CNC 8040

Operating Manual
(M model)

Ref. 0204-ing

FAGOR

The information described in this manual may be subject to variations due to technical modifications.

FAGOR AUTOMATION, S.Coop. Ltda. reserves the right to modify the contents of the manual without prior notice.

Please note that the availability of some of the features described in this manual depends on the software options you just obtained.

	GP model	M model
Number of axes with standard software	4	4
Number of axes with optional software	7	7
Solid graphics	-----	Option
Irregular pockets with islands	-----	Option
Rigid tapping	Option	Option
Tool life monitoring	-----	Option
Probing canned cycles	-----	Option
DNC	Option	Option
COCOM version	Option	Option
Profile editor	Option	Option
Tool radius compensation	Option	Standard
Tangential control	Option	Option
Retracing	-----	Option
Electronic threading	-----	Standard
Tool magazine management	-----	Standard
Machining canned cycle	-----	Standard
Multiple machining	-----	Standard
Conversational software (MC and MCO)	-----	Option

INDEX

VERSION HISTORY

INTRODUCTION

1. OVERVIEW

2. OPERATING MODES

3. EXECUTE / SIMULATE

4. EDIT

5. JOG

6. TABLES

7. UTILITIES

8. DNC

9. PLC

10. SCREEN EDITOR

11. MACHINE PARAMETERS

12. DIAGNOSIS

VERSION HISTORY (M).
(Mill Model)

April 2002
Software: 5.3x.

Feature	Manual
New expansion board models for the 8055i	Installation Programming
Bus CAN OPEN to control remote digital I/Os at the 8055i	Installation Operation Error solutions
New PLC instructions: IREMRD and OREMWR:	Installation
Leadscrew error compensation on rotary axes between 0-360 degrees	Installation
PLC statistic deletion with a single softkey	Operation
Show only the XY plane in top-view graphics	Operation
Absolute reference mark management via Sercos (see a.m.p. REFVALUE)	Installation

Introduction

SAFETY CONDITIONS

Read the following safety measures in order to prevent damage to personnel, to this product and to those products connected to it.

This unit must only be repaired by personnel authorized by Fagor Automation.

Fagor Automation shall not be held responsible for any physical or material damage derived from the violation of these basic safety regulations.

Precautions against personal damage

Before powering the unit up, make sure that it is connected to ground

In order to avoid electrical discharges, make sure that all the grounding connections are properly made.

Do not work in humid environments

In order to avoid electrical discharges, always work under 90% of relative humidity (non-condensing) and 45° C (113° F).

Do not work in explosive environments

In order to avoid risks, damage, do no work in explosive environments.

Precautions against product damage

Working environment

This unit is ready to be used in Industrial Environments complying with the directives and regulations effective in the European Community

Fagor Automation shall not be held responsible for any damage suffered or caused when installed in other environments (residential or homes).

Install the unit in the right place

It is recommended, whenever possible, to instal the CNC away from coolants, chemical product, blows, etc. that could damage it.

This unit complies with the European directives on electromagnetic compatibility. Nevertheless, it is recommended to keep it away from sources of electromagnetic disturbance such as.

- Powerful loads connected to the same AC power line as this equipment.
- Nearby portable transmitters (Radio-telephones, Ham radio transmitters).
- Nearby radio / TC transmitters.
- Nearby arc welding machines
- Nearby High Voltage power lines
- Etc.

Ambient conditions

The working temperature must be between +5° C and +45° C (41ºF and 113º F)
The storage temperature must be between -25° C and 70° C. (-13º F and 158º F)

Precautions during repair

Do not manipulate the inside of the unit

Only personnel authorized by Fagor Automation may manipulate the inside of this unit.

Do not manipulate the connectors with the unit connected to AC power.

Before manipulating the connectors (inputs/outputs, feedback, etc.) make sure that the unit is not connected to AC power.

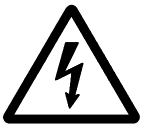
Safety symbols

Symbols which may appear on the manual

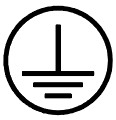
WARNING. symbol
It has an associated text indicating those actions or operations may hurt people or damage products.

Symbols that may be carried on the product

WARNING. symbol
It has an associated text indicating those actions or operations may hurt people or damage products.



"Electrical Shock" symbol
It indicates that point may be under electrical voltage



"Ground Protection" symbol
It indicates that point must be connected to the main ground point of the machine as protection for people and units.

