



Aurki, S. Coop. Ltda.



8020 T

OPERATING MANUAL

A T T E N T I O N

SUBSTITUTE THE FOLLOWING PAGES FOR THE
HEREWITH ATTACHED.

2. FRONT PANEL (See figure)

1. SCREEN : Displays every type of information:
 - Operating mode listing
 - Active operating mode
 - Program and block being executed
 - Program and block being edited
 - Map of programs contained in memory
 - Coordinate values of the axes
 - Following error
 - Feedrate, speed of the spindle, active functions
 - Tool table
 - G53/G59 zero offset table
 - Error codes
 - Graphic representation (models TS and TG)
2. OPERATE MODE : Allows the display on the screen of the operating modes list. This is the first step in accessing any of them.
3. DISPLAY MODE : Allows the display on the screen of different types of information within the selected operating mode.
4.  : Enables forward and backward movement of displayed program blocks, as well as the tool table and the cursor.

5. DELETE : Allows deletion of a complete program or program block. It can also erase the table of decoded M functions. Deletion of the graphic representation.
6. ENTER : To enter data into the CNC memory.
7. RECALL : To access a program, a block within a program or a tool within its corresponding table.
8. NEXT : To pass on to a subsequent stage in the different modes of operation of the CNC.
9. CL : To clear the characters one by one during the process of editing a block.
- 10.P : To access a program and to program parameters.
- 11.N : To access a block within a program and to identify subroutines.
- 12.7 ... M : Keyboard for program execution.

3.1.2. Display modes

Display modes in AUTOMATIC or SINGLE BLOCK are the following:

- 0 - STANDARD
- 1 - POSITION
- 2 - FOLLOWING ERROR
- 3 - SUBROUTINE STATE/PARAMETERS
- 4 - EDITING (BACKGROUND)
- 5 - DISPLAY AREA DEFINITION
- 6 - GRAPHIC

Display modes 5 and 6 are only available for models TS and TG.

3.1.2.1. Selection of the display mode

- Press DISPLAY MODE : The display modes will appear on the screen.
- Press the desired number.

3.1.2.2. STANDARD display mode (0)

This mode is automatically set on selecting the AUTOMATIC/SINGLE BLOCK operating mode.

Information displayed on the screen:

- Upper part : The message AUTOMATIC/SINGLE BLOCK followed by the program number, the number of the first block to be executed or being executed.

Underneath this, the contents of the first blocks of the program or of the block being executed and the subsequent blocks (2 or 3).
- Central part : Under the headings COMMAND, ACTUAL and TO GO appear the arrival readings of the X,Z,C axes, the position of the said axes and the remaining distance, respectively.

Under COMMAND, the programmed S value affected by the speed override percentage. Under ACTUAL, the real S value. Under TO GO, (RPM) or (M/MIN).
- Lower part : The programmed values of F and S and their %'s appear as well as the list of activated G,T, and M functions.

3.1.2.3. POSITION display mode (1)

The position of the X,Z,C axes, the real S value and the programmed T are displayed in large characters. Similarly, the number of the program, the block number and the state of the G,M,T,S and F functions are also displayed.

3.1.2.4. FOLLOWING ERROR display mode (2)

The following error of the X,Z,C axes is displayed, as well as the program number, the block number and the state of the G,M,T,F and S.

NOTE:

The CNC 8020 TS will display the C axis in display modes 0,1 and 2 only if P102(2)=1.

3.1.2.5. SUBROUTINE STATUS, CLOCK, PARTS COUNTER and PARAMETER VALUES display mode (3)

Similar to display mode (0) except that instead of the following blocks to be executed the active subroutines are displayed according to the following format:

Standard subroutines: N2.2

Number of the subroutine		Number of times left to be executed
-----		-----

Parametric subroutines :P2.2

Number of the subroutine		Number of times left to be executed
-----		-----

Repetition of subroutines (G25) : G25.2

It identifies a repetition of subroutines vis G25,G26,G27,G28 or G29		Number of times left to be executed
-----		-----

3.1.2.6. Background programming (4)

A new program can be edited while the CNC is executing another program in AUTOMATIC or SINGLE BLOCK mode. To do this:

- Press DISPLAY MODE; the screen displays:

- 0 - STANDARD
- 1 - POSITION
- 2 - FOLLOWING ERROR
- 3 - SUBROUTINE STATE
- 4 - EDITING (BACKGROUND)
- 5 - DISPLAY AREA DEFINITION
- 6 - GRAPHIC

- Press key number 4, the screen will display:

```

AUTOMATIC/SINGLE BLOCK P -----
***AVAILABLE KEYS ***
*DISPLAY MODES (DIS MODE)
*PROGRAM NUMBER (P)
*NEXT

```

The program number P ----- corresponds to the last program edited, which may or may not be the one being executed.

If DISPLAY MODE is pressed, the situation prior to pressing key 4 is recovered.

From then on the process is similar to the one described in operation mode 6 EDITING.

WARNING

The program being executed cannot be edited or modified.

It is recommended to assign program numbers not already recorded in the memory to programs edited in this mode. Since if the program being executed calls upon subroutines in other programs, error 001 may occur.

The available keys in AUTOMATIC/SINGLE BLOCK mode plus the MF0 remain active during background programming.

NOTE:

Models TS and TG incorporate display modes 5 and 6, used to graphically represent the part program. See section 3.10 for their description.

3.9. SPECIAL MODES

The information on this section is in the INSTALLATION AND START UP MANUAL.

3.10. GRAPHICS

With this feature (models TS and TG), the tool path can be displayed on the CRT, as the program is being executed in one of the following modes:

AUTOMATIC, SINGLE BLOCK, TEACH IN, DRY RUN.

In DRY RUN mode:

- If THEORETICAL PATH 3 is selected, the system checks the program and displays the tool tip's path in solid lines.
- If mode 0 or 1 is selected, the tool centers' path will be displayed in dotted lines.
- If, when executing a program in modes 0,1 or 3, there is a block involving movement plus the function (Tx.x) the relevant path will not be displayed.

In the remaining modes, the tool's real path is displayed in dotted lines. The distance between dots varies according to the value of F.

056

The CNC 8020 TS will issue this error code:

- a) When trying to generate a program using function G76 while the memory is locked.
- b) When the program being generated with function G75 P5 is the protected program or program number P99999.
- c) When G22 or G23 go after function G76.
- d) When more than 70 characters have been written after G76.
- e) When G76 (block content) is programmed without G76 P5 or G76 N5 being programmed previously.
- f) When in a G76 P5 or G76 N5 type function the 5 digits of the program number are not programmed.
- g) When a program number is changed without cancelling the previous one while a program is being generated (G76 P5 or G76 N5).
- h) When the program mentioned in a block G76 P5 exists in memory but is not located in the last position of the memory map.

NOTE: When a program is called for to be edited goes to the last position in the memory map and when it is executed it goes to the first position.

057

Write protected tape.

058

Sluggish tape rotation.

059

Communication error between CNC and tape reader.

060

Circuitry malfunction (Interpolar CPU).

061

Battery fault. It must be remembered that, from the moment this error code is generated, the 3.5 V Lithium battery will save the memory's information during 10 days, with the CNC switched off. Since the battery is not rechargeable, the complete battery module, located at the rear of the unit, has to be replaced. Consult FAGOR Service.

CAUTION: - Do not try to charge the battery.
 - Do not expose the battery to temperatures over 100 degree centigrades.
 - Do not short-circuit the battery.

To avoid the risk of explosion or combustion.

064*

External emergency activated.