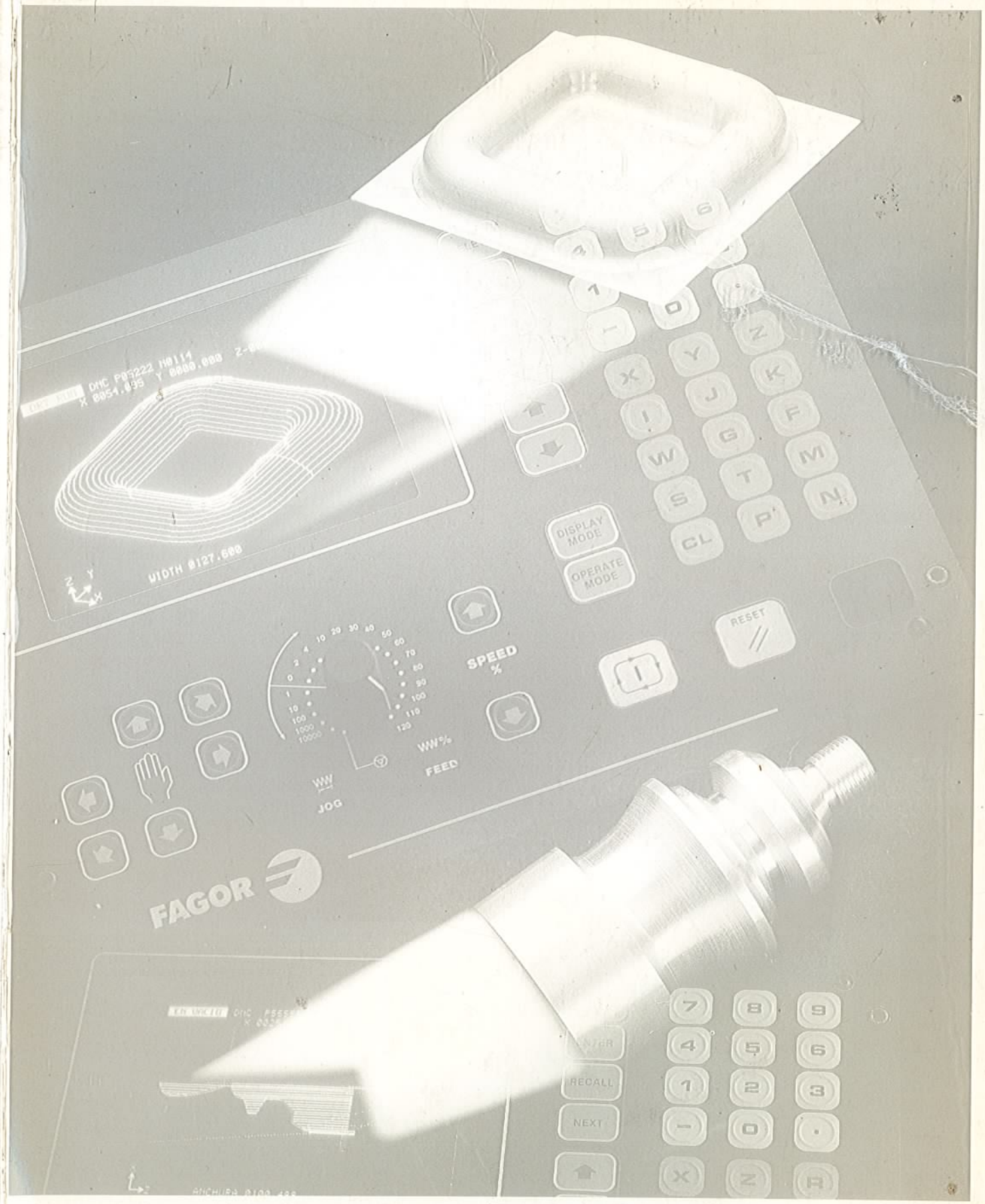


FAGOR 8020 T
Operating Manual

FAGOR

SOF 709

FAGOR 8020 T

OPERATING MANUAL

AURKI S.COOP.LTDA. reserves the right to make the necessary modifications to this manual without prior notice.

C O N T E N T S

1. INTRODUCTION

This manual contains the information required for operation of the CNC.

It describes the controls fitted on the front panel as well as their use and location.

Also the CNC operating modes and the information displayed on the screen are explained.

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

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.

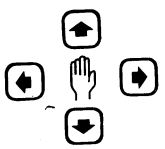
10. **P** : To access a program and to program parameters.

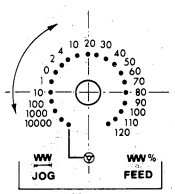
11. **N** : To access a block within a program and to identify subroutines.

12. **7 ... M** : Keyboard for program execution.

13. 

: Enables the programmed spindle speed % to be varied.

14.  : For shifting the machine axes manually.

15. 

: Knob enabling the programmed feedrate % to be varied and the different JOG modes to be selected (continuous, incremental and the manual pulse generator)

16.  : Cycle start button.

17.  : Cycle stop button. The CNC stops the execution of the block in progress, but maintains synchronism with the machine. To resume the program, press  .

18. 

: To reset the CNC. Accepting new values of machine-parameters, decoded **M** functions etc.

19.  : Decimal point or conditional block button.

3.1. 0 MODE: AUTOMATIC (Continuous cycle)
1 MODE: SINGLE BLOCK

The only difference between these two modes is that in single block mode (1), each time a block is executed the **ON** button has to be pressed to continue executing the program, whereas in automatic mode (0) the cycle is continuous.

3.1.1. Execution of a program

The execution of a program requires the following steps:

3.1.1.1. Selection of the AUTOMATIC operating mode (0) SINGLE BLOCK (1)

- Press **OPERATE MODE** : The list of 10 operating modes appears on the screen.

- Press the **0/1** key : The standard display corresponding to this operating mode appears; i.e. in the upper left-hand section of the screen the message **AUTOMATIC** followed by the number of the program P and the number of the first block to be executed N. More information appears which will be explained in section 3.1.2.

3.1.1.2. Selection of the program to be executed

Whenever a program number is wanted other than that appearing on the screen, the following sequence should be followed:

- Press the **P** key
- Key in the number of the desired program
- Press **RECALL**

The new program selected will appear on the screen, if it exists. If not, the screen will display: **N***

3.1.1.3. Selection of the first block to be executed

Once a program has been selected, the number of the first block to be executed appears to the right of the program number. If you wish to begin with a different block, the following procedure should be followed:

- Press the **N** key
- Key in the number of the block
- Press **RECALL**

The new number is displayed on the screen together with the contents of this block and those of the subsequent blocks.

3.1.1.4. Display of the contents of the blocks

To display the contents of the blocks prior or subsequent to those appearing on the screen:

- Press ⬆ : The previous blocks are displayed

- Press ⬇ : The later block are displayed

NOTE : The program always starts with the block whose number appears to the right of the program number, regardless of which ones are displayed on the screen.

3.1.1.5. Start cycle

- Press START

. Once the program and block number have been selected, just press this key to execute the program (in **AUTOMATIC**) or the block (in **SINGLE BLOCK**).

. If the program contains any **conditional block** it will be executed when the relevant input is activated (See INSTALLATION AND START-UP MANUAL. If it is not activated, the CNC will disregard such blocks.

3.1.1.6. Stop cycle

- Press STOP

The CNC stops the execution of the block in progress. To resume the cycle just press START .

The cycle is also stopped by means of:

- Code M00,M02,M30.

- Code M01 when the relevant input is activated.

- External signal **STOP**.

- The external signal **FEED HOLD** (the cycle continues when the signal disappears).

- The external signal **EMERGENCY STOP** (in this case the program must be restarted, since the CNC is reset to the initial state).