MATERIAL RETURNING TERMS

When returning the Monitor or the Central Unit, pack it in its original package and with its original packaging material. If not available, pack it as follows:

1.- Get a cardboard box whose three inside dimensions are at least 15 cm (6 inches) larger than those of the unit. The cardboard being used to make the box must have a resistance of 170 Kg (375 lb.).

2.- When sending it to a Fagor Automation office for repair, attach a label indicating the owner of the unit, person to contact, type of unit, serial number, symptom and a brief description of the problem.

3.- Wrap the unit in a polyethylene roll or similar material to protect it.

    When sending the monitor, especially protect the CRT glass

4.- Pad the unit inside the cardboard box with poly-utherane foam on all sides.

5.- Seal the cardboard box with packing tape or industrial staples.

FAGOR DOCUMENTATION FOR THE CNC

OEM Manual	Is directed to the machine builder or person in charge of installing and starting-up the CNC.
USER-M Manual	Is directed to the end user or person who will operate this CNC in M mode. It contains 2 manuals: Operating Manual — describing how to operate the CNC. Programming Manual — describing how to program the CNC.
USER-T Manual	Is directed to the end user or person who will operate this CNC in T mode. It contains 2 manuals: Operating Manual — describing how to operate the CNC. Programming Manual — describing how to program the CNC.
MC Manual	Is directed to the end user or person who will operate this CNC in MC mode.
TC Manual	Is directed to the end user or person who will operate this CNC in TC mode.
MCO/TCO Manual	Is directed to the end user or person who will operate this CNC in MCO/ TCO mode.
DNC Software Manual	Is directed to people using the optional DNC communications software.
DNC Protocol Manual	Is directed to people wishing to design their own DNC communications software to communicate with the CNC.
FLOPPY DISK Manual	Is directed to people using the Fagor Floppy Disk Unit and it shows how to use it.

1. OVERVIEW

In this manual an explanation is given of how to operate the CNC by means of its Monitor-Keyboard unit and the Operator Panel.

The Monitor-Keyboard unit consists of:

* The Monitor or CRT screen, which is used to show the required system information.

* The Keyboard, which allows communication with the CNC, allowing information to be requested by means of commands or by changing the CNC status by generating new instructions.

1.1 PART-PROGRAMS

Editing

To create a part-program, access the Edit mode. See chapter 5 in this manual.

The new part-program edited is stored in the CNC's RAM memory.

A copy of the part-programs may be stored in the "MemKey Card", at a PC connected through serial line 1 or 2 or in the hard disk (HD module). See chapter 7 in this manual.

When using a PC through serial line 1 or 2, proceed as follows:
- Execute the "Fagor50.exe" applications program at the PC.
- Activate DNC communications at the CNC. See chapter 8 in this manual.
- Select the work directory as shown in chapter 7 of this manual. Option: Utilities\ Directory\ Serial L.\ Change directory.

With the Edit mode of operation, part-programs residing in the CNC's RAM memory may be modified. To modify a program stored in the "MemKey Card", in a PC or in the hard disk, it must be previously copied into RAM memory.

Execution

Part-programs stored anywhere may be executed or simulated. See chapter 3 in this manual.

The user customizing programs must be in RAM memory so the CNC can execute them.

The GOTO and RPT instructions cannot be used in programs executed from a PC connected through the serial lines. See chapter 14 of the programming manual.

The subroutines can only be executed if they reside in the CNC's RAM memory. Therefore, to execute a subroutine stored in the "MemKey Card", in a PC or in the hard disk, it must be first copied into the CNC's RAM memory.

From a program in execution, another program can be executed which is in RAM memory, in the "MemKey Card", in a PC or in the hard disk using the EXEC instruction. See chapter 14 of the programming manual.

Utilities

This operating mode, chapter 7 of this manual, lets display the part-program directory of all the devices, make copies, delete, rename and even set the protections for any of them.

Ethernet

When having the Ethernet option and if the CNC is configured as another node within the computer network, the following operations are possible from any PC of the network:

- Access the part-program directory of the Hard Disk(HD).
- Edit, modify, delete, rename, etc.the programs stored on the hard disk (HD).
- Copy programs from the hard disk to the PC and vice versa.

To configure the CNC as another node within the computer network, see section 3.3.4 of the installation manual.

