CNC 8070

Quick reference

FAGOR

FAGOR ◗

INDEX

SCREEN DESCRIPTION

General description of the work-mode screen
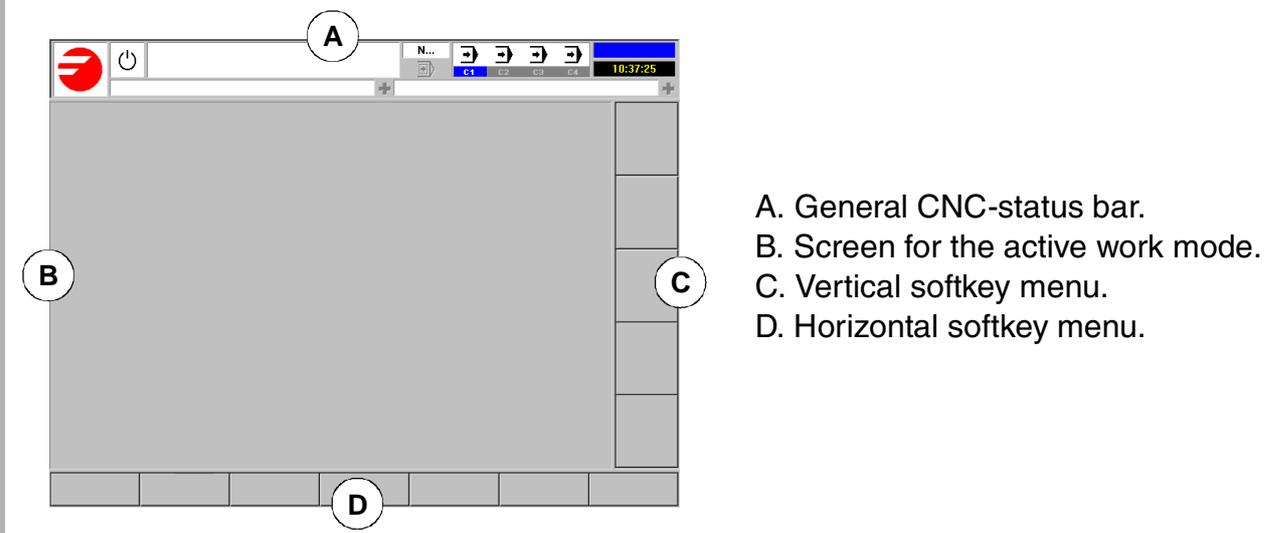
A. General CNC-status bar.
B. Screen for the active work mode.
C. Vertical softkey menu.
D. Horizontal softkey menu.

Detailed description of the CNC status bar
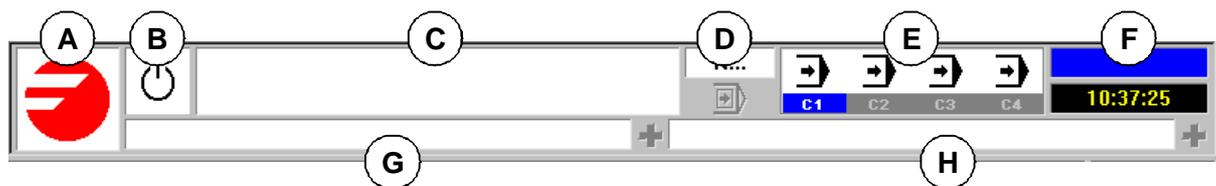
A. Icon (customizable) identifying the manufacturer.
B. Icon showing the status of the program of the active channel:
C. Program selected in the active channel for execution.
D. Number of the block in execution. The bottom icon indicates that the Single-block execution mode is active.
E. Information about the channels.
F. Active work mode (automatic, manual, etc.) selected screen number and total number of screens available.
G. Active CNC message.
H. PLC messages.

Turning the CNC off

[ALT] + [F4]   Turn the CNC off.

DESCRIPTION OF THE KEYS

Work modes

⊟	Automatic mode.	[CTRL] + [F6]
✋	Jog mode.	[CTRL] + [F7]
🖐	MDI mode.	[CTRL] + [F8]
✎	Editing - simulation mode.	[CTRL] + [F9]
●	User tables.	[CTRL] + [F10]
⬒	Tool and magazine table.	[CTRL] + [F11]
🔧	Utilities mode.	[CTRL] + [F12]
⬦	Configurable mode. It may be configured using machine parameter "USERKEY".	
	To show the modes window.	[CTRL] + [A]

Using the screens

↰	Previous menu.	[CTRL]+[F1]
⊞	Switch window.	[CTRL]+[F2]
📄	Key that may be customized by machine parameter.	[CTRL]+[F3]

Help

HELP ◇	It accesses the CNC's system help.

Execution keys

│	Cycle start key [START].	[CTRL]+[S]
◯	Cycle stop key [STOP].	[CTRL]+[P]
RESET	RESET key.	[CTRL]+[R]
⊡	"Single block" execution key.	[CTRL]+[B]
ZERO	Home search.	

Alphanumeric keyboard

←	Delete character (It deletes the character to the left of the cursor).
SUP	Deleted character.
INS	Insert / overwrite.
⇄	Tab.
ESC	Escape key.
ENTER ◇	Command validation.
RECALL ◇	[RECALL] key.

[CTRL]+[F5]

Feed selectors
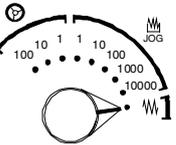
 Jog move type selector.

 Feedrate override % selector.

Operations in the user interface

[CTRL] + [W]	Minimize/Maximize the CNC.
[CTRL] + [J]	Show / hide the virtual operator panel.
[CTRL] + [M]	Show / hide the PLC message list.
[CTRL] + [O]	Show / hide the CNC message list.
[CTRL] + [K]	Show / hide the calculator.
[ALT] + [S]	Show / hide the channel synchronization window.

Program editor

[CTRL]+[TAB]	Toggle between the editor and the error window.
[CTRL]+[C]	Copy the selected text onto the clipboard.
[CTRL]+[X]	Cut the selected text.
[CTRL]+[V]	Paste the text that was "copied" or "cut".
[CTRL]+[Z]	Undo the last change.
[CTRL]+[G]	Save the program/Recover the original program.
[CTRL]+[HOME]	Move the cursor to the beginning of the program.
[CTRL]+[END]	Move the program to the end of the program.

