

CNC

8037 .M.

New features

Ref.1710

Soft: V02.3x



FAGOR AUTOMATION



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; *utelnnet* 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

VERSION V02.31

1. Punching press graphics.....	5
2. Synchronization of the axes movement with the feed hold	8
3. Set the machine coordinate (G174)	9
4. Anticipation of M functions	10
5. Tap withdrawal once the network has been reestablished	11
6. Tool withdrawal during shutdown	13
7. Save screen by pressing [SHIFT] + [Page Up]	16
8. Working with two additive handwheels simultaneously.....	17
9. CNC8055 (client) and PC (server) connection	19
10. Expansion of zero offsets to 40.....	20
11. General machine parameter CODEPAGE (P197)	21
12. Anticipated spindle stop	22
13. Deletion of temporary files on the hard disk.....	23
14. PANDRAW variable for grinding cycles	24

VERSION V02.33

1. Different accelerations for G00 and G01.....	25
2. Select handwheel movements in radii or diameters, when the axis is in diameters	26
3. CNCDISSTAT variable.....	26



FAGOR AUTOMATION

CNC 8037

SOFT: V02.3x



FAGOR AUTOMATION

CNC 8037

SOFT: V02.3X

VERSION V02.31

1 Punching press graphics

Using the new graphic representation of the punching press, when a punching program is run, the CNC displays the path, the points where the punch strikes and the shape of the punch. The punching press graphics work during both simulations and actual execution.

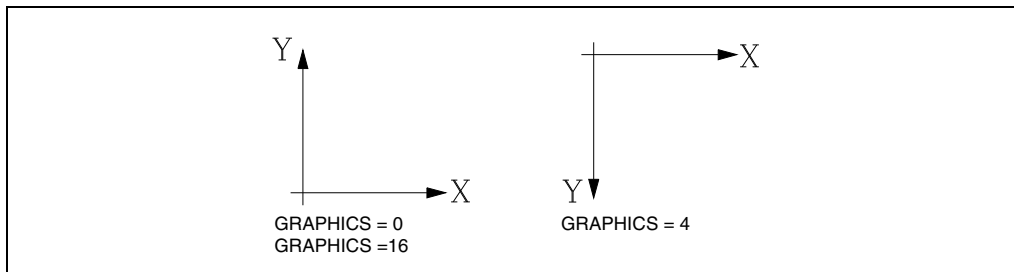
The punching press graphics are activated through the general machine parameter GRAPHICS (P16).

General machine parameter GRAPHICS (P16)

On M, MC, and MCO models, this parameter indicates the axis system being used for the graphic representation (milling machine graphics or punching press graphics), as well as the motion possibilities for the W axis added to those of the Z axis in the graphic representation (W additive).

Value	Meaning
0	Mill graphics.
1	Mill graphics with added W axis.
2	Boring Mill graphics.
3	Boring Mill graphics with added W axis.
4	Mill graphics (changed line graphics).
16	Punching press graphics

Default value: 0



When the punching press graphics are activated, the following options will be available:

- Show paths /G0 /G1 /G2 /G3.
- Define the sheet dimensions.
 - [GRAPHIC PARAMETERS] / [PUNCHING PRESS]
- Softkeys to define the area to be displayed.
 - [SHEET AREA]: Fixed zoom of the sheet dimensions.
 - [OPTIMUM AREA] Zoom showing the area of the points where the punch strikes.

Editing a punch profile

Each corrector is associated with a profile with the punch shape. When accessing the table of correctors, an image of the profile of each punch is displayed in the lower right corner.

The [EDITAR PERFIL] softkey in the menu can be used to begin editing the punch profile. When this softkey is pressed, the profile editor opens with the name of the program containing the profile.

The punch profile must be closed and there can only be one single profile. If an attempt is made to save more than one profile, a warning message will appear saying that the profile is not correct.



If you wish to work in millimeters, the punches must be edited in millimeters and the general machine parameter INCHES (P8) must have a value of 0.

If you wish to work in inches, the punches must be edited in inches and the general machine parameter INCHES (P8) must have a value of 1.

After editing a punch profile, when exiting the editing application, a program is created in which the punch shape is saved. This program will be saved in the directory "PunchOffsetProfile" and it will be called "PUNCHP-XXX.PIM", where XXX is the corrector number.

If any of the profiles is changed through FTP or by completing a Restore, the punch shape will be updated during the start-up of the CNC. The program will be on the hard disk, in the "PunchOffsetProfile" directory.