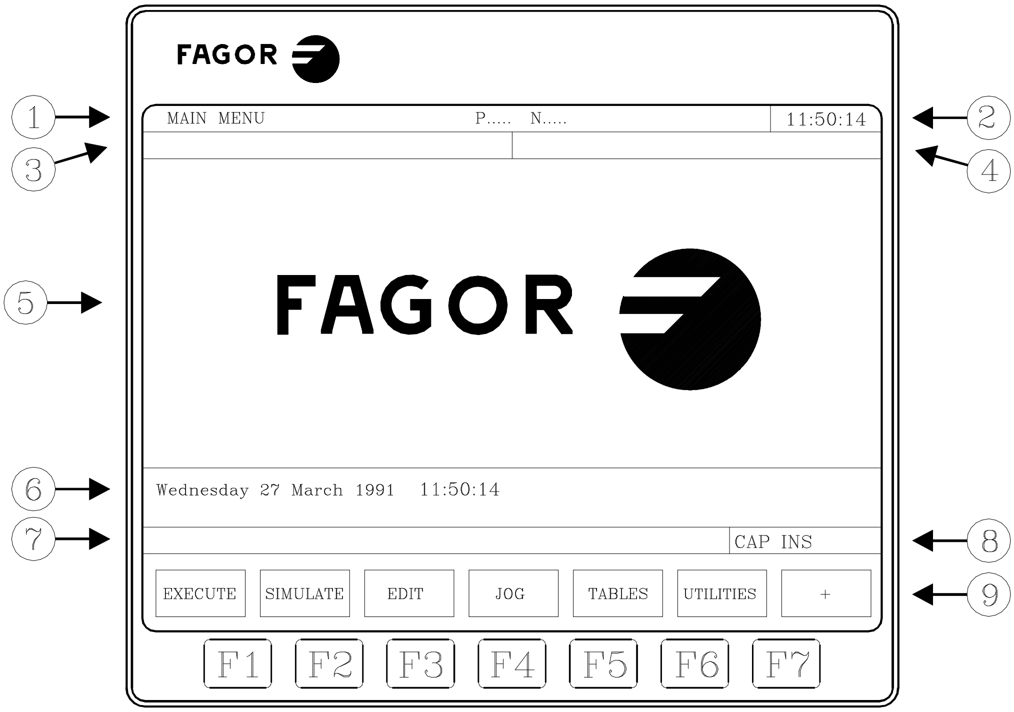
Operations that may be carried out with part-programs:

	RAM Memory	CARD A	HD	DNC
Consult the program directory in ...	Yes	Yes	Yes	Yes
Consult the subroutine directory in ...	Yes	No	No	No
Create work directory in ..	No	No	No	No
Change work directory in ..	No	No	No	Yes
Edit a program in ..	Yes	No	No	No
Modify a program in ..	Yes	No	No	No
Delete a program from ..	Yes	Yes	Yes	Yes
Copy from/to RAM memory to/from ...	Yes	Yes	Yes	Yes
Copy from/to CARD A to/from ...	Yes	Yes	Yes	Yes
Copy from/to HD to/from ...	Yes	Yes	Yes	Yes
Copy from/to DNC to/from ...	Yes	Yes	Yes	Yes
Rename a program in ..	Yes	Yes	Yes	No
Change the comment of a program in ..	Yes	Yes	Yes	No
Change protections of a program in ..	Yes	Yes	Yes	No
Execute a part-program in ..	Yes	Yes	Yes	Yes
Execute a user program in ..	Yes	No	No	No
Execute the PLC program in ..	Yes	*	No	No
Execute programs using the GOTO or RPT instructions from ..	Yes	Yes	Yes	No
Execute subroutines stored in ..	Yes	No	No	No
Execute programs stored in RAM, CARD A or HD using the EXEC instruction from ..	Yes	Yes	Yes	Yes
Execute programs via DNC with the EXEC instruction from ..	Yes	Yes	Yes	No
Open programs stored in RAM, CARD A or HD using the OPEN instruction from ..	Yes	Yes	Yes	Yes
Open programs via DNC using the OPEN instruction from ..	Yes	Yes	Yes	No
Consult from a PC and through Ethernet, the program directory in ...	No	No	Yes	No
Consult from a PC and through Ethernet, the subroutine directory in ...	No	No	No	No
Create from a PC and through Ethernet, a directory in...	No	No	No	No

* If it is not in RAM memory, it generates an executable code in RAM and it executes it..

