FAGOR

Aurki, S. Coop. Ltda.

8020 T

PROGRAMMING MANUAL

ATTENTION

SUBSTITUTE THE FOLLOWING PAGES FOR THE HEREWITH ATTACHED.

3.1. Parametric programming

It is also possible to program in a block any function by parameters, except the program number, the block number, G22 and G23 functions, in such a way that, when executing the block, the functions takes the current value of the parameter.

Combinations of fined values and parameters can be programmed in the same block, e.g.

N4 GP36 X37.5 YP13 FP10 S1500 TP4.P4 MP2

The CNC has 100 parameters (P00/P99).

3.2. Double meaning of some keys

It is possible to double the meaning of some keys, so that alphanumeric characters not included in the front panel can be edited, which also makes possible writing comments in the program blocks.

To do so, * is to be pressed as it is the useful gadget to carry the doubling out.

Once * has been pressed, a directory showing the available characters at that moment will appear in the screen:

+	*	/
(	)	!
=	,	?
B	D	E
H	J	L
C	O	Q
U	V	Y
CL	W	SP

The directory will appear according with the front panel keys, e.g.

- . 4 means positive character
- . 6 means / character
  and so on.

If CL is pressed, the directory disappears from the screen and the meaning of the keys is the one on them indicated.

If comments are wanted to be written (are to be written), they should be placed between brackets ( ).

The maximum number of characters, including the brackets, that can be written in a comment is 43 and should be written at the block end, e.g.: N4 G-- X-- F-- M-- (COMMENT)

When pressing W, meaning SP, a space is maintained among characters of the comment.

6. PREPARATORY FUNCTIONS

The preparatory functions are programmed by means of the letter G followed by one or two digits (G2).

They are always programmed at the start of the block and are used to determine the geometry and operating state of the CNC.

6.1. Table of G functions used in the CNC 8020 T

(Modal)	G00	: Rapid positioning
(Modal)	G01*	: Linear interpolation
(Modal)	G02	: Clockwise circular interpolation
(Modal)	G03	: Counter-clockwise circular interpolation
	G04	: Dwell, duration programmed by means of K
(Modal)	G05	: Round corner
	G06	: Circular interpolation with absolute programming of arc's center
(Modal)	G07*	: Square corner
	G08	: Arc tangent to previous path
	G09	: Arc programmed by two points (3 point arc definition)
(Modal)	G14	: Activate C axis in degrees
(Modal)	G15	: Machining of the cylindrical surface of the part
(Modal)	G16	: Machining of the face of the part
	G20	: Call for standard subroutine
	G21	: Call for parametric subroutine
	G22	: Definition of standard subroutine
	G23	: Definition of parametric subroutine
	G24	: End of subroutine
	G25	: Unconditional jump
	G26	: Conditional jump/call if it equals zero
	G27	: Conditional jump/call if it differs from zero
	G28	: Conditional jump/call if it is smaller
	G29	: Conditional jump/call if it is equal or greater
	G30	: Display error code defined by K
	G31	: Store present program's datum point
	G32	: Retrieve datum point stored by G31
(Modal)	G33	: Threadcutting
	G36	: Controlled corner rounding
	G37	: Tangential approach
	G38	: Tangential exit
	G39	: Chamfering
(Modal)	G40*	: Cancellation of radius compensation
(Modal)	G41	: Left hand radius compensation
(Modal)	G42	: Right hand radius compensation
(Modal)	G49	: Programmable FEED-RATE
	G50	: Loading of tool offsets by program
	G51	: Modification of offsets of engaged tool
(Modal)	G53-G59	: Zero offsets

6.7. G14,G15,G16. C axis programming

These functions are only available in the TS model.

- G14. Activate C axis in degrees.
- G15. Machining of the cylindrical surface of the part
  (main plane C,Z)
- G16. Machining of the face of the part
  (Main plane C,Z)

Once the typical turning operations are completed, other operations, like the milling of the cylindrical surface and/or the face of the part can be done with the CNC 8020 T since it can control the machine's main leadscrew carrying out linear interpolations between the C,X,Z axes.