A Backup/Restore will be able to be run on the "PunchOffsetProfile" directory so that all the profiles can be copied from one CNC to another.

Punch profile display:

The center of the tool (punch) is the point 0,0.

It will be possible to rotate the punch around its center. The rotational axes will be defined in the general machine parameter PUNCHROT (P194).

Activating the strike point display

The PLC mark GRAPOSEN (M5080) is used to activate the strike point display.



It is recommended to activate the PLC mark GRAPOSEN (M5080) in a subroutine associated with the M that activates the punching. This way, the mark will also be activated in the G functions simulation mode and the simulation of the graphics will be seen in this mode.

Once the mark is active, every time the axis reaches the programmed position, the shape of the punch will be drawn.

Example:

```
(SUB 7771)
(PLCMM5080 = 1)
(RET)
;
(SUB 7772)
(PLCMM5080 = 0)
(RET)
```

M function table

```
M50 S7771 00000000
M52 S7772 00000000
```

Note:

The following M functions activate and deactivate the punching:

M50: Activates the punching

M51: Deactivates the punching

PLC machine parameters and marks

PUNCHROT (P194)

General machine parameter that indicates the rotational axis of the punch when punch press graphics are used:

Value	Meaning	Value	Meaning
0	None.	5	V axis.
1	X axis.	6	W axis.
2	Y axis.	7	A axis.
3	Z axis.	8	B axis.
4	U axis.	9	C axis.

Default value: 0

GRAPOSEN (M5080)

PLC mark that permits activating the display of the strike point on a punch press machine. When the punching mode is activated, this mark must have the logic value of 1, and when it is deactivated the logical value must be set to 0.

When the mark is active, each positioning is considered a strike.

2 Synchronization of the axes movement with the feed hold

Its use is recommended with punching presses so that the delay that occurs between the time when the feed hold signal raises and the axes begin to move is always the same.

Synchronization activation

To activate synchronization, set bit 2 of the general machine parameter IPOTIME (P73)=1.

IPOTIME (P73)

This parameter has 16 bits counted from right to left.



Each bit has a function or work mode associated with it. By default, all the bits will be assigned the value of ·0·. Assigning the value of ·1· activates the corresponding function.

Bit	Meaning	Bit	Meaning
0		8	
1		9	
2	Activates the synchronization of the axes with the feed hold.	10	
3		11	
4		12	
5		13	
6		14	
7		15	

Default value in all the bits: 0



FAGOR AUTOMATION

CNC 8037

·M· MODEL
SOFT: V02.3x

3 Set the machine coordinate (G174)

Function G174 may be used to set the machine coordinate of an axis; in other words, temporarily set a new machine zero. The new machine zero remains active until the axis is homed; the CNC then restores the original machine reference zero (set in the machine parameters).

After executing function G174, the CNC assumes that the programmed coordinate defines the current position referred to machine reference zero (home). The zero offsets, movements with respect to machine zero, etc. will be referred to the coordinate programmed in G174.

Programming the function

Program function G174, and then the machine coordinate of a single axis. With this function, only the machine coordinate of an axis may be set; to set the machine coordinates of several, program one G174 for each one of them.

When it comes time to setting the machine coordinate, the CNC uses the predefined unit system in the control. If it is a linear axis, use millimeters or inches, as defined in general machine parameter INCHES (P8). If it is a rotational axis, use degrees. The CNC ignores all the other options, radius/diameter, mirror image, scaling factor, etc.

The active zero offsets before G174 remain active and now refer to the new machine coordinate.

Programming format:

The programming format is as follows:

```
G174 X..C
```

```
X..C      Machine coordinate of the indicated axis.
```

Example:

```
G174 X100
```

Considerations and limitations

By itself, function G174 does not cause any axis movement. After executing function G174, the CNC considers that the axis is homed and verifies that it is within the software travel limits.

The normal use of this function is on rotational axes without limits, which always turn in the same direction.

The CNC does not allow setting the machine coordinate on slaved axes, grantry, tandem or on axes that are part of the active kinematics or active transform. It is also not permitted to set the machine coordinate on the C axis of the lathe or on axes with encoded I/Os. Before setting the new machine coordinate, the CNC checks that the axis is in position and it is not synchronized; if this is not the case, it issues an error message.

When executing G174, if there is any active coordinate transformation (G47, G48, G49, etc.), the CNC will issue an error.

It is possible to use the function G174 from the PLC channel and from the user channel.

Properties of the function and Influence of the reset, turning the CNC off and of the M30 function

Function G174 is modal. The new machine zero is unaffected by either function M02 or M30, or by a reset, an emergency or by shutdown of the CNC. On power-up, the CNC assumes the machine coordinates that were active when the CNC was turned off.