🖐 MANUAL (JOG) MODE

JOG keys

The following keys may vary depending on the machine and the keyboard being used:

| X+ | 7+ | Jogging the axis in the positive direction. |

| X− | 7− | Jogging the axis in the negative direction. |

| 〰 | Rapid jogging of the axis. |

| X | 7 | Axis selection. |

| + | − | Selection of the moving direction. |

Home search

Manual (this operation will cancel the current part zero):

| X | Y | Z | Select the axis to be homed (using the alphanumeric keyboard). |

| ZERO | Press the home-search key. |

| | | ESC | Press [START] to carry out a home search.
Press [ESC] to cancel the operation. |

Automatic:

| ZERO | Press the home-search key. |

| | | ESC | Press [START] to carry out a home search.
Press [ESC] to cancel the operation. |

Coordinate preset

| X | Y | Z | Select the axis to be preset (using the alphanumeric keyboard). |

Key in the desired preset value.

| ENTER ◈ | ESC | Press [ENTER] to preset the entered value.
Press [ESC] to cancel the operation. |

Programmed movement:

| X | Y | Z | Select the axis to be moved.
Enter the coordinate of the target point. |

| | | ESC | Press [START] to make the move.
Press [ESC] to cancel the operation. |

Movement of the axes

Manual movement (jogging) of the axes using JOG:

    1. Continuous jog, (the axis moves while acting on the keyboard).
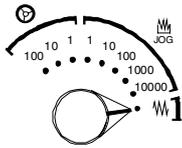
Turn the jog-type selector switch of the operator panel to the continuous jog position on the dial.
Jog the desired axis using the JOG panel (keypad).

    2. Incremental jog, (the axis moves a specific distance every time the operator acts on the keyboard).
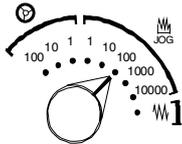
Turn the jog selector switch of the operator panel to one of the incremental jog positions.
Jog the desired axis using the JOG panel (keypad).

Jogging the axes with handwheels:

    Electronic handwheels may be used to move the axes.
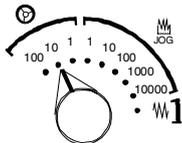
Turn the jog selector switch of the operator panel to one of the handwheel positions.

    Once the desired resolution has been selected and depending on the type of handwheel being used, general or individual, proceed as follows:

        1. General handwheel (may be used to jog any axis of the machine):

        On the JOG keyboard, press one of the keys for the axis to be jogged. If several axes are selected simultaneously, they all move at the same time. The CNC moves each axis as the handwheel is turned depending on the setting of the selector switch and on the turning direction of the handwheel.

        2. Individual handwheel (it is associated with a particular axis):

        The CNC moves each axis as its relevant handwheel is turned depending on the setting of the selector switch and on the turning direction of the handwheel.

Definition of machining conditions

Press the [S] key to select a spindle speed.
Press the [F] key to select a feedrate.

Enter the desired spindle speed or feedrate.

Press [START] to assume the entered value.
Press [ESC] to cancel the operation.

Spindle control

It is recommended to set the spindle turning speed (using the MDI mode) before selecting the turning direction.

    Starts the spindle clockwise at the active speed.

Spindle control

Starts the spindle counterclockwise at the active speed.

To stop the spindle.

+ — Spindle override (it may be used to vary the spindle turning direction).

Spindle orientation (it orients the spindle).

Tool selection and tool change

T Press the [T] key.
Key in the number of the tool to be placed in the spindle.
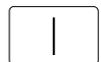
| ESC Press [START] to carry out the tool change.
Press [ESC] to cancel the operation.

Tool calibration

This operation is accessed with the "tool calibration" softkey.

The screen switching key may be used to select the calibration mode (with probe / without probe).

ESC Press [ESC] to quit this mode.

Data definition:

ENTER Place the cursor over the relevant data, key in the desired value and press [ENTER].

1. Tool calibration without a probe:

It may be used to calibrate the length (L) of the tool that is currently at the spindle.

The radius (R) must be defined directly. To calibrate the length, follow these steps:

• Define the Z coordinate of the part used for calibration.

• Select the tool and the offset.

| The CNC places the tool into the spindle and shows the dimensions defined in the tool table for that offset.

• Use the MDI mode to start the spindle.

• Approach the tool manually until it touches the part.

Press the "Calibration" softkey.

2. Tool calibration with probe (only when using a table-top probe installed on the machine):

It may be used to calibrate the length (L) or the radius (R) of the tool that is currently in the spindle.

To calibrate, follow these steps:

- Define the probing distance and feedrate.

- Select the tool and the offset.

    | ENTER ◇ | The shows the dimensions defined in the tool table for that offset.

    | | | The CNC places the tool into the spindle and shows the dimensions defined in the tool table for that offset.

- Select the axis and the probing direction on the softkey menu.

- Use the MDI mode to start the spindle.

    | | | The tool moves along the selected axis and direction until touching the probe.

Automatic loading of zero offsets or fixture offset tables
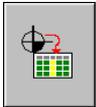
The CNC shows the list of available zero offsets and fixture offsets.

Select the zero offset or fixture offset where you wish to save the active offset.

| ENTER ◇ | Press [ENTER] to enter the offset in the table.

| ESC | Press [ESC] to cancel the operation at any time.

Note: For the CNC to assume one of the stored zero offsets or fixture offsets, execute, in MDI mode, the function G54, G55, ... , G59 or G159 associated with it.

✋ MDI MODE

The MDI mode may be used in any work mode.

Operations in MDI mode

The following operations are possible in this MDI work mode:

- Edit and execute individual blocks.

- Save the set of blocks executed as an independent program.

Standard MDI window:

It is possible to edit and execute new blocks or recover blocks saved in the block history, which may be modified before executing them.

- Editing:

  The blocks may be edited directly on the edit line or may be recalled from the block history.

  | ⇑ | ⇓ | To access the history and select a block.

  | ENTER ◇ | To accept the selection and show the block on the edit line.

  | ESC | To cancel the selection and quit the MDI mode.

- Execution:

  | | | Press [START] to execute currently displayed on the editing line.

  | ○ | Press [STOP] to interrupt the execution of the block.
  Press [START] to resume execution.

  | RESET | Press [RESET] to cancel the execution of the block and perform a general reset of the CNC to its initial conditions.

⊡ AUTOMATIC MODE

Program selection

A different program may be selected and executed in each channel. The program is executed in the channel where it is selected from.