6.22. G75. Probing

G75 prepares the CNC to receive the signals coming from a measuring probe.

Format: N4 G75 X+/-4.3 Z+/-4.3

The axes will move until the probe signal is received. The CNC will then consider the block to be completed and the real position of the axes will be stored as theoretical position.

Neither the feedrate will be changed by turning the **FEEDRATE** knob (frozen at 100%) nor the movement of the axes will be displayed until the probe signal is received.

If the axes arrive in position before the probe pins the part the CNC will give error code 65. After executing this block, the values of the different axes can be allocated to parameters.

The combination of this feature with mathematical operations with parameters allows the creation of special subroutines to measure parts or tools.

The CNC assumes functions G01 and G40 after a G75 block.

Control models 8020 TS, TG and T allows the manual measuring and loading of tool dimensions with a probe. Refer to the **OPERATING MANUAL**.

Moreover, Control model TS can execute the probing cycles described below.

6.22.1. G75 N2. Probing canned cycles

The CNC 8020 TS offers various probing canned cycles to measure tool and part dimensions.

The programming format is as follows:

G75 N* P?=K-- P?=K--

The figure after N defines the probing cycle to be executed.

The CNC's probing canned cycles are:

N0: Tool calibration
N1: Probe calibration
N2: Part measurement in X axis
N3: Part measurement in Z axis
N4: Part measurement in X axis and tool correction in X axis
N5: Part measurement in Z axis and tool correction in Z axis

After N*, the calling parameters P?=K? must be programmed.

P1: Theoretical X value
P2: Theoretical Z value
P3: Safety distance
P4: Probing feedrate
P5: Tolerance
P6: Table number of the tool to be calibrated

- If any parameter that corresponds to a cycle is not programmed, the CNC will assume the latest value assigned to that parameter. The cycles do not modified the calling parameters (which can be used in later cycles) but do alter the contents of parameters P70 thru P99.

- P1 must be programmed in radius or diameters depending on the setting of machine parameter P11.

- Parameters P3 and P5 must always be programmed in radius.

- Parameter P3 must be greater than zero.

- Parameter P5 must be equal or greater than zero.

- Error 3 will be issued if one of these two conditions are not met.

6.23. G76. AUTOMATIC BLOCK GENERATION

This feature is only available in the 8020 TS model.

This function is used to generate blocks that are automatically loaded into the CNC or to a computer (via DNC).

If the new program is going to be loaded into the CNC, a block of the type G76 P5 must be previously written.

But if the new program is to be sent directly to a computer a block of the type G76 N5 must be previously written.

Once G76 P5 or G76 N5 executed; each time that the CNC executes any block containing G76, it will load whatever is after G76 into the new program.

The programming format is:

N4 G76 (contents of the block to be created).

The contents of the block to be created are similar to the normal programming except that the preparatory functions G22 and G23 cannot be programmed.

After G76, the coordinates can be programmed in different ways:

a) X+/- 4.3 Z+/-4.3
   Loads the axes with the indicated values.

b) X Z
   Loads the axes with the theoretical values that they show at this time.

c) XP2 ZP2
   Loads the axes with the values of the parameter at this time.

Example:

```
N10 G76 P00345
N20 G76 G1 X F500 M3
N30 P2=P3 F2 K1
N40 G76 XP2 ZP5 M7
N50 G76 G0 X14 Z20 M5
```

and if in block 40 the parameter values are: P2=14.853 and P5=154.37, the CNC will generate the following program P00345.

```
N100 G1 X78.35 F500 M3
N101 X14.853 Z154.37 M7
N102 G0X14 Z20 M5
```

b) <u>SYNCHRONIZED TOOL (with the rotation speed of the spindle)</u>

To take advantage of this feature, an encoder must be conected to the live tool. This encoder will be connected to the model CNC 8020 TS via the feedback connector A2.

Programming format: N4 M45 K+/-3.4

The sync factor is determined by K and may be between K0 and K+/-655.3509 times the spindle speed in one direction (+) or the other (-).

When K is a fraction, it is recommended to use parametric programming to gain accuracy.