FAGOR AUTOMATION

CNC 8037

-M- MODEL
SOFT: V02.3x

4 Anticipation of M functions

The M functions anticipation feature may be used to transfer an M function to the PLC before the previous movement ends. This feature is very useful in punching presses. In these machines, it allows the next punching to be prepared from the PLC before the previous movement ends.

Definition of the M functions to be executed in advance

The table of auxiliary M functions has an 8-bit field for customization.

To define the M functions that will be executed in advance, use bit 5 of the desired M functions customization. The time by which these M functions are anticipated is indicated in the general machine parameter MANTIME (P196).

Customization bit 5 from the M auxiliary function table

Indicates whether the M auxiliary function is executed in advance.

Value	Meaning
0	The M auxiliary function is not executed in advance.
1	The M auxiliary function is executed in advance.

MANTIME (P196)

General machine parameter that indicates the time in milliseconds by which the M auxiliary functions are anticipated that are indicated by means of customization bit 5 from the M auxiliary functions table.

Possible values

Integers between 0 and 65535 ms.

Default value: 0 (not executed in advance)

Considerations and limitations

M functions can be anticipated in G5, G7 and G50, but they cannot be anticipated in G51.

Only those M auxiliary functions are anticipated that are executed from the main channel. The M functions that are executed from the PLC channel are not anticipated.

Only those M auxiliary functions are anticipated that do not have a predefined meaning for the CNC. The following M functions are not anticipated:

M0, M1, M2, M3, M4, M5, M6, M8, M9, M19, M30, M41, M42, M43, M44 and M45.

An M auxiliary function is only anticipated if there are no other low level blocks (F, G, etc.) between the previous movement block and the M function block.

The M functions that are anticipated must be programmed individually in the block; they cannot be programmed together with other M, S or T functions. Otherwise, the CNC will display the error: "The M function must be programmed by itself in the block".

The M functions that are anticipated may be programmed in the movement blocks. If the M function is customized to be executed after the movement block, the combination (movement - punching M) may be programmed in the same block.

The anticipation of the M functions only occurs in execution mode. M functions are not anticipated in any of the simulation modes.



If there are filters with set parameters on the axes, the anticipation time is greater than that indicated in the general machine parameter MANTIME (P196). In this case, in order to ensure the correct functioning of the feature, it will be necessary to set this parameter.

5 Tap withdrawal once the network has been reestablished

If a shutdown or an error occurs or the CNC is reset during the execution of a tapping (G84), it will be possible to remove the tool via an instruction executed in MDI.

This feature works with all types of taps (G84):

- R0 - Tapping with a clutch.
- R1 - Rigid tapping.
- R2 - Rigid tapping without spindle orientation

Operation

After a shutdown, reset or error on the machine, during the execution of a tapping, the high level instruction TAPRETRACT instruction will make it possible to remove the tool from the threading.

The instruction TAPRETRACT may be executed in MDI in both ISO mode and in conversational mode. In addition, it can also be executed by a button or pushbutton programming a CNCEX1 from the PLC.

When the withdrawal is executed, the longitudinal axis will be moved in the opposite direction of the threading that was active, until the starting plane. If there was no starting plane programmed before the threading, the tap is withdrawn to the reference plane indicated in the cycle.

The high level instruction TAPRETRACT will carry out the same withdrawal for any type of G84 threading.

If the instruction TAPRETRACT is executed in the following cases, the corresponding error will be issued:

- A tool withdrawal has been previously executed with the instruction TAPRETRACT.
- An attempt has been made to execute the maneuver without the necessary conditions being present (interruption of the execution of the G84 threading cycle due to shutdown, error or reset).

Considerations:

After a shutdown, reset or error in the machine during the execution of a rigid threading, during the withdrawal maneuver, the programmed F and S will not be affected by the withdrawal feedrate factor (J). During the withdrawal, only the factor established in the machine parameter TAPREOVR (P195) will be applied.

Slow speed tool withdrawal

In cases in which the S programmed in the threading is very high, it is possible to reduce both the programmed S and the F so that the withdrawal is carried out at a slower speed.

The withdrawal speed is changed in the percentage function indicated in the general machine parameter TAPREOVR (P195).

TAPREOVR (P195)

According to the percentage indicated in this general parameter, the withdrawal speed of the tool after a shutdown or error during a threading with a tap is changed.