3.1.1.7. Changing the operating mode

At any time during the execution of the automatic cycle a transfer may be made to the **SINGLE BLOCK** operating mode. To do this:

- Press **OPERATE MODE**: The operating modes list will appear.
- Press **1/O**.

NOTE : If any number other than **1/O** is depressed, the CNC will revert to the **AUTOMATIC** mode (0).

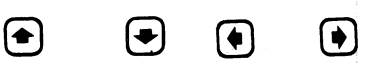
3.1.1.8. Tool inspection

If during the execution of a program you desire to inspect or change a tool, the procedure to be followed is:

a) Press 

The program being run is stopped and the blinking message **INTERRUPTED** is displayed at the upper right hand section of the screen.

b) Press **NEXT**. M05 is then executed except if graphic display mode is active (in this case ZOOM would be engaged). The screen will display:
JOG KEYS AVAILABLE
OUT

c) By pressing the jog keys  the axes can be moved to the desired point.

d) After the tool has been changed or inspected:

Press  (Depending on the situation when **NEXT** has been pressed, M03 or M04 will be executed).

The screen will display:
RETURN
AXES OUT OF POSITION
(Axes manually moved).

By means of the jog keys  , the axes are shifted to the point in which the cycle was interrupted.

The CNC will prevent the axes from going beyond this point.

When the axes reach the desired position:
RETURN
AXES OUT OF POSITION
NONE

will be displayed on the screen.

e) Press 

The cycle will continue.

3.1.2. Display modes

In **AUTOMATIC** or in **SINGLE BLOCK** there are seven display modes:

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

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)

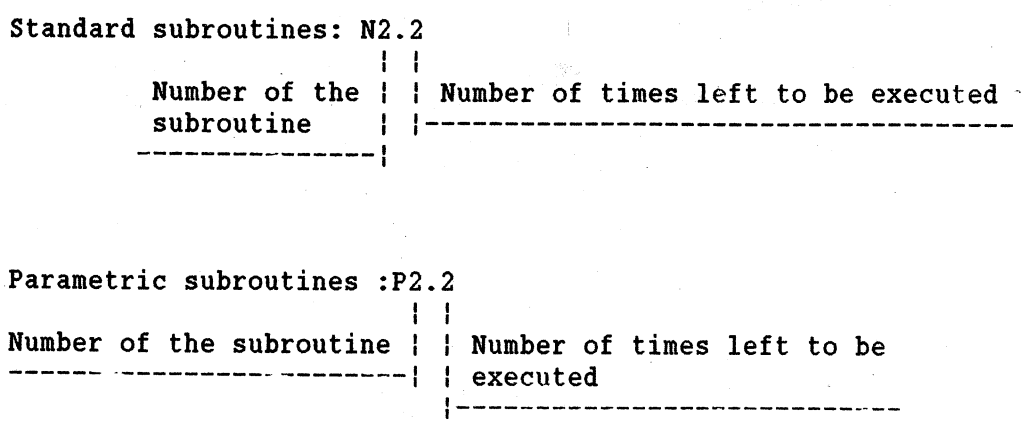
The following error of the **X,Z,C** axes is displayed, as well as the program number, the block number and the state of the **G,M,T,F** and **S**.

NOTE:

In the 0,1,2 display modes, the **C** axis will be displayed if machine-parameter 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:



THE CLOCK that indicates the time that the CNC has been executing programs since last turned on (in hours, minutes and seconds).

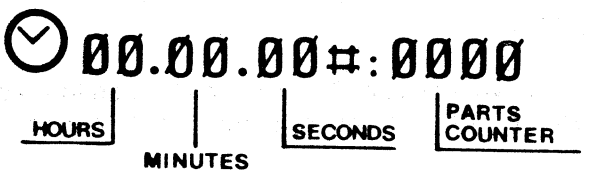
When the execution of a program is interrupted, the clock counting is also interrupted.

To the right of the clock, the parts counter (4 digits) is displayed. It is set to zero, when the CNC is turned on or, every time the number of the program to be executed is changed.

To reset the clock, press **DELETE** and then **T** when the clock is displayed.

The parts' counter increments one unit every time the functions **M30** or **M02** is read in the program.

To reset the counter, press **DELETE** and then **N** when the counter is displayed.



Being in the display mode (3), when **P** key is pressed, a list of 8 parameters with their relevant actual value will be displayed on the screen. If the keys ⬆ and ⬇ are pressed, the other 100 parameters with their values will be displayed. Example:

```
P46 = -1724.9281
P47 = -.10842021  E-2
```

NOTE: E-2 means 10 to the power of (-2).

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:

  Display modes 5 and 6 are used for graphics and will be described in section 3.10. of this manual.

3.1.3. Display and modification of the tool offsets without stopping the cycle

- Push **T**
- Key-in the desired tool offset number (01-32)
- Press **RECALL**

The value of the required tool offset will be displayed.

On the lower left part **I** will appear.

If the value of **I** is to be modified, the amount to be added or subtracted (always in diameters) must be keyed-in.

This value will be shown at the right of **I**.

If the value to be altered is that of **K**,

- Press **K**
- Key-in the value to be added or subtracted
- Press **ENTER**

The CNC assumes the modified values of **I,K** the next time the modified tool offset is programmed.

NOTE : To return to standard display mode after entering tool data, press **CLEAR**.

3.1.4. CNC reset

If the key RESET // is pressed, the blinking word **RESET?** is displayed on the upper right-hand corner of the CRT.

If this key is pressed again the CNC is set to power-on RESET // conditions.

3.2. PLAY-BACK

This method of programming is basically the same as the **EDITING** mode, except with regard to programming the values of the X,Z coordinates.

It allows the machine to be operated manually and the coordinate values reached to be entered as program coordinates.

The execution of a program requires the following steps:

3.2.1. Selection of the operating mode

- Press **OPERATE MODE**

- Press key **2** : The following information will appear on the screen:

```
   PLAY-BACK P -----
***AVAILABLE KEYS***
  *DISPLAY MODES (DIS MODE)
  *PROGRAM NUMBER (P)
  *NEXT
```

3.2.2. Locking/Unlocking of memory

Same as section 3.6.2. in **EDITING** mode (6).

3.2.3. Deletion of a complete program

Same as section 3.6.3. in **EDITING** mode (6).

3.2.3. Change of program number

Same as section 3.6.4. in **EDITING** mode (6).

3.2.5. Display of memorized subroutines

Same as section 3.6.5. in **EDITING** mode (6).

3.2.6. Selection of a program

Same as section 3.6.6. in **EDITING** mode (6).

3.2.7. Creation of a program

The creation of a program in **PLAY BACK** mode is the same as in **EDITING** mode except that the axes can be moved by means of the **JOG** keys. The axis coordinate values are displayed in the lower part of the screen.

In a block which only contains the coordinates of one point, after using the **JOG** keys to move the axes, press **ENTER** and the coordinates of the point will be stored in the memory.

If in addition to the coordinates of a point it is desired to write into the block further information such as G,S,M,T functions etc., each time the key of the corresponding axis is pressed the CNC will take as the value of the axis the coordinate at which the machine is at that moment. After editing the block, press **ENTER** to record it in the memory.

This method of editing is highly practical when creating a program for copying a part using functions G08 and G09.

When G08 has been written into a block requiring it, use the **JOG** keys to move the machine to the final point of the tangent arc to the previous path, then press **ENTER** and the block will be stored in the memory.

When G09 has been written into a block which requires it, use the **JOG** keys to move the machine to an intermediate point on the arc and press the **ENTER** key. The CNC will take the coordinates as those of the intermediate point on the arc. Then move the machine to the final point of the arc and once the **ENTER** key has been pressed the block will be stored in the memory.

3.2.8. Deletion of a block

Same as in **EDITING** mode (6).