To select a program, press the softkey "Select program" of the softkey menu and the CNC will show a list of all the programs available.

Single-block execution mode

When this mode is active, the execution of the program is interrupted at the end of each block.

It selects the single block execution mode.

Tool inspection

This option is only available when the running program has been interrupted.

Press the relevant key to start the tool inspection.

Reposition the axes and the spindle:

To end tool inspection and reposition the spindle and the axes at the point where tool inspection began, press the relevant softkey.

After pressing this softkey, the CNC will show a list of the axes that are out of position.

• Reposition the axes:

Select the axes with the relevant softkey.
Press [START] to perform the repositioning.

Press [STOP] to interrupt the repositioning and select other axes.

• Restore the spindle turning direction:

Select the relevant softkey.
Press [START] to restore the turning direction.

Block search

This option may ver used to resume the execution of a program from the point where it was interrupted or aborted.

The CNC shows the options available for selecting the stop condition.

| ⏎ | It returns to the main menu. |

• Stop block:

It sets as the last block for program history recovery the one currently selected with the cursor.

• To select a subroutine from the list:

Select the folder containing the subroutine.

Select the program containing the subroutine or write its name in the bottom window.

| ENTER ◇ | ESC | Press [ENTER] to open the program. |
| | | Press [ESC] to cancel the selection and close the list of programs. |

• Number of times:

It sets as stop condition, that the block selected as the stop block has been executed a specific number of times.

ENTER ◇	Enter the number of times.
	Press [ENTER] to accept the value.
ESC	Press [ESC] to reject the value.

• Automatic search:

With this option, it is possible to recover the program history up to the block where the execution was interrupted.

• Starting block:

It establishes the block currently selected with the cursor as the first block of the block search. There are two ways to select the block:
  • Using the cursor
  • Using the options "Text search" or "go to line" of the softkey menu.

✎ EDITING - SIMULATION MODE

Open the program to be edited

To select a program for the editing - simulation mode. This program may be a new one or an existing one. A different program may be edited and executed in each channel.

To select a program from the list:

Select the folder that contains the program. If a new program is selected, it will be saved in this folder.

Select the program from the list or write its name in the bottom window. To edit a new program, write the name of the program in the bottom window.

ENTER ⬦	ESC

Press [ENTER] to accept the selection and open the program.
Press [ESC] to cancel the selection and close the list of programs.

Program simulation

The graphic window shows the program selected at the editing window and its name appears at the bottom center of the screen.

The program simulation procedure is the following:

Choose the type of graphic representation, its dimensions and the point of view.

Activate the desired simulation options using the softkey menu.

Press the [START] softkey to start the simulation.
Press the [STOP] softkey to interrupt the simulation.
Press the [RESET] softkey to cancel the simulation.

"Single block" mode of simulation:

It selects the single-block mode using the relevant softkey.
It may be selected before or while simulating the program.

In this mode, the program is interrupted at the end of each block.
Press the [START] softkey to continue.

Simulation options:

The available simulation options are accessed from the softkey menu.
Pressing the softkey displays a window that shows the following options:

Tool radius compensation.

M01 Conditional stop during simulation.

Software limits.

Block skip.

LIST OF "G" FUNCTIONS

M	The function is modal.
D	By default, this function is active
	?: The initial value depends on the machine parameter.
V	This function is displayed in the G-code history.

Function	M	D	V	Meaning
G00	*	?	*	Rapid traverse
G01	*	?	*	Linear interpolation
G02	*		*	Clockwise circular (helical) interpolation
G03	*		*	Counterclockwise circular (helical) interpolation
G04			*	Dwell
G05	*	?	*	Controlled corner rounding (modal)
G06			*	Arc center in absolute coordinates (not modal)
G07	*	?	*	Square corner (modal)
G08			*	Arc tangent to previous path
G09			*	Arc defined by three points
G10	*	*		Mirror image cancellation
G11	*		*	Mirror image on the first axis of the channel
G12	*		*	Mirror image on the second axis of the channel
G13	*		*	Mirror image on the third axis of the channel
G14	*		*	Mirror image in the programmed directions
G17	*	?	*	Main plane X-Y, and longitudinal axis Z
G18	*	?	*	Main plane Z-X, and longitudinal axis Y
G19	*		*	Main plane Y-Z, and longitudinal axis X
G20	*		*	Main plane by two directions and longitudinal axis
G30				Polar origin preset
G33	*		*	Electronic threading with constant pitch
G36			*	Automatic radius blend
G37			*	Tangential entry
G38			*	Tangential exit
G39			*	Automatic chamfer blend
G40	*	*		Cancellation of tool radius compensation
G41	*		*	Left-hand tool radius compensation
G42	*		*	Right-hand tool radius compensation

Function	M	D	V	Meaning
G50	*	?		Semi-rounded corner
G53	*			Zero offset cancellation
G54	!		*	Absolute zero offset 1
G55	!		*	Absolute zero offset 2
G57	!		*	Absolute zero offset 4
G58	!		*	Absolute zero offset 5
G59	!		*	Absolute zero offset 6
G60			*	Square corner (not modal)
G61			*	Controlled corner rounding (not modal)
G63	*		*	Rigid tapping
G70	*	?	*	Programming in inches
G71	*	?		Programming in millimeters
G72			*	Scaling factor
G73	*		*	Coordinate system rotation (pattern rotation)
G74			*	Home search
G80	*	*		Canned cycle cancellation
G81	*		*	Drilling canned cycle
G82	*		*	Drilling canned cycle with variable peck
G83	*		*	Deep-hole drilling canned cycle with constant peck
G84	*		*	Tapping canned cycle
G85	*		*	Reaming canned cycle
G86	*		*	Boring canned cycle
G87	*		*	Rectangular pocket canned cycle.
G88	*		*	Circular pocket canned cycle
G90	*	?		Programming in absolute coordinates
G91	*	?	*	Programming in incremental coordinates
G92	!		*	Coordinate preset
G93	*		*	Machining time in seconds
G94	*	?		Feedrate in millimeters/minute (inches/minute)
G95	*	?	*	Feedrate in millimeters/revolution (inches/revolution)
G96	*		*	Constant surface speed
G97	*	*		Constant turning speed
G98	*	*		Withdrawal to the starting plane
G99	*		*	Withdrawal to the reference plane
G100			*	Probing