Value	Meaning
0	The tool withdrawal will be carried out at 100% of the programmed F and S value.
1 .. 120	The tool withdrawal will be carried out according to the programmed F and S percentage indicated.

Default value: 0

The CNC displays the new F and S values proportional to the programmed TAPREOVR (P195).

This parameter is only applied when a withdrawal is carried out through the high level instruction TAPRETRACT.

Example:

Threading cycle values: F500 y S500.

General machine parameter TAPREOVR (P195) = 10.

Taking into account the indicated values, when the withdrawal instruction is carried out, the withdrawal will be completed at F50 and S50.

Tool compensation during the withdrawal

If G43 (tool length compensation) is active during the threading process and a shutdown or error occurs, this G43 will be accounted for during the withdrawal, so that it is performed correctly, without exceeding the limits.

Once the withdrawal is finished, G43 will remain active or inactive, depending on the general machine parameter ILCOMP (P12).

- If ILCOMP (P12) = 0, the length compensation is canceled.
- If ILCOMP (P12) = 1, the length compensation remains active.

PLC marks

There are two PLC marks for the tap withdrawal maneuver carried out through the high level instruction TAPRETRACT.

TAPACFAIL (M5577)

The value 1 of this mark indicates that it was performing a rigid tapping and there was a power drop. The value 0 of this mark indicates that the process of removing the tool has finished.

This mark can also be set to 0 when homing or setting the PLC RESTAPAC mark to 1.

RESTAPAC (M5081)

Through this PLC mark, it is possible to disable the tool withdrawal to be carried out through the instruction TAPRETRACT. The machine manufacturer may determine the conditions in which this feature is disabled.

When this mark has a value of 1, the PLC TAPACFAIL (M5577) mark is set to 0. This way, the withdrawal instruction is disabled.



FAGOR AUTOMATION

CNC 8037

-M- MODEL
SOFT: V02.3x

6 Tool withdrawal during shutdown

If a power drop occurs during the machining of a part, the tool or part may be damaged if the tool is in contact with the part. To prevent this, the CNC will carry out a tool withdrawal procedure.



This feature only works on vertical milling machines, since the withdrawal is carried out on the Z axis. In addition, it only works with the CAN servo system and ACSD drives. The drive version must be V2.04.

CNC variables

ACFDISX..Z

Distance the indicated axis moves when writing this variable from the PLC. This variable is incremental and is programmed in millimeters. This variable may be written from the PLC and read from the DNC and CNC.

PLC marks

ACFAIL (M5576)

CNC output that is set to the logic value 1 when it is detected that there will be a power drop.

ACFMOV1 (M5118)	ACFMOV2 (M5168)	ACFMOV3 (M5218)	ACFMOV4 (M5268)
ACFMOV5 (M5318)	ACFMOV6 (M5368)	ACFMOV7 (M5418)	

CNC input that indicates to the drive connected to the axis that it must not activate the emergency stop when a power drop is detected.