3.3. TEACH-IN

This method of programming is basically the same as the **EDITING** mode, except that the blocks which are written may be executed before being entered into memory. This enables a part to be produced block by block while being programmed (See NOTE*).

The execution of a program requires the following steps:

3.3.1. Selection of the operating mode

- Press **OPERATE MODE**
- Press key 3: The following information will appear on the screen:

```
   TEACHIN P -----
***AVAILABLE KEYS***
  *DISPLAY MODES (DIS MODE)
  *PROGRAM NUMBER (P)
  *NEXT
```

3.3.2. Locking/Unlocking of memory

Same as section 3.6.2. in **EDITING** mode (6).

3.3.3. Deletion of a complete program

Same as section 3.6.3. in **EDITING** mode (6).

3.3.4. Change of program number

Same as section 3.6.4. in **EDITING** mode (6).

3.3.5. Display of memorized subroutines

Same as section 3.6.5. in **EDITING** mode (6).

3.3.6. Selection of a program

Same as section 3.6.6. in **EDITING** mode (6).

3.3.7. Creation of a program

Same as section 3.6.7. in **EDITING** mode (6) except that the block may be executed before pressing **ENTER**. To do this:

- Press  . The CNC executes the block.
- If it is correct, it may be recorded in memory by pressing **ENTER**.
- If it is incorrect, press **DELETE**. Then,
- Rewrite the block.

NOTE* :

- On pressing  , the CNC executes the block and the display mode changes to **AUTOMATIC** display mode.

- On pressing **ENTER** or **DELETE**, the display reverts to the **TEACH-IN** display mode.

- When the blocks are executed, the CNC retains the sequence of the completed blocks.

- Radius compensation cannot be performed in this mode.

- If a subroutine is called, the CNC will execute all its blocks.

3.3.8. Deletion of a block

Same as in **EDITING** mode (6).

3.4. DRY RUN

This operating mode is used for testing a program in a dry run before producing the first part.

3.4.1. Execution of a program

The execution of a program requires the following steps:

3.4.1.1. Selection of the operating mode

- Press **OPERATE MODE**
- Press key **4**. The screen will display:

DRY RUN

0 - G, FUNCTIONS
1 - G,S,T,M FUNCTIONS
2 - RAPID TRAVERSE
3 - THEORETICAL PATH
---

0 - G FUNCTIONS

The CNC will only execute the programmed **G** functions.

1 - G,S,T,M FUNCTIONS

The CNC will only execute the programmed **G,S,T,M** functions.

2 - RAPID TRAVERSE

The CNC will execute the program completely. The movements are executed at max. programmable feed **FO** regardless of the F's programmed.

The FEEDRATE knob allows the **%** of the feedrate to be varied.

3 - THEORETICAL PATH

The CNC will execute the program without moving the axes and without taking tool compensation into account.

3.4.1.1.1. Selection of execution mode

- Key-in the desired number (0,1,2,3).
  FINAL BLOCK:N
  Will be displayed at the bottom of the screen.

- Key-in the number of the last block whose execution in Dry Run mode is desired. If **0** is pressed, the program will be executed completely.

- Press **NEXT**
  The screen will display the same information as in **AUTOMATIC (0)** and **SINGLE BLOCK (1)**.

3.4.1.2. Selection of the program to be executed

Same as section 3.1.1.2.

3.4.1.3. Selection of starting block

Same as section 3.1.1.3.

3.4.1.4. Display of the contents of the blocks

Same as section 3.1.1.4.

3.4.1.5. Start cycle

Same as 3.1.1.5.

3.4.1.6. Stop cycle

Same as section 3.1.1.6.

3.4.1.7. Change of operation mode

At any time during the **DRY RUN** of a program a transfer may be made to operation modes **AUTOMATIC** and **SINGLE BLOCK**. To do this:

- Press **OPERATE MODE**: The operation modes list will appear.

- Press **0** or **1**.

NOTE: If any number other then **0** or **1** is depressed, the CNC will revert to the DRY RUN state.

3.4.1.8. Tool inspection

Same as section 3.1.1.8.

3.4.2. Display modes

Same as section 3.1.2.

NOTE: Regardless of the form of execution selected, the CNC will always examine the program as it executes it and will indicate possible programming errors.

If during the execution of a program in **DRY RUN** mode we change to **AUTOMATIC** or **SINGLE BLOCK** mode one more block is executed in **DRY RUN** mode before changing over to the mode selected. The programmed position of the axes is reached in the first **AUTOMATIC** or **SINGLE BLOCK** block.

3.4.3. CNC reset

Same as section 3.1.5.

3.5. JOG/ZERO SETTING

This operating mode is used for:

- Jogging the axes
- Searching for the machine-reference position of the axes
- Preselecting values on the axes
- Entering or executing **F,S,T** and **M**
- Operation as a readout
- Automatic loading of tool offset table
- Resetting the CNC
- Manual pulse generator operation.

The method of working in this operating mode is as follows.

3.5.1. Selection of the operating mode

- Press **OPERATE MODE**
- Press key **5**. On the screen will appear:

```
JOG/ZERO SETTING
X ---- . ---
Z ---- . ---
S ---- . T--
F0000% -- S0000%  T -- . --
M
```

The numbers to the right of **X,Z** being the value of the coordinates.

3.5.2. Search for machine-reference axis by axis

- Press the key corresonding to the axis to be referenced. In the lower left-hand part of the screen **X/Z** will appear according to the key pressed.

- Press **NEXT**. To the right of the axis letter will appear **ZERO SETTING?**.

- Press  . The axis will move at a rate of travel selected by means of machine-parameter towards the machine-reference position. On pressing the reference microswitch, it will change to a feedrate of 100 mm/min. On receiving the machine-reference pulse from the feedback system, it will stop, setting the counter to the value set as machine-parameter (P33,P73).
  If the reference microswith was pressed when pressing  , the axis will withdraw until the microswitch is released. Then the search will be carried out normally.

To cancel the machine reference search before pressing  , the **CL** key must be pressed. To cancel the search after pressing  ,  must be pressed.

3.5.3. Preselection of a cooordinate value

- Once displayed, press the key of the axis on which the preselection is required.

- Key in the required value.

- Press **ENTER**. The new value will appear on the screen.

To cancel the preselection, before pressing **ENTER**, operate **CL** as many times as characters to be deleted.

3.5.4. Jogging the axes

3.5.4.1. Continuous

- Front panel knob in any position of the % FEED zone.

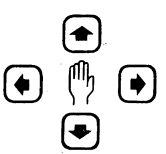
- According to the axis and the direction in which it is desired to move, the following should be pressed,

- As established by means of the machine-parameter:

  . (P12=Y). On releasing the key, the movement is stopped.
  . (P12=N). Two possibilities:

  - Press  to stop the movement.

  - Press



  To invert the direction or to move another axis.

NOTE:

On selecting the **JOG** operating mode the feedrate **F0** remains selected. If no other feedrate has been entered subsequently, the axes move at the % of **F0** indicated by the front panel knob.

3.5.4.2. Incremental

- Front panel knob in the **JOG** zone.
- Press any of the following keys:

The axis will move in the direction chosen, a distance equal to that indicated on the knob position.

NOTE:

a) Same as the NOTE for point 3.5.4.1.

b) The positions of the knob are 1,10,100,1000 and 10000 and indicate the value of the movement in microns or in 0.0001 inches.

3.5.5. Entering of F,S,M and T

The required values of **F,S,M** and **T** may be entered in this operating mode. The last three are affected by the values given to P97, bits 5,6 and 7.