1.2 MONITOR INFORMATION LAYOUT

The monitor is divided into the following areas or display windows:



1.- This window indicates the selected operating mode, as well as the program number and the number of active blocks.

The program status is also indicated (in execution or interrupted) and if the DNC is active.

2.- This window indicates the time in the " hours : minutes : seconds " format.

3.- This window displays the Messages sent to the operator from the part program or via DNC.

The last message received will be shown regardless of where it has come from.

4.- This window will display messages from the PLC.

If the PLC activates two or more messages, the CNC will always display the one with the highest priority, which is the message with the smallest number. In this way, MSG1 will have the highest priority and MSG128 will have the lowest.

In this case the CNC will display the character **+** (plus sign), indicating that there are more messages activated by the PLC, it being possible to display them if the ACTIVE MESSAGE option is accessed in the PLC mode.

In this window the CNC will also display the character ***** (asterisk), to indicate that at least one of the 256 user-defined screens is active.

The screens which are active will be displayed, one by one, if the ACTIVE PAGES option is accessed in the PLC mode.

5.-Main window.

Depending on the operating mode, the CNC will show in this window all the information necessary.

When a CNC or PLC error is produced the system displays this in a superimposed horizontal window.

The CNC will always display the most important error and it will show:

* The "down arrow" key to indicate that another less important error has also occurred and to press this key to view its message.

* The "up arrow" key to indicate that another more important error has also occurred and to press this key to view its message.

6.-Editing window.

In some operating modes the last four lines of the main window are used as editing area.

7.-CNC communications window (errors detected in edition, nonexistent program, etc.)

8.-This window displays the following information:

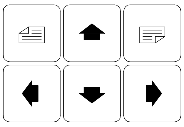
SHF	Indicates that the SHIFT key has been pressed to activate the second function of the keys. For example, if key 9 is pressed after the SHIFT key, the CNC will understand that the "$" character is required.
CAP	This indicates capital letters (CAPS key). The CNC will understand that capital letters are required whenever this is active.
INS/REP	Indicates if it is insert mode (INS) or substitution (REP) mode. It is selected by means of the INS key.
MM/INCH	Indicates the unit system (millimeters or inches) selected for display.

9.-Shows the different options which can be selected with soft-keys F1 thru F7.

1.3 *KEYBOARD LAYOUT*

Depending on the utility of each key, the CNC keyboard may be divided as follows:

Alphanumeric keypad for data entry into memory, axes selection, tool compensation, etc.

	Keys to scroll the information displayed on the screen back and forth, page up and down or a line at a time and to move the cursor throughout it.
CL or CLEAR	To delete the character where the cursor is or the last one entered if the cursors is at the end of the line.
INS	To choose between "insert" and "overwrite" mode.
ENTER	To validate the CNC and PLC commands generated in the editing window.
HELP	To access the help system from any operating mode.
RESET	To initialize the history of the program being executed by assigning to it the values set by machine parameter. The program must be stopped in order for the CNC to accept this key.
ESC	To return to the previous option of the operation shown on the screen.
MAIN MENU	To go directly to the main menu of the CNC.
RECALL	In conversational modes, it assigns the value of a coordinate to the selected field.
PPROG	In conversational modes, to access the list of part-programs stored in memory.
F1 to F7	Softkeys or function keys to select the different operating options shown on the screen.

Specific keys to select canned cycles in MC and TC modes.

There are also the following special keyboard sequences:

SHIFT RESET	The result of this keystroke sequence is the same as turning the CNC off and back on. This option must be used after modifying the machine parameters of the CNC so they can be effective.
SHIFT CL	With this keystroke sequence, the CRT screen goes blank. Press any key to recover its normal display.
	If while the screen is off, an error message or a PLC or CNC message comes up, the screen will recover its normal display.

SHIFT

To display at the right side of the screen the position of the axes and the status of the program in progress.

It may be used in any operating mode.

Press the same keystroke sequence to recover the previous display.

1.3.1 EDIT, SIMUL AND EXEC KEYS

The keyboards with a new look for M and T models have 3 new keys:

"EDIT" to access the editing mode directly.
"SIMUL" to access the simulation mode directly.
"EXEC" to access the execution mode directly.