Diagram of the electrical cabinet

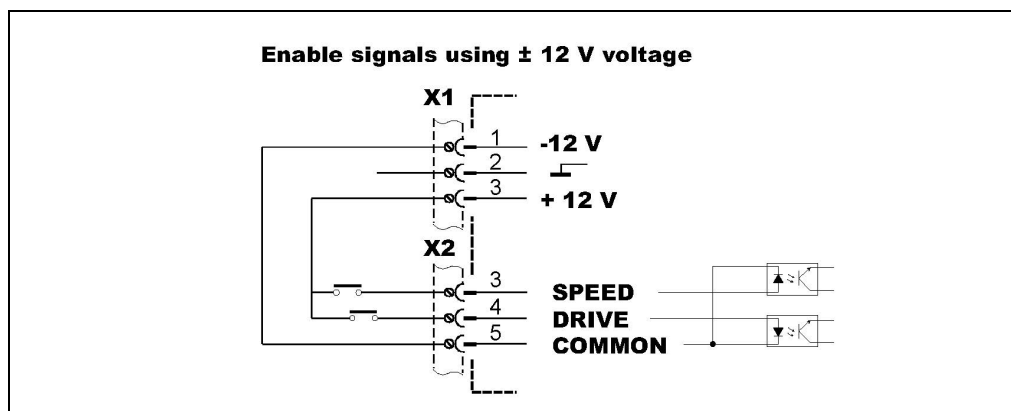
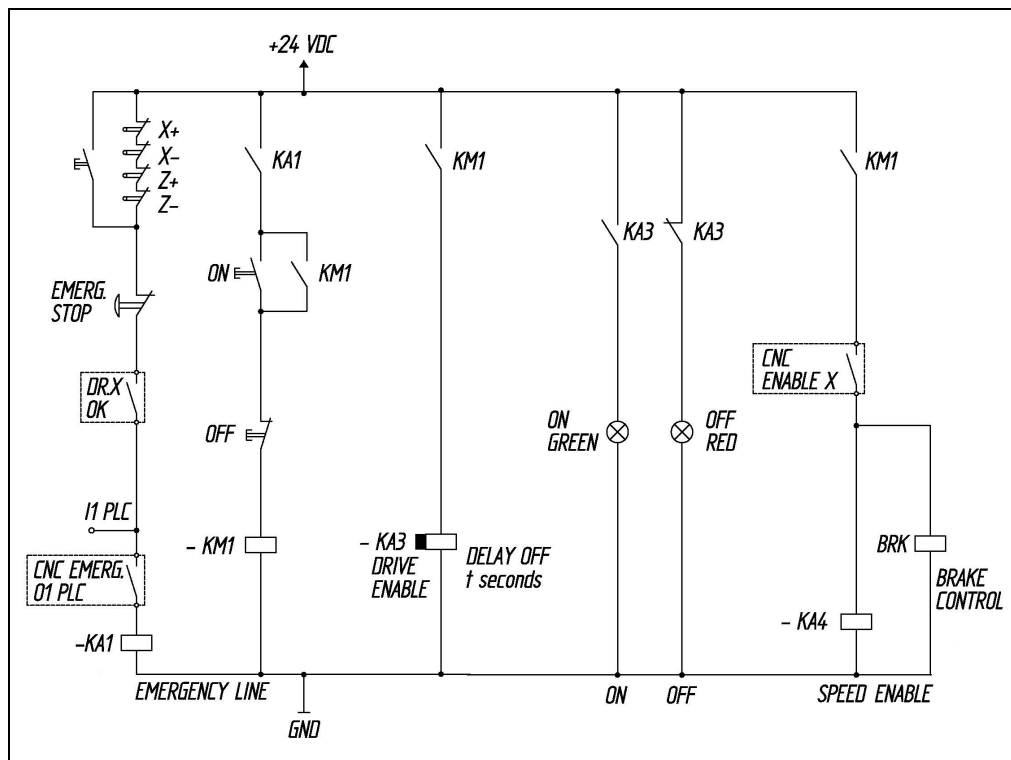
Diagram that shows the wiring maneuver for the physical signals SPENA and DRENA that are connected to the drive:

- The signals SPENA and DRENA remain at 24 V for 300 to 400 ms after the power drop.
- The 24 V of the logic of these signals must be supplied by the 24 V output of the drive.
- Timers must be connected so that the physical signals SPENA and DRENA remain active for a time at 24 V after a power drop.



When completing the setup, it is recommended to set the physical signals SPENA and DRENA that are wired to the drive continuously to 24 V.

Once verified that it functions correctly, wire these signals to the timers.



FAGOR AUTOMATION

CNC 8037

-M- MODEL
SOFT: V02.3x

PLC programming example

```

DEF MACFAIL M3000
DEF MACFAIL_SPZ M3001
DEF MTRETRACT M3002
CY1
() = SET MACFAIL_SPZ
END
;
;
PRG
; ----- Sercos STATUS -----
;
;
DRSTAFZ = B1R102          ; Z axis drive digital status
DRSTASZ = B0R102          ; Z axis drive digital status
;
CPS R102 GE 1
= M402                    ; Z axis drive ready
;
;
; ----- Axis enable -----
;
;
CPS R102 GE 2          ; Z axis Power ON
= M410                ; Axis drive with Power
DFU M410
= SET MACFAIL_SPZ
;
( M410                ; Machine with power
AND NOT LOPEN )      ; CNC Loop Open
OR
( MACFAIL_SPZ AND MTRETRACT ) ; ACFAIL Allow SPENA and tool retract
= SERVOZON          ; Z axis loop closed
= SPENAZ            ; Z axis speed enable ON
= TG3 13 200        ; Timer axis DRENA
= TG3 14 100        ; Timer Z axis brake
T14 = NOT O7        ; Z brake output;
T13                 ; Timer axis DRENA
= DRENAZ            ; Z axis Drive enable ON

; Tool retract only in program execution
; Do not tool retract with the emergency button.
; Do not allow tool retract while threading
I1 AND INCYCLE AND NOT TAPPING AND NOT RIGID AND NOT THREAD
= MTRETRACT
; Activate tool retract only when torque on in the driver Z
( MTRETRACT AND DRSTAFZ AND DRSTASZ ) OR T307
= ACFMOVZ            ; Z axis will not do an emergency stop when power cut
;
; Detect Power cut
DFU ACFAIL AND ACFMOVZ
= CNCWR(10000,ACFDISZ,M1) ; Move Z axis up 1mm when power cut
= TG1 307 200        ; 200 mseg after speed enable and drive enable will be deactivated
; and the brake will be set.
T307 = MACFAIL
DFD MACFAIL = RES MACFAIL_SPZ ; Reset drive speed enable
END