3.5.5.1. Entering of an F value

- Press the **F** key
- Key in the required value
- Press

3.5.5.2. Entering of an S value

- Press the **S** key.
- Key in the required value
- Press 

3.5.5.3. Entering of an M number

- Press the **M** key.
- Key in the required number (between 0 and 99).
- Press 

3.5.5.4. Entering of a T number

- Press the **T** key.
- Key in the required number T2.2.
- Press 

3.5.6. Automatic measurement and loading of tool dimensions on the tool offset table

By using a part of known dimensions, the dimensions of the tools can be measured and loaded automatically in JOG mode. Machine-parameter P130 must be set to zero.

- Press **DISPLAY MODE**. The screen will display: **<T M>***
- Press **X**.
- Key in the parts dimensions along **X** axis (in radius or diameters accordingly.
- Press **ENTER**.
- Press **Z**.
- Key in the part's dimension along **Z** axis.
- Press **ENTER**.
- Key in the desired tool number (T2.2).
- Press **START**.
- JOG the X axis to touch the part.
- Press **X**.
- Press **NEXT**. The new **X** tool dimensions is then loaded in the table. The reading of the **X** axis must be the **X** value of the part.
- Jog the Z axis to touch de part.
- Press **Z**.
- Press **NEXT**. (Same as above for the **Z** axis.
- For subsequent tools, start the process by keying-in the new tool number (T2.2) and repeat the sequence described.
- To resume standard **JOG** operation, press **DISPLAY MODE**.

3.5.7. Operation of the CNC 8020 T as a readout

Once the **JOG** operating mode is selected, if the external **JOG** command is applied, the CNC acts as a readout. In this case, the machine has to be moved by means of external controls and the analog signals must be generated outside the CNC. The **S**,**M** and **T** functions may be entered in this form of operation, they are affected by the values given to P97, bits 5,6 and 7. If, when operating in this mode, the **software** travel limits (set via machine-parameters) are overrun, the CNC will send the relevant error code and will only allow the machine to be moved to bring it back to the permitted zone.

3.5.8. Change of measurement units

Every time, the key **I** is pressed the measurement units change from mm to inches and vice-versa.

3.5.9. CNC reset

Once the **JOG** operating mode is selected, when RESET is pressed, the CNC is reset to power-on conditions.

3.5.10. Manual pulse generator operation

With this option the axes can, one at a time, be shifted manually. For this:

- Select the **JOG** operation mode.
- Turn the front knob to ⊗ position.
- Press any of the two **JOG** keys which correspond to the axis to be moved by the Manual pulse generator. If a **FAGOR** Manual pulse generator (mod 100 P) is used, the axis can also be selected by means of the built-in selection pushbutton the relevant axis will be displayed (blinking) on the CRT.

- Turn the manual pulse generator. The axis will move according to the setting of the relevant machine-parameter.

To change the axis being operated:

- Press any of the two **JOG** keys of the new axis or the selection pushbutton if a **FAGOR** Manual pulse generator (mod 100 P) is used.
- Turn the manual pulse generator.

To end the manual pulse generator operation:

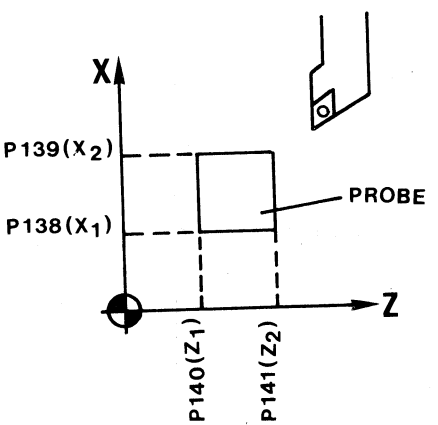
- Turn the switch to any other position.
- or press the **STOP** key.
- or keep the selection pushbutton pressed until the CRT stops displaying (blinking) the selected axis, if a **FAGOR** Manual pulse generator (mod 100 P) is used.

3.5.11. Measurement and loading of tool offsets with a probe

If machine parameter P130 is other than zero, the tool dimensions **X,Z** can be measured and loaded quickly in the **JOG** mode. To do this, a tool measuring probe must be installed with its sides parallel to the axes and in an established position on the machine.

The values of the sides of the probe on each axis and with respect to the machine reference zero must be entered in the following parameters:

P138 minimum value X1 in radius
P139 maximum value X2 in radius
P140 minimun value Z1
P141 maximum value Z2



The sequence to be followed is:

1- Press **DISPLAY MODE.** **<T M>*** will be displayed.

2- Select the tool number by keying in: **Txx.xx**

3- Move the tool with the **JOG** keys up to a position close to the probe side to be touched.

4- Press the axis key (**X** or **Z**).

5- Press the **JOG** key that indicates in which direction the axis must be moved to carry out the probing movement. The feedrate is determined by **P130.**

6- Once the probing is done, the machine stops and the CNC loads, in the corresponding position of the tool offset table, the value measured on **X** or **Z** setting the **I** or **K** value to zero.

7- Repeat steps from point **3** to do the same on the other axis.

8- Once the measured tool has been removed, repeat from step 2 to load the rest of the tools.

The feedrate override knob has no effect during the probing movements and is set to 100%.

The nose radius and location code values will be entered manually in operating mode 8 or by programming G50.

To go back to the **JOG** mode, press **DISPLAY MODE.**

3.6. EDITING

This is the fundamental operating mode for programming the CNC 8020 T. In this mode programs, subroutines as well as unconnected blocks may be written, amended and deleted.

The method of working in this operating mode is as follows:

3.6.1. Selection of the operating mode

- Press **OPERATE MODE**

- Press key 6. There appears on the screen:

```
   EDITING P -----
***AVAILABLE KEYS***
  *DISPLAY MODES (DIS MODE)
  *PROGRAM NUNBER (P)
  *NEXT
```

3.6.2. Locking/Unlocking of memory

- Press **DISPLAY MODE**. The following appears on the screen:

DISPLAY MODES

0 . CURRENT PROGRAM
1 . PROGRAM MAPPING
2 . STANDARD SUBROUTINE MAPPING
3 . PARAMETRIC SUBROUTINE MAPPING
4 . PROGRAM RENAME
5 . LOCK/UNLOCK MEMORY

- Press key 5. **CODE** will appear on the screen.

- Key in **MKAI1** to lock the part program memory.
  MKAIO to unlock the part program memory.

- Press **ENTER**.

NOTE:

a) In the event of keying in any code other than those indicated, on pressing **ENTER**, the said code will be erased, with the CNC waiting for the correct code.

b) Locking the memory implies not being able to alter the programs, but they can be displayed.

c) There are two ways to return to the state of section 3.6.1.:

- Press **DISPLAY MODE**
- Press the **0** key, or
- Press **OPERATE MODE**
- Press the **6** key

3.6.3. Deletion of a complete program

- Press **DISPLAY MODE**. The following appears on the screen:

  DISPLAY MODES

  0 . CURRENT PROGRAM
  1 . PROGRAM MAPPING
  2 . STANDARD SUBROUTINE MAPPING
  3 . PARAMETRIC SUBROUTINE MAPPING
  4 . PROGRAM RENAME
  5 . LOCK/UNLOCK MEMORY

- Press key **1**. A list of up to 14 programs in memory appears on the screen as well as the number of characters used and those remaining available.

- Press **DELETE**. The message **DELETE PROGRAM** appears on the screen.

- Once the **DELETE** key has been pressed if no program is to be deleted, press **OPERATE MODE** or **DISPLAY MODE**.

- If a program is to be deleted:

  Key in the number of the program to be deleted. Check the number.

- If the number is correct, press **NEXT**.

- If the number is not correct : . Press the **CL** key . We cancel the number with this key.

  . Key in the correct number.

  . Press **NEXT**.

DELETION OF ALL PROGRAMS

If all the programs in the memory must be deleted, key-in 99999 when **DELETE PROGRAM** is displayed, and then press **NEXT**; if the key **Y** is pressed inmediately afterwards, all the programs except the one identified by parameter **P125** will be deleted.