Function	M	D	V	Meaning
G101	*			Include probe offset
G102	*			Exclude probe offset
G108	*	*		Feedrate blending at the beginning of the block
G109			*	Feedrate blending at the end of the block
G112	*			Changing of parameter range of an axis
G130	*		*	Percentage of acceleration to be applied per axis
G131	*		*	Percentage of acceleration to be applied to all the axes
G132	*		*	Percentage of jerk to be applied per axis
G133	*		*	Percentage of jerk to be applied to all the axes
G134	*		*	Percentage of Feed-Forward to be applied
G135	*		*	Percentage of AC-Forward to be applied
G136	*	?	*	Circular transition between blocks
G137	*	?		Linear transition between blocks
G138	*		*	Direct activation/cancellation of tool compensation
G139	*	*		Indirect activation/cancellation of tool compensation
G151	*	*	*	Programming in diameters
G152	*			Programming in radius
G157	*		*	Excluding axes in the zero offset
G158	*		*	Incremental zero offset
G159	!		*	Additional absolute zero offsets
G160			*	Multiple machining in straight line
G161			*	Multiple machining in rectangular pattern
G162			*	Multiple machining in grid pattern
G163			*	Multiple machining in a full circle
G164			*	Multiple machining in arc pattern
G165			*	Machining programmed with an arc-chord
G170	*			Hirth axes OFF
G171	*	*		Hirth axes ON
G180			*	OEM Subroutine
G181			*	OEM Subroutine
G182			*	OEM Subroutine
G183			*	OEM Subroutine
G184			*	OEM Subroutine
G185			*	OEM Subroutine
G186			*	OEM Subroutine

Function	M	D	V	Meaning
G187			*	OEM Subroutine
G188			*	OEM Subroutine
G189			*	OEM Subroutine
G192	*		*	Turning speed limit in constant cutting speed mode
G193			*	Interpolating the feedrate
G196	*		*	Constant cutting point feedrate
G197	*	*		Constant tool center feedrate
G198	*			Setting of lower software travel limits
G199	*			Setting of upper software travel limits
G200				Exclusive manual intervention
G201	*			Activation of additive manual intervention
G202	*	*		Cancellation of additive manual intervention
G261	*		*	Arc center in absolute coordinates (modal)
G262	*	*		Arc center referred to starting point
G263	*		*	Arc radius programming
G264	*		*	Cancellation of arc center correction
G265	*	*		Activation of arc center correction
G266			*	Feedrate override at 100%
G281			*	Conversational center-punching cycle
G282			*	Conversational drilling cycle 1
G283			*	Conversational drilling cycle 2
G284			*	Conversational tapping cycle
G285			*	Conversational reaming cycle
G286			*	Conversational boring cycle 1
G287			*	Conversational rectangular pocket cycle
G288			*	Conversational circular pocket cycle
G289			*	Conversational simple pocket cycle
G290			*	Conversational surface milling cycle
G291			*	Conversational rectangular boss cycle
G292			*	Conversational circular boss cycle
G293			*	Conversational point-to-point profiling cycle
G294			*	Conversational profiling cycle
G295			*	Conversational slot milling cycle
G296			*	Conversational pre-emptied pocket cycle
G297			*	Conversational boring cycle 2

LIST OF "M" FUNCTIONS

Function	Meaning
M00	Program stop.
M01	Conditional program stop.
M02/M30	End of program.
M17/M29	End of subroutine.
M06	Tool change.
M03	Start the spindle clockwise.
M04	Start the spindle counterclockwise.
M05	Stop the spindle.
M19	Spindle orientation.
M41-M44	Spindle gear change.

TECHNOLOGICAL FUNCTIONS

Machining feedrate (F)

The machining feedrate may be selected by programmed using the "F" code which remains active until another value is programmed.
The programming units depend on the active work mode (G93, G94 or G95) and the type of axis being moved (linear or rotary).

Spindle speed (S)

The spindle speed is selected by program using the spindle name followed by the desired speed.
The speeds of all the spindles of the channel may be programmed in the same block.
The programmed speed stays active until another value is programmed.
The programming units will be RPM unless selected otherwise.
If G96 is active, the programming units will be m/min.

Tool number (T)

The "T" code identifies the tool to be selected.
The tools may be in a magazine managed by the CNC or in a manual magazine (referred to as ground tools).

Tool offset number (D)

The tool offset contains the tool dimensions.
Each tool may have several offsets associated with it.
To activate an offset, it must be previously defined. To do that, the CNC offers a section of the tool table where several offsets may be defined.

CANNED CYCLES

Preparatory (G), technological (F, S) and auxiliary (M, H) functions must be defined before the canned cycle.
Functions G98, G99 and the positioning move to the machining point must also be programmed before.

G81. Drilling canned cycle

G81 Z I K
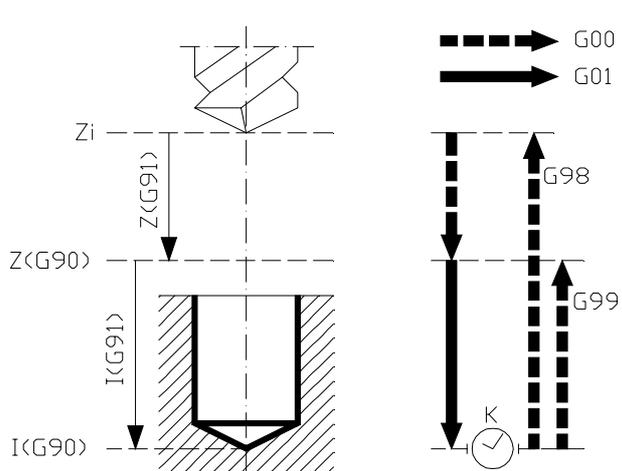
Z	Reference plane.
I	Drilling depth.
K	Delay, in seconds, between the drilling and the withdrawal movement.

G82. Drilling canned cycle with variable peck

G82 Z I D B H C J K R L

Z	Reference plane.
I	Drilling depth.
D	Distance between the reference plane and the part surface.
B	Drilling peck (step).
H	Distance or coordinate it returns to, in rapid (G0), after each drilling step.
C	Approach coordinate.
J	It defines after how many drilling pecks the tool returns in rapid (G0) to the reference plane (Z).
K	Dwell, in seconds, at the bottom of the hole.
R	Factor that increases or reduces the drilling peck (step) "B". The first peck will be "B", the second "RB" and so on.
L	Minimum value for the drilling peck. Only if R is other than 1.

