screen and shows the axes or the part unmachined.

The display area cannot be changed while executing or simulating the part. If it is, interrupt the execution or simulation by pressing the [STOP] key.

Zoom:

With this option, it is possible to enlarge or reduce the graphics display area.

It shows a window superimposed on the current graphics and another one over the drawing at the lower right-hand side of the screen. These new windows indicate the new display area being selected.

Use the [⬅] [➡] [⬆] [⬇] keys to move the window around, the [+] [-] keys to enlarge or shrink its size and press [ENTER] for those new values to be assumed.

Every time a new display area is selected, the CNC keeps the current graphic representation. It does not delete it.

When pressing [START] to resume or re-start the execution or simulation, it deletes the current graphic representation and starts a new one with the new values.

The zoom function cannot be used while executing or simulating the part. If it is, interrupt the execution or simulation by pressing the [STOP] key.

Graphic parameters:

Simulation speed:

Select on the top right side of the screen the percentage of simulation speed to be applied.

Use the [⬅][➡] keys to select the percentage and press [ENTER] for the CNC to assume that value.

Tool path colors:

It only makes sense in line graphics (not in solid ones). It lets you choose the colors to represent rapid traverse, the tool path without compensation and the tool path with compensation.

Use the [⬆] [⬇] keys to select, on the right side of the screen, the type of path and use the [⬅] [➡] keys to select the color to be applied.

Press [ENTER] for the CNC to assume those values.

Colors of the solid graphics:

It only makes sense in solid graphics (not in line graphics). It lets you use the colors to represent the cutter, the part, the axes and the clamps.

Use the [⬆] [⬇] keys to select, on the right side of the screen, the type of path and use the [⬅] [➡] keys to select the color to be applied.

Press [ENTER] for the CNC to assume those values.

Clear screen:

When selecting this option, the CNC clears the screen and shows the axes or the part unmachined.

The screen cannot be cleared while executing or simulating the part. If it is, interrupt the simulation by pressing the [STOP] key.

Once the type of graphics, the display area, the graphic parameters, etc have been selected, press the [START] key to start graphic simulation.

During graphic simulation, the CNC takes into account the simulation speed and the position of the right feedrate override switch (0%-120% FEED).

When selecting a new simulation speed, the CNC applies 100% of it regardless of the position of the switch.

Once the switch is turned, the CNC begins to apply the selected %.

To interrupt the simulation, press [STOP].

To quit the simulation mode, press either [GRAPHICS] or [ESC].

5.

EXECUTION AND SIMULATION
Graphic representation

FAGOR

·MCO/TCO·
Option

Soft: V01.0x