```



FAGOR AUTOMATION

CNC 8037

·M· MODEL
SOFT: V02.3x

7 Save screen by pressing [SHIFT] + [Page Up]

When the key sequence [SHIFT] + [Page Up] is pressed, an image of the currently active screen will be saved in the CNC.

If a USB memory drive (Pendrive) is connected when the screen is saved, the image will be saved in the <PAN> directory of said memory. If the <PAN> directory does not exist on the USB memory drive, it will be created automatically.

If there is no USB memory drive (Pendrive) connected when the screen is saved, the image will be saved in the <PAN> directory of the hard disk (KeyCF) of the CNC.

The saved image can be sent by DNC or FTP.

Image format:

The image format will be ".bmp" and the name of the saved file will be the following:

S month day hour minute second.bmp (no spaces in the file name).

Example of a saved screen:

Name of a screen saved on October 30, 2015 at 9:32 and 50 seconds:

S1030093250.bmp



FAGOR AUTOMATION

CNC 8037

·M· MODEL
SOFT: V02.3x

8 Working with two additive handwheels simultaneously

This feature makes it possible to operate while moving two additive handwheels at the same time.

Parameter setting

The general machine parameters from AXIS1 (P0) to AXIS7 (P8) and from AXIS9 (P136) to AXIS12 (P142) must have a handwheel defined (value of 11 or 12) and a handwheel associated with an axis (values of 21 to 29).

The general handwheel is associated with the axis defined in the general machine parameter MPGAXIS (P76). In additive handwheel mode, the flywheel moves only the axis indicated in the general parameter MPGAXIS (P76).

To enable this feature, assign value 1 to bits 10, 11 and 15 of the general machine parameter ADIMPG (P176). The value of all bits in this parameter will be as follows:

ADIMPG (P176) = 1000 1100 0000 0000.

ADIMPG (P176)

This parameter enables manual intervention with an additive handwheel.

This function allows jogging the axes while a program is being executed. This movement will be applied as if it were another zero offset.

This parameter has 16 bits counted from right to left.



Each bit has a function or work mode associated with it. By default, all the bits will be assigned the value of ·0·. Assigning the value of ·1· activates the corresponding function.

Bit	Meaning
0 - 9	Not used.
10	Working with two additive handwheels simultaneously.
11	Selecting the additive handwheel as handwheel associated with the axis
12	The resolution of the handwheel is set by g.m.p. ADIMPRES.
13	Manual intervention enabled with look-ahead.
14	Cancel the additive offset after M02, M30, emergency or Reset.
15	Manual intervention with additive handwheel is available.

Default value in all the bits: 0



As of this version, in order for the CNC to accept a new value for the general machine parameter ADIMPG (P176), it is necessary to press the keystroke sequence [SHIFT] + [RESET] or shut down the CNC and then turn it back on.



FAGOR AUTOMATION

CNC 8037

·M· MODEL
SOFT: V02.3x

Considerations

In JOG movement mode, the only active handwheel is that indicated with value 11 in its corresponding parameter (general machine parameters AXIS1 to AXIS12). To move an axis, first select the axis and then move it with the handwheel.

In automatic mode, the handwheels will behave as handwheels associated with an axis. The user may move 2 handwheels at the same time. To do this, the PLC MANINT* marks must be activated.

For example, to move the X and Z axes with the two handwheels, the PLC marks MANINTX and MANINTZ must be activated.

Parameter setting example:

Define an electronic and mechanical handwheel in general machine parameters P1 to P8.

AXIS1 (P0) = 1	Axis X.
AXIS2 (P1) = 2	Axis Y.
AXIS3 (P2) = 3	Axis Z.
AXIS4 (P3) = 10	Main spindle.
AXIS5 (P4) = 11	General handwheel.
AXIS6 (P5) = 23	Handwheel associated with the Z axis.
MPGAXIS (P76) = 1	Associate the handwheel with the Z axis. When the CNC is in automatic mode, when the handwheel is moved, the Z axis will move.
ADIMPG (P176) = 1000 1100 0000 0000	



FAGOR AUTOMATION

CNC 8037

-M- MODEL
SOFT: V02.3x

9 CNC8055 (client) and PC (server) connection

In addition to a local hard disk (on the CNC itself), the CNC can also have a remote hard disk accessible through Ethernet. The CIFS protocol is used to communicate with the remote hard disk.