G83. Deep-hole drilling canned cycle with constant peck

G83 Z I J B K
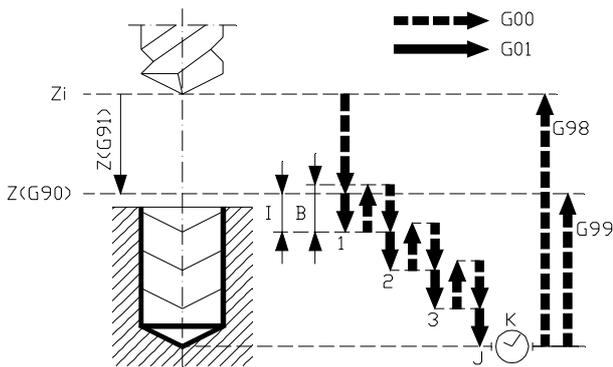
Z Reference plane.

I Drilling peck (step).

J Number of pecks required by the drilling operation.

B Rapid withdraw (G0) distance after each drilling step.

K Dwell, in seconds, at the bottom of the hole.

G84. Tapping canned cycle

G84 Z I K R
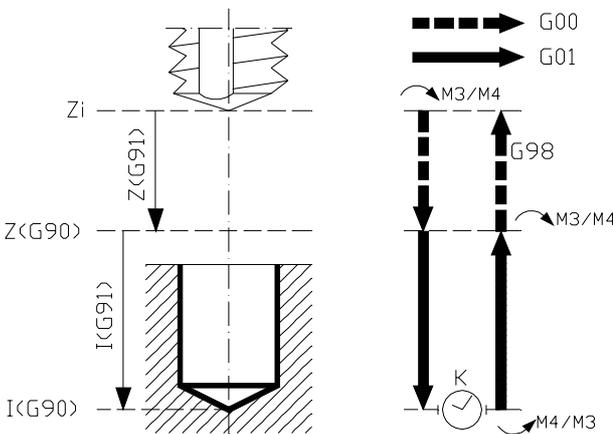
Z Reference plane.

I Tap depth.

K Delay, in seconds, between the tapping and the withdrawal movement.

R Type of tapping.

  R0: normal tapping.
  R1: rigid tapping.

G85. Reaming canned cycle

G85 Z I K
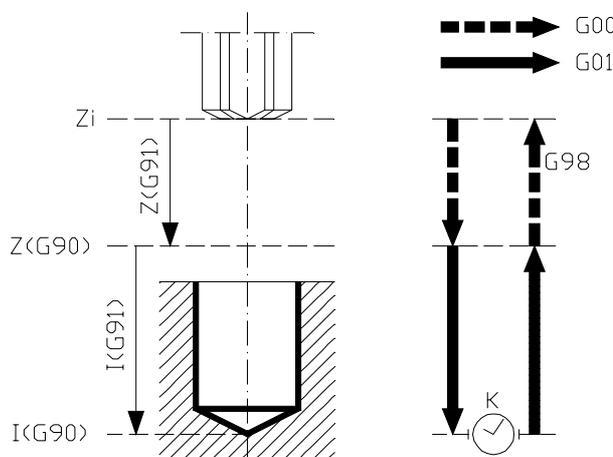
Z Reference plane.

I Reaming depth.

K Delay, in seconds, between the reaming and the withdrawal movement.

G86. Boring canned cycle

G86 Z I K R

Z   Reference plane.

I   Boring depth.

K   Delay, in seconds, between the boring and the withdrawal movement.

R   Type of withdrawal: R0 rapid (G0), R1 at work feedrate (G01).

G87. Rectangular pocket canned cycle.

G87 Z I D A J K M Q B C L H V

Z   Reference plane.

I   Pocket depth.

D   Distance between the reference plane and the part surface.

A   Angle, in degrees, between the pocket and the abscissa axis.

J   Half length of the pocket.

K   Half width of the pocket.

M   Type of corner. (0) square, (1) rounded, (2) chamfered.

Q   Rounding radius or chamfer size.

B   Depth of pass.

C   Milling pass or width.

L   Finishing pass.

H   Feedrate for the finishing pass. If not programmed or programmed with a 0 value, it is carried out at the roughing feedrate.

V   Tool penetrating feedrate. If not programmed or programmed with a 0 value, it is carried out at 50% of the feedrate in the plane.

G88. Circular pocket canned cycle

G88 Z I D J B C L H V
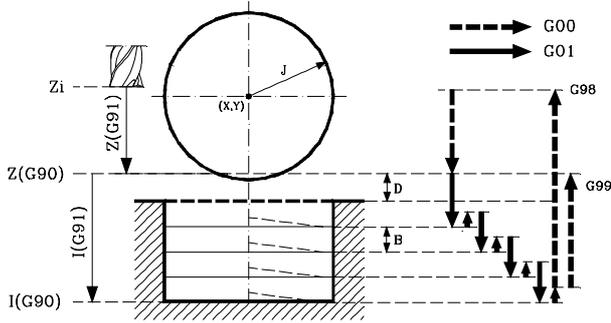
Z   Reference plane.

I   Pocket depth.

D   Distance between the reference plane and the part surface.

J   Pocket radius.

B   Depth of pass.

C   Milling pass or width.

L   Finishing pass.

H   Feedrate for the finishing pass. If not programmed or programmed with a 0 value, it is carried out at the roughing feedrate.

V   Tool penetrating feedrate. If not programmed or programmed with a 0 value, it is carried out at 50% of the feedrate in the plane.

MULTIPLE MACHINING

G160. Multiple machining in straight line

G160 A | X I | P Q R S T U V
        | X K |
        | I K |
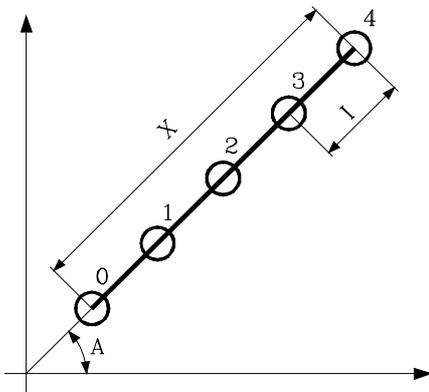
A   Angle, in degrees of the machining path with respect to the abscissa axis.

X   Length of the machining path.

I   Step between machining operations.

K   Total number of machining operations in the section, including that of the machining definition point.