NOTE:

If there are more than 14 programs stored in memory, it may happen that the one to be deleted does not appear on the screen. In this case, by operating the keys ⬇ / ⬆ , the various programs may be moved back and forth until the desired program is displayed.

There are two ways to return to the state of section (3.6.1.):

- Press **DISPLAY MODE**
- Press the **0**

or,

- Press **OPERATE MODE**
- Press the **6** key

3.6.4. Change of program number

- Press **DISPLAY MODE**. The screen will display:

  DISPLAY MODES

  0 . CURRENT PROGRAM
  1 . PROGRAM MAPPING
  2 . STANDARD SUBROUTINE MAPPING
  3 . PARAMETRIC SUBROUTINE MAPPING
  4 . PROGRAM RENAME
  5 . LOCK/UNLOCK MEMORY

- Press **4**. The screen will display **OLD: P.**

- Key in the existing number of the program whose number is to be modified. It will be displayed to the right of **P.**

- Press **NEXT**. The screen will then display
  NEW: P

- Key in the new number allocated to this program. It will be displayed to the right of **P.**

- Press **NEXT**. The change of number has been completed.

  If there is no program recorded under the **old** number, the screen will display:

  PROGRAM NUMBER: P -----
  DOESN'T EXIST IN MEMORY

- If the new number is already allocated to another program, the control will display:

  ALREADY EXISTS IN MEMORY

  Press **DISPLAY MODE** or **OPERATE MODE** to continue.

3.6.5. Display and search of memorized subroutines

- By pressing 2 or 3 all the subroutines, standard or parametric, recorded in the memory are displayed.

- To find out which programs include the subroutines displayed on the screen, key in the subroutine number and press **RECALL**. To repeat the process for another subroutine, press **NEXT**.

3.6.6. Selection of a program

- If the number of the required program is the one which appears on the screen when the **EDITING** operating mode is selected, to obtain it just press **NEXT**.

- If a different program is wanted :
  - Press the **P** key.
  - Key in the program number.
  - Press **NEXT**. The program selected will appear on the screen.

3.6.7. Creation of a progam

If there is a program in the control's memory with the same number as the one to be recorded, there are two methods for recording the new program.

- Completely erase the existing program.

- Not to erase it and write it block by block (as described further on) over the existing program, taking care to allocate the same numbering as the previously recorded blocks to the blocks being written.

  If there is no other program in memory with the same number, proceed as follows:

3.6.7.1. Unassisted programming

Format of a block
(dimensions in millimeters) : N4 G2 X+/-4.3
Z+/-4.3 I+/-4.3 K+/-4.3
R+/-4.3 A+/-4.3
F4 S4 T2.2 M2 (in this order)

(dimensions in inches) : N4 G2 X+/-3.4
Z+/-3.4 I+/-3.4 K+/-3.4
R+/-3.4 A+/-4.3
F4 S4 T2.2 M2 (in this order)

Programming:

- The CNC automatically numbers the blocks in multiples of 10. If a different number is wanted, press **CL** and then, key in the block number. It will appear in the lower lef-hand part of the screen. The blocks may not be correlative.

- If a **conditional block** is desired, after keying-in the block number press **.** (decimal point).

  Write the **G** functions and coordinate values respecting the programming format.

- Press the **F** key and key in the feedrate value

- Press the **S** key and key in the spindle speed.

- Press the **T** key and key in the tool number.

- Press the **M** key and key in the number of the auxiliary function wanted. Up to a maximum of 7 may be programmed.

- If the block is correct, press **ENTER**. The CNC accepts the block as a program block.

  Refer to the programming manual for incompatibilities when programming various functions.

3.6.7.2. Modification and deletion of a block

I) During the writing process

a) Modification of characters

If during the writing of a block a character already written has to be modified:

- Use the ⬆ ⬇ keys to place the cursor on the character to be modified or deleted.
- To modify, simply key in the new character. To delete, press the **CL** key.
- If **DELETE** is pressed, the characters to the right of the cursor will be deleted.

b) Insertion of characters

If during the writing of a block a character has to be inserted within that block:

- Use the ⬆ ⬇ keys to place the cursor at the point where the new character is to be inserted.
- Press **NEXT**. The portion of the block that follows the cursor starts blinking.
- Key in the new character required.
- Press **NEXT**. The blinking stops.

II) Block already entered in memory

a) Modification and insertion of characters

- Key in the block number concerned.
- Press **RECALL**. The block appears in the lower part of the screen.
- Proceed as in the previous item.
- Press **ENTER**. The modified block is put into the memory.

b) Deletion of the block

- Key in the block number
- Press **DELETE**

If during the programming of a block, the CNC fails to respond to any key pressed, it means that there is something incorrect in what is being entered.

3.6.7.3. Assisted programming

Access to assisted programming is available in any of the program editing modes, i.e. **PLAY BACK (2)**, **TEACH-IN (3)** or **EDITING (6)**.

The procedure for accessing assisted programming is as follows:

During the writing of a block, pressing the **NEXT** key will make the cursor disappear from the block being written and the screen will display:

PROGRAMMING GUIDE

1 - MOVEMENT PROGRAMMING
2 - CANNED CYCLES
3 - SUBROUTINE/JUMPS
4 - GEOMETRICAL AIDS
5 - ARITHMETICAL FUNCTIONS
6 - G FUNCTIONS
7 - M FUNCTIONS
NEXT = END

Pressing the desired number will display pages which explain the various functions available to the CNC and how they are programmed. Once the appropriate page is accessed, press the **NEXT** key to continue writing the block. The cursor will reappear and the information required will stay on screen.

Supposing, for example, that when editing a program it is desired to program in a block the canned cycle for threadcutting (G86), the sequence will be:

Press **NEXT**
Press 2
Press ↓
Press 3

If the **NEXT** key is then pressed, the cursor will appear and it becomes possible to write the block, observing on the screen the meaning of the various parameters of the function selected.

When the writing of the block is completed, pressing **ENTER** stores the block in the memory and the standard display of editing modes will appear on the screen.

If, while any page of the assisted programming is on screen, it is desired to revert to the standard display mode, there are two possibilities:

a) When nothing is written in the block, press **RECALL** if the cursor is displayed (if it is not, press **NEXT**)

b) When some information is already written in the block, if the cursor is displayed, press **ENTER** or **DELETE**.

3.7. INPUT/OUTPUT

This is used for transferring part programs or machine-parameters from/to peripherals.

The method of working in this operating mode is as follows:

3.7.1. Selection of the operating mode

- Press **OPERATE MODE**
- Press key 7. The screen will display:

INPUT/OUTPUT

0 . INPUT FROM CASSETTE
1 . OUTPUT TO CASSETTE
2 . INPUT FROM GENERAL DEVICE
3 . OUTPUT TO GENERAL DEVICE
4 . CASSETTE'S DIRECTORY
5 . DELETE CASSETTE'S PROGRAM
6 . DNC ON/OFF

NOTE:

. To enable any of the operations 0,1,2,3,4 and 5, which are displayed in the INPUT/OUTPUT mode, to be carried out, point 6 (DNC ON/OFF) must be OFF (the highlighted message OFF will be displayed).

. If the highlighted message displayed is ON, press key 6. The CNC must be OFF when connecting/disconnecting peripheral units.

. When using the FAGOR cassette recorder, parameter P99(6) must be set to 0.

3.7.2. Entering a program from the FAGOR cassette recorder (0)

- Press the 0 key. The screen will display:
  PROGRAM NUMBER: P -----

- Key in the number of the program to be read in. If 99999 is entered, the CNC gets ready to accept machine-parameters, the decoded M's table and the table of leadscrew compensation parameters.