As remote hard disk, it is possible to use the hard disk of a PC or just a folder. The PC that makes its hard disk (server) public must be connected to the local network.

Once communication has been established, the directory of the connected PC will appear in the CNC browser with the name "REMOTE DISK".

The interface and the softkeys of the CNC will be the same as if it were a local hard disk. The CNC directories cannot be seen from the PC.

Ethernet parameters

The following Ethernet parameters are available to configure communications via this protocol:

USER (P3)

Name of the user for connection to the CNC on the PC. If the parameter is not set, indicate that there is no user.

DOMAIN (P4)

Windows network domain. If the parameter is not set, indicate that there is no domain.

PASSWORD (P5)

User password for connection to the CNC on the PC. If the parameter is not set, indicate that there is no password.

IPSNFS (P28)

IP address of the server acting as remote hard disk. If other than 0, the remote hard disk is activated.

Possible values

Four numbers between 0 and 255 separated by dots.

Default value: 0.0.0.0 (there is no remote hard disk)

DIRNFS (P29)

Directory of the server that is used as remote hard disk.

Possible values

It admits up to a maximum of 22 characters (without blank spaces).

Default value: Nameless

NFSPROTO (P32)

To activate the CIFS communication protocol, set value as 2.

10 Expansion of zero offsets to 40.

G159 N1 to N40. Absolute zero offsets.

To apply any zero offset defined in the table.

The first six zero offsets are the same as programming G54 through G59, except that the values of G58 and G59 are absolute. This is because function G159 cancels functions G54 through G57 and, consequently, there is no active zero offset to add the G58 or G59 to.



FAGOR AUTOMATION

CNC 8037

·M· MODEL
SOFT: V02.3x

11 General machine parameter CODEPAGE (P197)

General machine parameter that enables extended ASCII characters to be displayed. This makes it possible to display those characters in the comments of the part programs, regardless of the language in which the CNC is configured through the general machine parameter LANGUAGE (P122).

To display extended ASCII characters, a value other than 9 must be assigned in this general parameter.

Furthermore, if this parameter is configured with a value of 9, an error will not be returned when executing a part program that contains extended ASCII characters in mainland Chinese.

CODEPAGE (P197)

Define the language in which to display the extended ASCII characters in the CNC.

Value	Meaning
0..9	Permits the display of extended ASCII characters.
9	An error is not returned when executing a part program with extended ASCII characters in mainland Chinese.
10..12	Permits the display of extended ASCII characters.

Default value: 9

12 Anticipated spindle stop

This feature permits moving up the spindle stop order. This is useful in laser cutting machines, in which the analog setpoint output of the spindle is used to regulate the laser power and it is necessary for the final movement to coincide with the time when the spindle stops (S=0).

SANTIME

CNC variable that permits the time, in milliseconds, to be programmed that the start of the spindle deceleration is anticipated before the end of the movement. This variable can be read and written from the CNC and the PLC, but only read from the DNC.

The variable SANTIME does not stop block preparation.

During the execution of a program, if a movement block is followed by an S0 block and the variable SANTIME has a value other than 0, the spindle stop order is advanced. The time of the anticipation will be that indicated in milliseconds in the SANTIME variable.

This variable will only be taken into account if G5 is active. In the rest of the cases, the spindle stop will not be advanced.



If an anticipation of the spindle stop is programmed and there are active filters, the anticipation will be greater than that defined in the SANTIME variable. In this case, the anticipation must be adjusted.

13 Deletion of temporary files on the hard disk

The new softkey [CLEAN DISK] makes it possible to delete the temporary files that have been created on the hard disk. To access this softkey, go to: DIAGNOSIS / TESTING / HARD DISK.

Example of files that are deleted when the softkey [CLEAN DISK] is pressed:

- Screens saved with [SHIFT] + [Page Up].
- Files with drive variables.



14 PANDRAW variable for grinding cycles

WINDRAW55 application. Number of the screen created by the user or the manufacturer and is being consulted.

Even if ESC is pressed (and then START is pressed and the cycle is executed) and the focus changes, the screen number remains.