G161. Multiple machining in rectangular pattern

G161 A B | X I | Y J | P Q R S T U V
         | X K | Y D |
         | I K | J D |
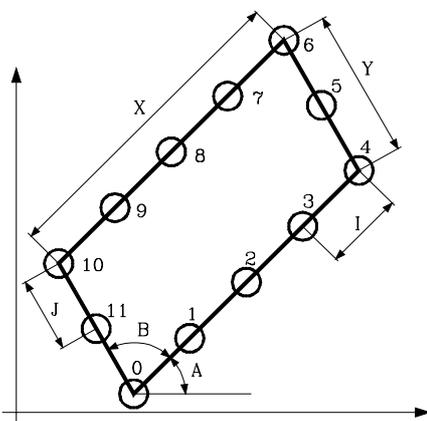
A   Angle, in degrees of the machining path with respect to the abscissa axis.

B   Angle between both machining paths.

X   Length of the parallelogram.

I   Step between machining operations along the path.

K   Total number of machining operations along the path, including that of the machining definition point.

Y   Width of the parallelogram.

J   Step between machining operations along the path.

D   Total number of machining operatiqons along the path, including that of the machining definition point.

G162. Multiple machining in grid pattern

G162 A B │ X I │ Y J │ P Q R S T U V
          │ X K │ Y D │
          │ I K │ J D │
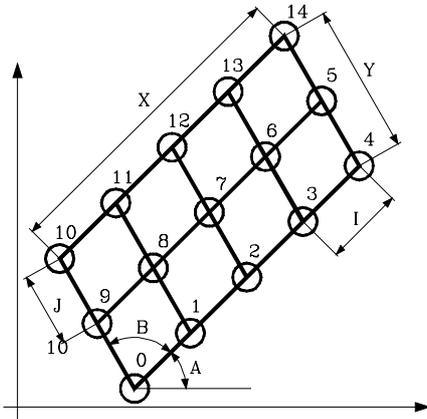
A	Angle, in degrees of the machining path with respect to the abscissa axis.
B	Angle between both machining paths.
X	Length of the drid.
I	Step between machining operations along the path.
K	Total number of machining operations along the path, including that of the machining definition point.
Y	Width of the drid.
J	Step between machining operations along the path.
D	Total number of machining operations along the path, including that of the machining definition point.

G163. Multiple machining in a full circle

G163 X Y │ I │ C F P Q R S T U V
          │ K │
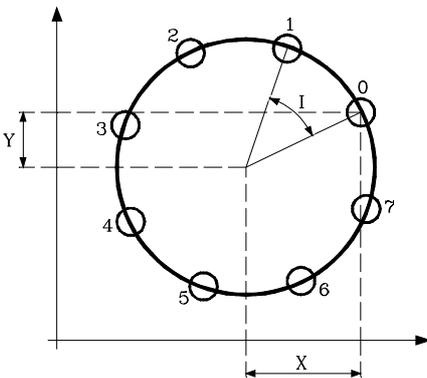
X	Distance from the starting point to the center along the abscissa axis.
Y	Distance from the starting point to the center along the ordinate axis.
I	Angular step between machining operations.
K	Total number of machining operations including that of the machining definition point.
C	It indicates how it will move between the machining points.
	C=0 In rapid (G00).
	C=1 Linear interpolation (G01).
	C=2 In clockwise circular interpolation (G02).
	C=3 In counterclockwise circular interpolation (G03).
F	Feedrate for the movement between points. Only if C is other than 0.

G164. Multiple machining in arc pattern

G164 X Y B │I│ C F P Q R S T U V
           │K│
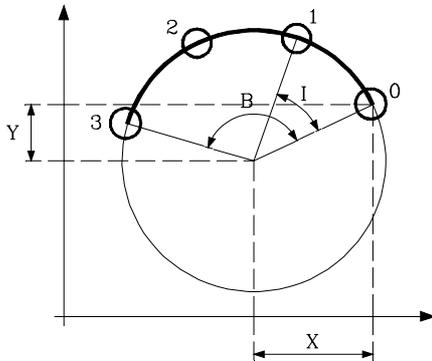
X   Distance from the starting point to the center along the abscissa axis.

Y   Distance from the starting point to the center along the ordinate axis.

B   Angular distance in degrees of the machining path.

I   Angular step between machining operations.

K   Total number of machining operations including that of the machining definition point.

C   It indicates how it will move between the machining points.

     C=0   In rapid (G00).

     C=1   Linear interpolation (G01).

     C=2   In clockwise circular interpolation (G02).

     C=3   In counterclockwise circular interpolation (G03).

F   Feedrate for the movement between points. Only if C is other than 0.

G165. Machining programmed with an arc-chord

G165 X Y │A│ C F
         │I│
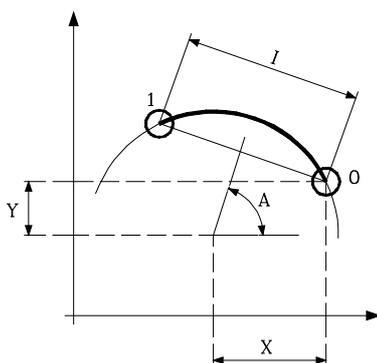
X   Distance from the starting point to the center along the abscissa axis.

Y   Distance from the starting point to the center along the ordinate axis.

A   Angle, in degrees of the perpendicular bisector of the chord with respect to the abscissa axis.

I   Length of the chord.

C   It indicates how it will move between the machining points.

     C=0   In rapid (G00).

     C=1   Linear interpolation (G01).

     C=2   In clockwise circular interpolation (G02).

     C=3   In counterclockwise circular interpolation (G03).

F   Feedrate for the movement between points. Only if C is other than 0.

HIGH LEVEL LANGUAGE LISTING

Programming instructions

Display instructions:

#ERROR [<number>]	It displays the indicated error number and its associated text according to the CNC's error code (interrupts program execution).
#ERROR ["<Error text>"]	It displays the indicated error text (it interrupts program execution).
#WARNING [<number>]	It displays the indicated warning number and its associated text according to the CNC's error code (it does not interrupt program execution).
#WARNING["<warning text>"]	It displays the indicated text (it does not interrupt program execution).
#MSG ["<message>"]	The indicated message appears at the top of the screen (it does not interrupt the execution of the program).
#DGWZ [Xmin, Xmax, Ymin, Ymax, Zmin, Zmax]	
	It defines the graphic area.