- Press **NEXT**. Four possibilities:

a) A program exists in the control's memory with the same number. The screen will display:
PROGRAM NUMBER: P -----
ALREADY EXISTS AT MEMORY
DELETE? (0/1)

If deletion is not wanted:
- Press **0**
- Return to the state in section 3.7.1.

If deletion is wanted:
- Press **1**. The screen will display:
  PROGRAM NUMBER: P ----- DELETED

From this moment the program starts to be transferred from the cassette, taking place as described in possibility c).

b) The program selected does not exist on the tape.

On starting to transfer from the cassette, if the program does not exist on the tape:

DOESN'T EXIST IN THE CASSETTE
TO GO ON PUSH NEXT/OPERATE MODE

- Press **NEXT**. It returns to the situation of section 3.7.1. or,

- Press **OPERATE MODE**. The operating modes list will appear.

c) The program selected exists on the tape and not in the control' s memory.

The transfer is carried out normally. During this process the screen will display: **RECEIVING**

- If in the program being read there is any erroneous block number (example, N xxxxx).

  PROGRAM NUMBER: P ----- READ
  DATA READ NON VALID
  N xxxxx
  TO GO ON PUSH NEXT/OPERATE MODE

  In this case only the part of the program up to the erroneous block is memorized. **¡It is recommended to delete the whole program!.**

- If the numbering of the blocks of the program transferred is correct:

  PROGRAM NUMBER: P ----- READ
  TESTING

That means that the CNC carries out a syntactic test of the program. If there is any programming error the relevant error code and the affected block are displayed and the program is recorded completely. If there is no error, the CRT will display.

TO GO ON PUSH NEXT/OPERATE MODE

- Press **NEXT**. It will return to the situation of section 3.7.1. or,

- Press **OPERATE MODE**. The operating modes list will appear.

d) If the part program memory is locked (or the machine-parameters memory in case of P99999), the state in section 3.7.1. is re-stablished.

3.7.2.1. Transmission errors

- If during transmission **TRANSMISSION ERROR** appears on the screen, this indicates that the transmission is not correct.

- If during transmission **DATA READ NON VALID** appears on the screen, this indicates that there is an incorrect character on the tape, or a non permitted block number has been written.

To continue working with the CNC, press **NEXT** or **OPERATE MODE**.

NOTE:

The lid of the cassette recorder should be open when turning the unit ON/OFF, to prevent tape damage.

3.7.3. Transfer of a program to the FAGOR cassette (1)

- Press key 1. The screen will display:
  PROGRAM NUMBER: P

  If P99999 is entered, the CNC gets ready to transmit machine-parameters and the decoded M's table, and the leadscrew error compensation table. Key in the number of the program to be transferred.

- Press **NEXT.**

Three possibilities:

a) The selected program does not exist in the control's memory. The screen will display:
   DOESN'T EXIST AT MEMORY
   TO GO ON PUSH NEXT/OPERATE MODE

   - Press **NEXT.** We return to the situation of section 3.7.1.
     or,

   - Press **OPERATE MODE.** The operating modes list will appear.

b) There is a program with the same number on the tape.

   On pressing **NEXT** the screen will display:
   ALREADY EXISTS IN THE CASSETTE
   DELETE ? (0/1)

   If deletion is not wanted:

   - Press 0. This returns to the state of section 3.7.1.

   If deletion is wanted:

   - Press 1. The screen will display:
     PROGRAM NUMBER: P -----
     DELETED

   From this moment, the program starts to be transferred to the cassette, taking place as described in possibility c).

c) The selected program exist in the control but not on the tape.

   The transfer takes place normally. During this process the screen displays **TRANSMITTING:**

   On completion the following text will appear:
   PROGRAM NUMBER P: ----- TRANSMITTED TO GO ON PUSH NEXT/OPERATE MODE

   - Press **NEXT.** This returns us to the situation of 3.7.1.
     or,

   - Press **OPERATE MODE.** The operating modes list will appear.

3.7.3.1. Transmission errors

Same as section 3.7.2.1.

3.7.4. Entering a program from a peripheral other than the FAGOR cassette recorder (2)

Same as section 3.7.2. (by means of an FAGOR cassette) except that the 2 key must be pressed and a new error message may appear: **MEMORY OVERFLOW**

This indicates that control's memory is full. The part of the program for which there was capacity will have been recorded in the control.

NOTE:

To enter a program from a peripheral other than the FAGOR cassette, the following points must be taken into account:

- The first thing that must be read after a series of **NUL** is a **%** followed by the program number (99999 indicates machine-parameters), followed by **LF**.

- The blocks are identified by an **N** located at the beginning of the line; i.e. inmediately after a **LINEFEED**. If anything is written between the **LINEFEED** and the **N**, this will not be taken as the indicator of the block number, but as an extra character.

- Spaces, the **RETURN** key and the + sign are not taken into account.

- The program ends with a series of more than 20 **NUL**, or with the character **ESCAPE** or **EOT**.

3.7.5. Transfer of a program to a peripheral other than the FAGOR cassette recorder (3)

Same as section 3.7.3. (by means of an FAGOR cassette) except that the 3 key is pressed.

The CNC ends the program with the character **ESC (ESCAPE)**.

3.7.6. FAGOR cassette's directory (4)

- Press the **4** key. The screen will display:

  . Number of programs on the tape with the number of characters.
  . Number of free characters on the tape.

- Pressing **NEXT** returns to the condition of section 3.7.1.

3.7.7. Deletion of a FAGOR cassette program (5)

- Press the 5 key. The screen will display:
  PROGRAMAM NUMBER: P -----

- Key in the number of the program selected.

- Press **NEXT.**

Once the program has been deleted, the screen will display:

PROGRAMAM NUMBER: P ----- DELETED
TO GO ON PUSH NEXT OR OPERATE MODE

- Press **NEXT.** The condition of section 3.7.1. returns or,

- Press **OPERATE MODE.** The operating modes list will appear.

3.7.8. Interruption of the transmission process

In this operating mode (INPUT-OUTPUT) any transmission process may be interrupted by pressing **CL.**

The screen will display:
PROCESSING ABORTED
TO GO ON PUSH NEXT OR OPERATE MODE

- Press **NEXT.** This returns us to the state of section 3.7.1. or,

- Press **OPERATE MODE.** The operating modes list will appear.

3.7.9. DNC. Communication with a computer

The CNC 8020 incorporates a **DNC** feature which allows two-way communication with a host computer to perform the following functions:

. Directory and program deletion commands
. Transfer of programs and tables
. Execution of infinite programs
. Machine's remote control
. Advanced DNC system's status report

To activate the DNC feature P99(5) must be **1.**

Also, **INPUT/OUTPUT** mode **6** must show the highlighted message **ON.** Otherwise press **6.**

See DNC manual for more detailed information.

When in **INPUT/OUTPUT** mode (7), if **RESET** is pressed the CNC will assume initial conditions.

3.8. G53/G59 ZERO OFFSETS AND TOOL TABLE

This is used for entering in memory the dimensions (length and radius) of up to 32 tools and the values of up to seven G53/G59 zero offsets.

The method of working in this operating mode is as follows:

3.8.1. Selection of the operating mode

- Press **OPERATE MODE**
- Press the 8 key. The screen will display:

TOOL OFFSET G53/G59

```
T01 - X ---- . ---  Z ---- . ---  F -
      R ---- . ---  I --   . ---  K -- . ---
T02 - X ---- . ---  Z ---- . ---  F -
      R ---- . ---  I --   . ---  K -- . ---
T03 - X ---- . ---  Z ---- . ---  F -
      R ---- . ---  I --   . ---  K -- . ---
```

3.8.2. Read-out of tool table

If a read-out is wanted of the dimensions of a tool which does not appear on the screen, there are two methods:

a) . Key in the number of the tool.
   . Press **RECALL**

b) Press ⬆ or ⬇ (located to the right of the screen) to move the tools displayed back and forth until the required tool is reached.