FAGOR AUTOMATION

CNC 8037

·M· MODEL
SOFT: V02.3x

VERSION V02.33

1 Different accelerations for G00 and G01

The movements programmed using G00 (rapid feedrate) are executed using the rapid feedrate found in the axis machine parameter "G00FEED".

Two new axis parameters are available to define the acceleration and jerk used for the movements in G00 on each axis: ACCTIMG0 (P105) and JERKLIG0 (P106).

If the value of these two parameters equals 0, the CNC will operate in both G00 and G01, using the values of the axis parameters ACCTIME (P18) and JERKLIM (P67). In this manner, the G00 and G01 acceleration and jerk will be the same.

To achieve a different axis acceleration during the G00 operation, change the axis parameter value ACCTIMG0 (P105) to the desired value.

To achieve a different axis jerk during the G00 operation, change the axis parameter value JERKLIG0 (P106) to the desired value.

Axis machine parameters

ACCTIMG0 (P105)

Defines the acceleration stage or the time it takes the axis to reach the feedrate selected with axis parameter GOFFED (P38) when performing G00 movements. This time is also valid for the deceleration stage.

Possible values

Integers between 0 and 65535 ms.

Default value: 0

If the ACCTIMG0 (P105) and JERKLIG0 (P106) values equal 0, when performing a G00 movement, the acceleration will be that indicated for the axis machine parameter ACCTIME (P18).

JERKLIG0 (P106)

Defines the acceleration derivative for the G00 movements. It may be used to limit the acceleration changes to smooth the machine movements on small speed increments or decrements and with FFGAIN values close to 100%.

The CNC ignores this parameter when moving with electronic handwheels, mechanical handwheels, look ahead, threading (G33) and rigid tapping. If the ACCTIMG0 (P105) and JERKLIG0 (P106) values equal 0, when performing a G00 movement, the jerk will be that indicated for the axis machine parameter JERKLIM (P67).

The smaller the value assigned to JERKLIG0, the smoother the machine's response, but the acc/dec time will be longer.

Possible values

Between 0 and 99999.9999 m/s³.

Default value: 0

Recommended values:

In millimeters $JERKLIG0 = 82 * G00FEED / ACCTIMG0^{**2}$

In inches $JERKLIG0 = 2082 * G00FEED / ACCTIMG0^{**2}$

If the stability of the machine is affected by the values mentioned earlier, the JERKLIG0 value should be lowered to half as much.



FAGOR AUTOMATION

CNC 8055
CNC 8055i

-M- MODEL
SOFT: V02.3x

2 Select handwheel movements in radii or diameters, when the axis is in diameters

The bit 13 of the general machine parameter HDIFFBAC (P129) selects whether the handwheel movements and incremental jog are made using radii or diameters, when the axis coordinates are displayed in diameters.

HDIFFBAC (P129)

This parameter has 16 bits counted from right to left.



Each bit has a function or work mode associated with it. By default, all the bits will be assigned the value of ·0·. Assigning the value of ·1· activates the corresponding function.

Bit	Meaning	Bit	Meaning
0	Handwheel ·1·	8	
1	Handwheel ·2·	9	
2	Handwheel ·3·	10	
3	Handwheel ·4·	11	
4		12	
5		13	The movement of the axis using the handwheel will be performed in diameters.
6		14	Axis filters for movements with the handwheel.
7		15	It limits the movement.

Default value in all the bits: 0

Bit 13:

The bit 13 indicates whether the handwheel movements and incremental jog are made using radii or diameters, when the axis coordinates are displayed in diameters.

- (0) The movements are performed using radii.
- (1) The movements are performed using diameters.

3 CNCDISSTAT variable

The new variable CNCDISSTAT indicates the status of the CNC to execute a program.

- If the variable CNCDISSTAT is set to 0, program execution is permitted.
- If the variable CNCDISSTAT is set to a value other than 0, program execution is not permitted.

The manufacturer's PLC program may read this variable using the instruction CNCRD, so as to determine the CNC status.



FAGOR AUTOMATION

CNC 8055
CNC 8055i

·M· MODEL
SOFT: V02.3x



FAGOR AUTOMATION

CNC 8037

SOFT: V02.3X



FAGOR AUTOMATION

CNC 8037

SOFT: V02.3X



FAGOR AUTOMATION

Fagor Automation S. Coop.

Bº San Andrés, 19 - Apdo. 144
E-20500 Arrasate-Mondragón, Spain

Tel: +34 943 719 200
+34 943 039 800

Fax: +34 943 791 712

E-mail: info@fagorautomation.es
www.fagorautomation.com