Enabling and disabling instructions:

#ESBLK #DSBLK	It activates and deactivates the single block treatment.
#ESTOP #DSTOP	It enables and disables the STOP signal.
#EFHOLD #DFHOLD	It enables and disables the FEED-HOLD input coming from the PLC.

Programming referred to machine reference zero (home):

#MCS ON #MCS OFF	It activates and deactivates the machine coordinate system so the movements programmed after it are executed in the machine reference system.

Program instructions:

#EXEC [<path><prg>,<channel>]	It executes a program in the indicated channel.
#EXBLK [<block>,<channel>]	It executes a block in the indicated channel.

Electronic axis slaving:

#LINK [<master>, <slave>, <error>] ... [...]	
	It defines and activates the master-slave pair of axes.
#UNLINK	It cancels the active couplings (slaving).

Subroutine instructions:

#PATH ["<path>"]   It defines the path for searching subroutines.

#CALL <path><subroutine>   It calls a local or global subroutine.

#PCALL <path><subroutine><parameters>

It calls a local or global subroutine with the possibility to initialize parameters.

#MCALL<path><subroutine><parameters>

It calls a local or global subroutine with the possibility to initialize parameters.
With this type of call, the subroutine becomes modal.

#MDOFF   It makes the function non-modal.

#RET   It defines the end of a subroutine.

Axis parking:

#PARK <axis>
#UNPARK <axis>   It parks and unparks the selected.

Axis swapping:

#SET AX [<axis>, ..] <offsets>   Define a new axes configuration.

#CALL AX [<axis>, <pos>, ..] <offsets>

it adds one or more axes to the preset configuration and it also allows defining its position.

#FREE AX [<axis>, ..]   Removes the programmed axes from the current configuration.

#RENAME AX [<axis1>, <axis2>] .. [..]

For each programmed axis pair, the first axis takes the name of the second one.

Spindle swapping:

#FREE SP [<spindle>, ..]   Removes the defined spindles from the current configuration.

#CALL SP [<spindle>, ..]   It adds one or several spindles to the current configuration.

#SET SP [<spindle>, ..]   Defines a new spindle configuration.

#RENAME SP [<spindle1>, <spindle2>] .. [..]

For each programmed spindle pair, the first spindle takes the name of the second one.

Selecting the master spindle of a channel:

#MASTER <spindle>   It defines the master spindle of a channel.

Longitudinal tool axis selection:

#TOOL AX [<axis><sign>]   It selects any machine axis as the new longitudinal axis.

"C" axis: Activate the spindle as "C" axis:

#CAX [<spindle>,<name>]	It activates the spindle as "C" axis.
#CAX OFF	It cancels the "C" axis, going back to spindle mode.

"C" axis: Machining of the face of the part:

#FACE [<abscissa>, <ordinate>, <longitudinal axis>]

It activates the machining of the face and it defines the work plane. The longitudinal axis of the tool is optional.

#FACE OFF   It cancels the machining of the face of the part.

"C" axis: Machining of the turning side of the part:

#CYL [<abscissa>, <ordinate>, <longitudinal axis>, <radius>]

It activates the machining of the turning side and it defines the work plane. The radius is optional.

#CYL OFF   It cancels machining of the turning side of the part.

Collision detection:

#CD ON [<horizon>]
#CD OFF   It activates and deactivates the collision detecting process.

Related to manual intervention:

#CONTJOG [<feedrate>] <axis>   It defines the continuous jogging feedrate for the indicated axis.

#INCJOG [[<increment, feedrate>][...][...][...][...]] <axis>

It defines the indicated incremental movement and axis feedrate for each incremental JOG position of the selector switch.

#MPGRESOL [<pos1>, ..., <pos3>] <axis>

It defines the distance per handwheel pulse for the indicated axis at each position of the selector switch.

#SET OFFSET [<lower>, <upper>] <axis>

It defines the upper and lower limits of the indicated axis, within which the axes can be jogged during additive manual intervention.

#SYNC POS   This instruction synchronizes the preparation coordinate with the execution one and assumes the additive manual offset.

High speed machining:

#HSC ON [CONTERROR <error>]

#HCS OFF   It activates and cancels the high speed machining mode.

Acceleration control:

#SLOPE [<type>,<jerk>,<accel>,<move>]

It is used to set the influence of the values defined with functions G130, G131, G132 and G133 in the behavior of the acceleration.

Splines (Akima):

#SPLINE ON and #SPLINE OFF

It activates and cancels the spline adaptation.

#ASPLINE MODE [<initial>,<final>]

It sets the type of initial and final tangents of the spline that determines the transition from the previous and to the next path.

#ASPLINE STARTTANG <axes>

It sets the initial tangent of the spline.

#ASPLINE ENDTANG <axes>   It sets the final tangent of the spline.

Polynomial interpolation:

#POLY [<axis>[a, b, c, d, e] ... SP<> EP<>]

This type of interpolation allows machining a polynomial of up to a fourth degree.

Coordinate transformation:

#KIN ID [<kinematics>]   It defines the kinematics of the spindle.

#CS   It defines, stores, activates, cancels up to 5 machining coordinate systems.

#ACS   It defines, stores, activates, cancels up to 5 fixture coordinate systems.

#RTCP ON   It changes the tool orientation without modifying the position occupied by its tip on the part.

#RTCP OFF   It cancels RTCP transformation.

#TOOL ORI   It positions the tool perpendicular to the work plane.

#TLC ON [n]
#TLC OFF   It activates and cancels the TLC function.

Definition of macros:

#DEF "macro" = "CNCblock"   It defines macros.

#INIT MACROTAB   It resets the table of macros erasing the ones stored in it.

Block repetition:

#RPT [<label1>, <label2>, <n>]

It repeats the execution of the portion of the program defined between two blocks.

Communication and synchronization between channels:

#MEET [<mark>, <channel>, ..]	It activates the mark indicated in the channel and waits for it to be activated in the rest of the programmed channels.
#WAIT [<mark>, <channel>,..]	It waits for the mark to be activated in the indicated channel.
#SIGNAL [<mark>, ..]	It activates the mark in its own channel.
#CLEAR [<mark>, ..]	It clears the synchronism marks of the channel.