3.8.3. Entering the dimensions of the tools

- Key in the number of the tool. This will appear on the lower left of the screen.

- Press **X**.

- Key in the value of the length of the tool along X axis. Maximum value +/-8388.607 mm or +/-330.2599 inches.

- Press **Z**.

- Key in the value of the length of the tool along Z axis. Maximum value +/-8388.607 mm or +/-330.2599 inches.

- Press **F**.

- Key in the location code (0-9) of the tool.

- Press **R**.

- Key in the value of the tool radius.
  Maximum value +/-1000.000 mm or +/-39.3700 inches.

- Press **I**.

- Key in the X axis offset. This value has to be entered in diameters. It is used to compensate tool wear.
  Maximum value +/-32.766 mm or +/-1.2900 inches.

- Press **K**.

- Key in the **Z** axis offset.
  Maximum value +/-32.766 mm or +/-1.2900 inches.

- Press **ENTER**.

3.8.4. Modification of tool dimensions

I) During the writing process

a) Modification of characters.

If during the writing of tool dimensions a character already written has to be modified (X,Z,F,R,I,K or a number).

- Use the ⬆ ⬇ keys to place the cursor on the character to be modified or deleted.

- To modify, simply key in the new character. To delete, press the **CL** key.

- If **DELETE** if pressed, the characters to the right of the cursor will be deleted.

b) Insertion of characters

If during the writing of tool dimensions a character has to be inserted within that block:

- Use the ⬆ ⬇ keys to place the cursor at the point where the new character is to be inserted.

- Press **NEXT**. The portion of the block that follows the cursor starts blinking.

- Key in the new characters required.

- Press **NEXT**. The blinking stops.

II) Tool dimensions already entered in memory

- Key in the desired tool number.
- Press **RECALL**.
- Proceed as in the previous item.
- Press **ENTER**. The modified dimensions are entered into the memory.

If during the writing process the CNC fails to respond to any key pressed, it means that there is something incorrect in what is being entered.

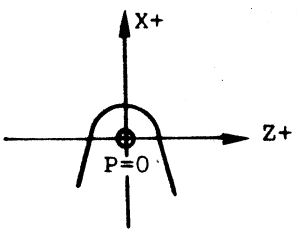
- Tool dimensions already written can be completely erased by pressing **DELETE**, if the cursor is situated at the beginning of the block.

3.8.5. Change of measurement units

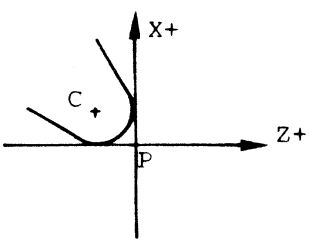
Every time the key I is pressed the measurement units change from mm to inches and vice-versa.

LOCATION CODES

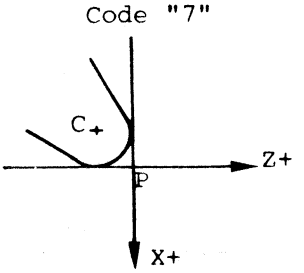
Code "0" and "9"



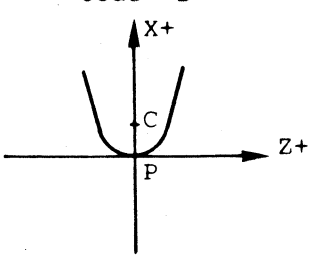
Code "1"

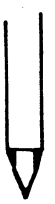


Code "7"

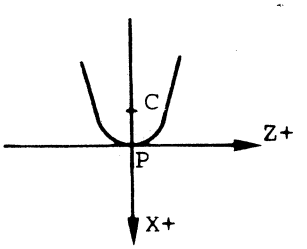


Code "2"

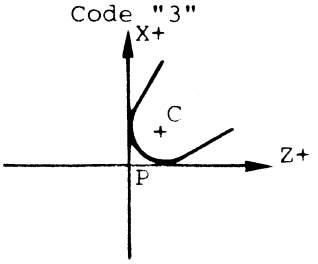




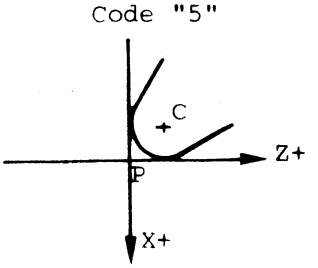
Code "6"



Code "3"

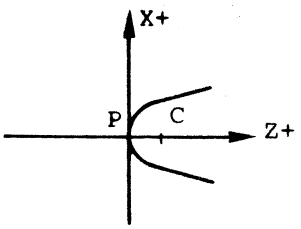


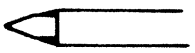
Code "5"



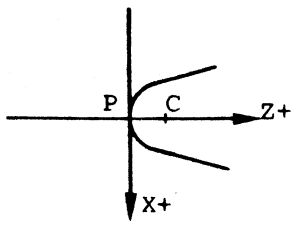
P: Tool tip
C: Tool nose center

Code "4"

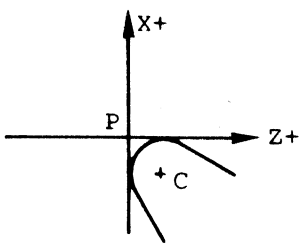




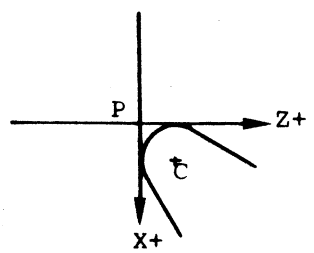
Code "4"



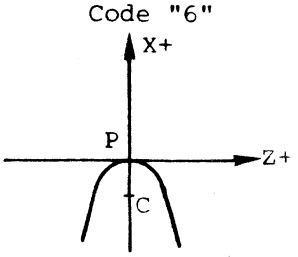
Code "5"



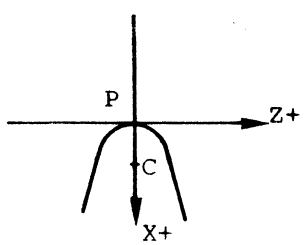
Code "3"



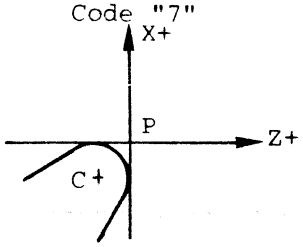
Code "6"



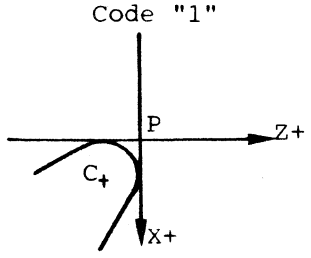
Code "2"



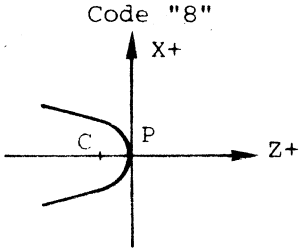
Code "7"

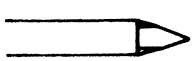


Code "1"

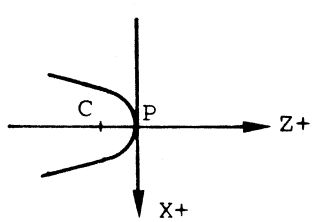


Code "8"





Code "8"



X-12.061 2.63.645

3.8.6. G53/G59 Zero offsets

In this same operating mode (8), pressing key G displays on the screen:

G53/G59 TOOL TABLE

```
G53  X ---- . ---  Z ---- . ---
G54  X ---- . ---  Z ---- . ---
G55  X ---- . ---  Z ---- . ---
G56  X ---- . ---  Z ---- . ---
G57  X ---- . ---  Z ---- . ---
G58  X ---- . ---  Z ---- . ---
G59  X ---- . ---  Z ---- . ---
```

3.8.6.1. Entering the values of the zero offsets

- Key in the offset number (G53-G59)
- Write the X,Z values desired
- Press **ENTER**

NOTE:

The **X,Z** values are referred to the machine reference zero point.