At the MC, TC and TCO models, these direct accesses are available when operating in M or T mode (non-conversational). To access them, use the "P.PROG" key instead of "EDIT" and "GRAPHICS" instead of "SIMUL".

Direct access to the editing mode through the "EDIT" key

By pressing this key in the editing and simulation modes, the program simulated or executed last may be edited.

When pressing this key in any other work mode, it goes to editing the program edited last.

If the relevant program is being executed or simulated, the one edited last will be edited.
If there is no previous program, it will request the name of a new program.

To restrict the editing function to the last program edited, simulated or executed, set the NEXEDI variable to one of the following values:

NEXEDI	=0	no restriction. It opens the last one edited, simulated or executed.
NEXEDI	=1	Always the program edited last
NEXEDI	=2	Always the program simulated last
NEXEDI	=3	Always the program executed last

If the relevant program is being executed or simulated, it will display a warning.
If there is no previous program, it will request the name of a new program.

Direct access to the simulation mode through the "SIMUL" key

When pressing this key, it starts simulating the program handled (edited, simulated or executed) last.

If there is no previous program, it will request the name of a new program.

If the simulation or execution mode is active, only the active mode will be displayed, no program is selected.

To restrict the simulation function to the program edited, simulated or executed last, set the NEXSIM variable to one of the following values:

NEXSIM	=0	no restriction. It opens the program edited, simulated or executed last
NEXSIM	=1	always the program edited last
NEXSIM	=2	always the program simulated last
NEXSIM	=3	always the program executed last

If the relevant program is being executed or simulated, it will display a warning.
If there is no previous program, it will request the name of a new program.

Direct access to the execution mode through the "EXEC" key

When pressing this key, it starts executing the program handled (edited, simulated or executed) last.

If there is no previous program, it will request the name of a new program.

If the simulation or execution mode is active, only the active mode will be displayed, no program is selected.

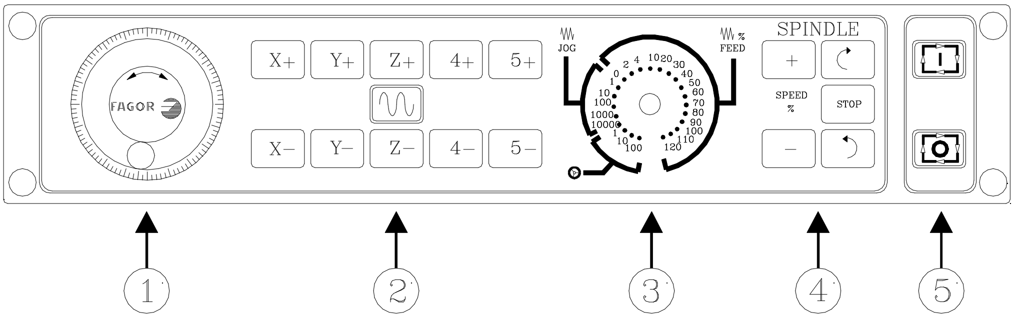
To restrict the simulation function to the program edited, simulated or executed last, set the NEXEXE variable to one of the following values:

NEXEXE	=0	no restriction. It opens the program edited, simulated or executed last
NEXEXE	=1	always the program edited last
NEXEXE	=2	always the program simulated last
NEXEXE	=3	always the program executed last

If the relevant program is being executed or simulated, it will display a warning.
If there is no previous program, it will request the name of a new program.

1.4 OPERATOR PANEL LAYOUT

According to the utility which the different parts have, it can be considered that the Operator Panel of the CNC is divided in the following way:



1.- Position of the emergency button or electronic handwheel.

2.- Keyboard for manual movement of axes.

3.- Selector switch with the following functions:

Select the multiplication factor of the number of pulses from the electronic handwheel (1, 10 or 100).

Select the incremental value of the movement of the axes in movements made in the "JOG" mode.

Modify the programmed axis feedrate between 0% and 120%

4.- Keyboard which allows the spindle to be controlled, it being possible to activate it in the desired direction, stop it or vary the programmed turning speed between percentage values established by means of spindle machine parameters "MINSOVR" and "MAXOVR", with an incremental step established by means of the spindle machine parameter "SOVRSTEP".

5.- Keyboard for CYCLE START and CYCLE STOP of the block or program to be executed.