Example:

To program K=1/3, if M45 K0.3333 is programmed, a lesser accuracy will be obtained than if the following is programmed:

```
N - P1=K1 F4 K3
N - M45 KP1
```

If the rpm's are greater than the limit established by machine-parameter P126, the CNC will display error 17. Also, when the following error of the synchronized tool is too large, the CNC will display error 71.

Nothing else can be programmed in the block whether is format a) or b).

To stop the rotation of the live tool in either case M45 S0 or only M45 must be programmed.

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 programed 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. Il 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 replace. 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.

N E W   F U N C T I O N S

G06. Circular interpolation in absolute coordinates by programming the centre of the arc

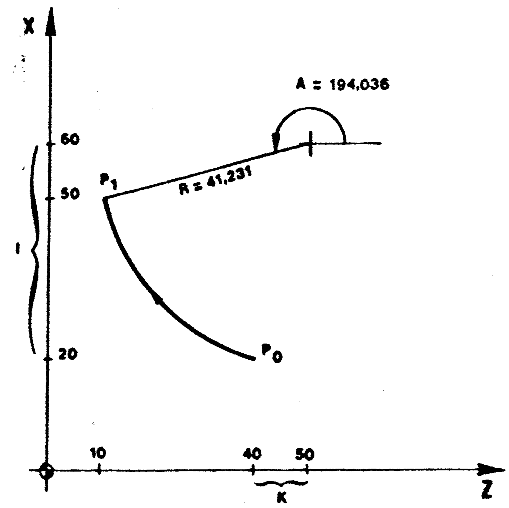
The coordinates of the center of the arc (I,K) can be programmed, in absolute coordinates, by adding the function G06 to a block of circular interpolation, in other words, relating the starting point and not the beginning of the arc.

The function G06 isn't modal, therefore, it is to be programmed whenever the coordinates of the arc center are to be shown in absolute coordinates.

When programming in such a way, the value of I is to be in radius or diameters referred to the parameter P11.

Examples: Let us suppose the programming is in absolute coordinates (G90) and the axis coordinate is in diameters.

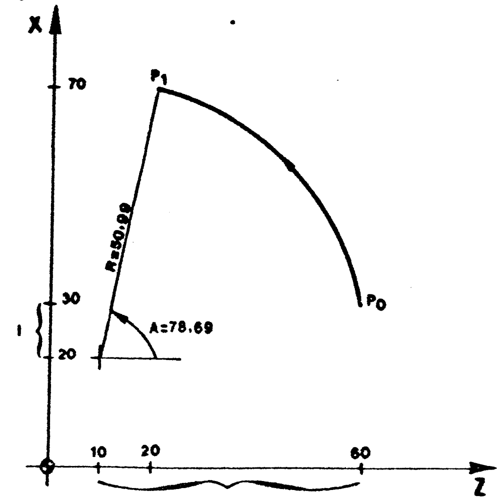
1



Starting point P0 (X40 Z40)

Cartesian coordinates	: N4 G02 X100 Z10 I40 K10
Polar coordinates	: N4 G02 A194.036 I40 K10
Radius programming	: N4 G02 X100 Z10 R41.231
Programming by G06	: N4 G02 G06 X100 Z10 I120 K50

2



Starting point P0 (X60 Z60)

Cartesian coordinates	: N4 G03 X140 Z20 I-10 K-50
Polar coordinates	: N4 G03 A78.69 I-10 K-50
Radius programming	: N4 G03 X140 Z20 R50.99
Programming by G06	: N4 G03 G06 X140 Z20 I40 K10

G49. Programmable **FEED-RATE**

It is possible to indicate, per program, the F programmed feed-rate at which desired to work by function G49.

When G49 is active, the M.F.O. switch is unabled.

The programming format is: **G49 K (1/120)**

After G49 K, the F feed-rate required is programmed. An entire value can be programmed between 1 and 120.

G49 is modal. In other words, once the rate is programmed, it is kept until either programming another one or canceling the function. To cancel the function G49 K ( ), either G49 K0 or G49 are only to be programmed.

The function G49 is also canceled when **M02,M30,RESET,EMERGENCIA** are applied.

G49 K is to be programmed on its own, in the block.