3.8.6.2. Modifying the values of the zero offsets

Same procedure as in section 3.8.4.

3.8.6.3. Change of units of measurement

Same procedure as in section 3.8.5.

3.8.7. Accessing the tool table

When the origin transfer table is displayed, pressing key **T** will make the screen revert to the tool table.

3.8.8. Total deletion of zero offsets or tool table

- Key **K,A,I.**
- Press **ENTER.**

The table displayed (tools or zero offsets) is totally deleted.

In mode 8 (G53-G59 tool table), press **RESET** to revert the CNC to initial conditions.

3.9. SPECIAL MODES

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

3.10. GRAPHICS

With this feature, 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**.

3.10.1. Display area definition

Prior to the representation of graphics on the CRT, the display area must be defined. Select **DISPLAY MODE** 5 after selecting the desired operation mode.

Press **1 or 0** to display or not the tool path.

If **1** has been pressed, key-in the coordinate values (X and Z) of the point desired to be at the center of the screen, and the width of the image. Press **ENTER** after every value.

The display area definition is lost when the CNC is turned **OFF**.

Key **6** to axes the graphics after the area has been defined.

Then, execute the program; the position and size of the graphic will depend on the values given to the center point and width.

Using the **FEEDRATE** override, the graphic drawing speed can be altered.

The coordinate values of the point being displayed are shown at the top of the CRT. The value of the width is displayed at the bottom.

3.10.2. Zooming (windowing)

Entire graphics or parts of them can be enlarged or reduced by this feature. To use this function the program must be either interrupted or completed. If **NEXT** is pressed in one of these two situations, a rectangle identifying the window will be displayed over the existing graphic.

Its dimensions can be altered pressing ⬆ or ⬇ and its position by using keys **8**,**2**,**4** and **6**.

The coordinate values of the window's center are displayed at the top of the CRT. The width and the percentage, at the bottom. The display of there values can be useful to check the coordinate values of a particular point (by placing the center of the window over it) and also to measure distances between two points.

If **5** is pressed, the windowed area will fill the CRT.

Using the **FEEDRATE** override knob, the graphic drawing speed can be altered.

To repeat the whole sequence, start by pressing **NEXT**.

To exit the zoom mode and continue, press **CL**.

3.10.3. Redefinition of the display area by zooming

With the zoom function active after pressing **NEXT**, if the **ENTER** is pressed, the position and width of the rectangle override the previous values given to the display area when it has been defined.

The position and the size of the graphic can thus be altered.

NOTE:

It is recommended that a sufficiently large width is assigned to the display area the first time it is defined to guarantee that the complete graphic will appear on the screen and then zoom in to center and enlarge it.

3.10.4. Deletion of graphics

Press **DELETE** to erase the graphic displayed, once the program has been executed or interrupted.

NOTE:

The CNC stores information on the last 500 blocks (involving movement) executed. If the program has more than 500 movement blocks only the last 500 will be displayed in zoom mode.

4. ERROR CODES

Code	Meaning
001	N is not the first character of a block. If running the execution of a program, while another program is being **modified** (background programming), a subroutine located in the program being modified or in a subsequent program, is called from the program in execution. The programs are stored as they are entered. Their order can be seen in **Program mapping.** The program being executed is always shifted to the first position. If during the execution of a program, another program is edited with a number not previously stored in the memory, this situation cannot arise.
002	Too many digits to define a function.
003	Negative value (or parameter) assigned to a function which cannot accept minus sign. Incorrect value given to a canned cycle parameter.
004	Calling a canned cycle from a inadequate position.
005	Incorrectly written parametric block.
006	More than 15 parameters affected in a block.
007	Division by zero.
008	Square root of a negative number.
009	Too great a value assigned to a parameter.
010*	The range or the S value, when working on constant surface speed, has not been programmed.
011	More than seven M functions in the same block.
012	. Function G50 improperly programmed. . Tool dimensions too large. . G53/G59 values too large.
013	Canned cycle not properly defined.
014	An incorrect block has been programmed, which is either incorrect by itself or incorrect in relation to the sequence of the program up to that point.
015	Functions G20, G21, G22, G23, G24, G25, G26, G27, G28, G29, G30, G31, G32, G50, G51, G53/G59, G72, G74, G92 or G93 I, K do not stand alone in a block.
016	The subroutine or block called for does not exist or the block looked for with the function **F17** does not exist.
017	Too high or negative threadcutting pitch.

018 Wrong definition when a point is defined by angle plus angle or angle plus cartesian value.

019 After defining G20,G21,G22 or G23 there is no subroutine number to which it refers or too many nesting levels. N is not the first character after G25,G26,G27,G28,G29. Too much mutual internesting.

020 More than one spindle range programmed in a particular block.

021 There is no block in the address defined by the parameter assigned to F18,F19 or F20,F21 or F22.
X axis is not programmed in the block addressed by the parameter assigned to F18.
Z axis is not programmed in the block addressed by the parameter assigned to F19.
I is not programmed in the block addressed by the parameter assigned to F21.
K is not programmed in the block addressed by the parameter assigned to F22.

022 Any one of the axis is repeated when programming on G74.

023 K has not been programmed after G04.

024 Decimal point missing in formats T2.2 or N2.2.

025 Error in a block defining or calling a subroutine or a subprogram, or a jump.

026 Memory capacity overrum. Available tape or CNC memory too small for the program to be stored.

027 I/K not defined in circular interpolation or threadcutting.

028 A tool in the table has been defined as having a number greater than Txx.32 or an external tool greater than the maximum defined by machine parameter.

029 Too high a value in a function 4.3 or 3.4. This code is often generated when an F value is first programmed in mm/min. and then switched to mm/rev. without changing the F value.

030 A non-existent G has been programmed.

108** Error in Z axis leadscrew error compensation parameters.

110** Error in X axis leadscrew error compensation parameters.

NOTE: Errors identified by an asterisk (*) deactivate enables and cancel the analog outputs.

Errors identified by two asterisks (**); besides deactivating enables and cancelling analog outputs, they activate the **EMERGENCY** output, setting the CNC to initial conditions.

Errors 094,095**,096**,098**,099**,108**,110** are originated by a **CHECKSUM ERROR** in their relevant area (tool-table, parameters, M-table, leadscrew compensation parameters).

The **CHECKSUM ERROR** usual cause is a battery malfunction, generating a loss of the recorded values. To reset the normal operation.

. Enter again the tool-table, machine-parameter, the decode M functions table, the leadscrew compensation parameters, according to the code displayed by the CNC.

. Keep the CNC on for 4-5 hours to enable the battery to recover the proper voltage level.

WARRANTY

With service contra According to contract clauses.

Without service contract:

The equipment is under warranty for 15 months from factory delivery date.

This warranty covers both material and labor repair costs at FAGOR.

In case of repair at customer's workshop, any travel expenses are payable by the customer.

This warranty does not cover damages and malfunctions arised from causes not related to normal operation of the equipment, such as blows, poor assembling or handling by untrained personnel, etc.

IMPORTANT NOTE

AURKI S.COOP.LTDA. (in Spain) periodically offers CNC operating and programming courses. They are oriented mainly to those end-users who would like to obtain the maximum benefit from all the features that this CNC offers.

If interested, contact the Communication Department of AURKI S.COOP.LTDA. in Spain by mail or telephone: Phone No. 943 - 799511.