Additional programming instructions:

#COMMENT BEGIN #COMMENT END	It indicates the beginning and the end of a comment.
#FLUSH	It evaluates a block when it is being executed.
#WAIT FOR [<condition>]	It interrupts program execution until the condition is met.
#SELECT PROBE [<probe>]	It selects the probe.
#TANGFEED RMIN [<radius>]	When applying constant tangential feedrate (G196), with this instruction it is possible to set a minimum radius so this type of feedrate is only applied on arcs whose radius is larger than this minimum.
#ROUNDPAR	It selects and defines the type of corner rounding to be applied using G5 and G61.
#TIME [<time>]	It interrupts the execution of the program for the indicated time period (in seconds).
#SCALE [<scale>]	It enlarges or reduces programmed parts (workpieces).

Flow controlling instructions

Jump to a block ($GOTO):

$GOTO N<expression>	It causes a jump to a block defined with "N<expression>:".
$GOTO [<label>]	It causes a jump to a block defined with "<label>".

Conditional block repetition ($DO):

$DO ... $ENDDO <condition>	While the condition is true, it repeats the execution of the blocks contained between $DO and $ENDDO.

Conditional execution ($SWITCH):

$SWITCH <expression1> ... $CASE<expression2> ... $ENDSWITCH

It calculates the result of <expression1> and executes the blocks contained between the $CASE instruction, whose <expression2> has the same value as the calculated result and the corresponding $BREAK instruction.

Conditional execution ($IF):

$IF <condition> ... $ENDIF   It analyzes the programmed condition.
- If the condition is true, it executes the blocks contained between the $IF and $ENDIF instruction.
- If the condition is false, the execution continues at the block after $ENDIF.

$IF <condition> ... $ELSE ... $ENDIF

It analyzes the programmed condition.
- If the condition is true, it executes the blocks contained between the $IF and $ENDIF instruction.
- If the condition is false, it executes the blocks contained between the $ELSE and $ENDIF instructions.

$IF <condition1> ... $ELSEIF<condition2> ... $ENDIF

It analyzes the programmed condition.
- If <condition1> is true, it executes the blocks contained between $IF and $ELSEIF.
- If <condition1> is false, it analyzes <condition2>. If it is true, it executes the blocks contained between the $ELSEIF and $ENDIF instruction.
- If all the conditions are false, the execution continues at the block after $ENDIF.

Block repetition ($FOR):

$FOR <n> = <expr1>, <expr2>, <expr3> ... $ENDFOR

When executing this instruction, <n> takes the value of <expr1> and it changes its value up to the value of <expr2>, in steps indicated by <expr3>. At each step, it executes the blocks contained between $FOR and $ENDFOR.

Conditional block repetition ($WHILE):

$WHILE <condition> ... $ENDWHILE

While the condition is true, it executes the blocks contained between $WHILE and $ENDWHILE.

OPERATORS AND FUNCTIONS

Arithmetic

+	Add	**	Exponent
-	Subtract / change sign	+=	Compounded addition
*	Multiply	-=	Compounded subtraction
/	Divide	*=	Compounded multiplication
MOD	Module or remainder of a division	/=	Compounded division

Relational

==	Equal to	>=	Greater than or equal to	>	Greater than
!=	Not equal	<=	Smaller than or equal to	<	Smaller than

Binary

&	Binary AND	^	Exclusive OR (XOR)
I	Binary OR	INV[...]	Inverse

Logic

*	Logic AND	+	Logic OR

Boolean

True	True	False	Not true

Trigonometric

SIN[...]	Sine	ASIN[...]	Arc-sine	ARG[...]	Arctangent y/x
COS[...]	Cosine	ACOS[..]	Arc-cosine		
TAN[...]	Tangent	ATAN[...]	Arc-tangent		

Mathematical

ABS[...]	Absolute value	LOG[...]	Decimal logarithm	DEXP[..]	Decimal exponent
SQR[...]	Square function	LN[...]	Neperiam logarithm		
SQRT[..]	Square root	EXP[...]	"e" function		

Other functions

INT[...]	Returns the integer	FUP[...]	Returns the integer plus one. If the number is an integer, it returns it.
FRACT[...]	Returns decimal portion		
ROUND[..]	Rounds up or down to the nearest integer	EXIST[...]	It checks whether the selected variable or parameter exists or not

▣ USER TABLES

Description of vertical softkeys

✚	It changes the group of icons offered by the menu.	Find text.
	Display units (millimeters/inches).	Select axes.
	Initialize table.	Save tool.
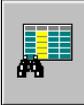
(Recall table. / Print table. / Accessing the tables of other channels.)

Zero offset tables

This table stores the zero offsets for each axis. When accessing from a channel, only the axes of that channel are shown.

The offsets of other channels may be accessed from the vertical softkey menu.

PLC offset (PLCOF):

- The values may be set from the PLC or from the part-program using high level language variables, but they cannot be set directly in the table.

Absolute zero offsets:

- It is used to place the part zero in different positions of the machine.
- It may be set directly in the table, from the PLC or from the part-program using high level language variables.

Fixture table

This table stores the clamp offsets for each axis. When accessing from a channel, only the axes of that channel are shown.

The offsets of other channels may be accessed from the vertical softkey menu.

Fixture offset:

- It may be set directly in the table, from the PLC or from the part-program using high level language variables.
- The clamp offsets are used to set the position of the clamping system of the machine.
- To apply a clamp offset it must be activated from the program using the relevant variable.

Arithmetic parameter tables

It may be set directly in the table, from the PLC or from the part-program.

The offsets of other channels may be accessed from the vertical softkey menu.

There are tables of common, global and local parameters.

🔧 **UTILITIES**

List of icons

Cut:
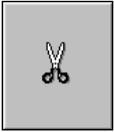
Copies the selected files on the clipboard. After pasting the content of the clipboard, the files are deleted from the folder.

Copy:

Copies the selected files on the clipboard.

Paste:

Pastes the files from the clipboard into the selected folder. If the files were placed using the "Cut" option, they will be removed from their original location.

Rename:

It is used to change the name of the folder or file currently selected.
- If when renaming a folder, there is already another folder with that new name, the proposed name change will be ignored.
- If when renaming a file, there is a previous file with the same name in the folder, the new file replaces the previous one.

Modifiable file:

M

It is used to change the "modifiable" attribute of the selected files. This attribute allows protecting the files so they cannot be modified in the editing mode.

Hidden file:

H

It is used to changed the "hidden" attribute of the selected files. This attribute allows protecting the files so they are not displayed when selecting a program to be edited or executed.

Remove file:

It is used for deleting the selected folder or files.
To delete the files, the CNC will show a dialog box requesting confirmation of the command whereas the empty folders will be deleted directly without requesting confirmation.
The folders can only be deleted if they are empty.