2. *OPERATING MODES*

After turning on the CNC, or after pressing the sequence of SHIFT-RESET keys, the FAGOR logo will appear in the main window of the monitor or the screen previously prepared as page 0 by means of the GRAPHIC EDITOR.

If the CNC shows the message **" Initialize? (ENTER / ESC) "**, it should be borne in mind that after pressing the ENTER key, all the information stored in memory and the machine parameters are initialized to default values indicated in the installation manual.

On the lower part of the screen the main CNC menu will be shown, it being possible to select the different operating modes by means of the softkeys F1 thru F7.

Whenever the CNC menu has more options than number of softkeys (7), the character "+" will appear in softkey f7. If this softkey is pressed the CNC will show the rest of the options available.

The options which the main CNC menu will show after turning it on, after pressing the key sequence SHIFT-RESET or after pressing the "MAIN MENU" softkey are:

EXECUTE Allows the execution of part programs in automatic or single block.

SIMULATE Allows simulation of parts programs in several modes.

EDIT Allows editing new and already-existing part programs.

JOG Allows manual control of the machine by means of the Control Panel keys.

TABLES Allows CNC tables relating to part programs (Zero Offsets, Tool Offsets, Tools, Tool Magazine and global or local arithmetic parameters) to be manipulated.

UTILITIES Allows program manipulation (copy, delete, rename, etc.)

STATUS It shows the CNC status and that of the DNC communication lines. It also lets activate and deactivate the communication with a PC through DNC.

DNC Allows communication with a computer via DNC to be activated or deactivated.

PLC Allows operation with the PLC (edit the program, monitor, change the status of its variables, access to the active messages, errors, pages, etc).

GRAPHIC EDITOR Allows, by means of a simple graphics editor, the creation of user-defined screens (pages), which can later be activated from the PLC, used in customized programs or presented when the unit is powered on (page 0).

MACHINE PARAMETERS Allows the machine parameters to be set to adapt the CNC to the machine.

DIAGNOSIS Makes a test of the CNC.

While the CNC is executing or simulating a part program it allows any other type of operating mode to be accessed without stopping the execution of the program.

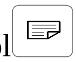
In this way it is possible to edit a program while another is being executed or simulated.

It is not possible to edit the program which is being executed or simulated, nor execute or simulate two part programs at the same time.

2.1 *HELP SYSTEMS*

The CNC allows access to the help system (main menu, operating mode, editing of commands, etc.) at any time.

To do this, you must press the HELP key and the corresponding help page will be shown in the main window of the screen.

If the help consists of more than one page of information, the symbol  indicating that this key can be pressed to access the following page or the  indicating that it is possible to press this key to access the previous page.

The following help is available:

*** OPERATING HELP**

This is accessed from the operating mode menu, or when one of these has been selected but none of the options shown have been selected. In all these cases, the softkeys have a blue background color.

It offers information on the operating mode or corresponding option.

While this information is available on screen it is not possible to continue operating the CNC via the softkeys, it being necessary to press the HELP key again to recover the information which was on the main screen before requesting help and continuing with the operation of the CNC.

The help system can also be abandoned by pressing the ESC key or the MAIN MENU key.

*** EDITING HELP**

This is accessed once one of the editing options has been selected (part programs, PLC program, tables, machine parameters, etc.) In all these cases, the softkeys have a white background.

It offers information on the corresponding option.

While this information is available, it is possible to continue operating with the CNC.

If the HELP key is pressed again, the CNC analyzes if the present editing status corresponds to the same help page or not.

If another page corresponds to it, it displays this instead of the previous one and if the same one corresponds, it recovers the information which was in the main window before requesting help.

The help menu can also be abandoned after pressing the ESC key, to return to the previous operating option, or the MAIN MENU key to return to the main menu.

* CANNED CYCLES EDITING HELP

It is possible to access this help when editing a canned cycle.

It offers information on the corresponding canned cycle and an editing assistance for the selected canned cycle is obtained at this point.

For the user's own cycles a similar editing assistance can be obtained by means of a user program. This program must be prepared with screen customizing instructions.

Once all the fields or parameters of the canned cycle have been defined the CNC will show the information which exists in the main window before requesting help.

The canned cycle which is programmed by means of editing assistance will be shown in the editing window, and the operator can modify or complete this block before entering it in memory by pressing the ENTER key.

Editing assistance can be abandoned at any time by pressing the HELP key. The CNC will show the information which existed on the main window before requesting help and allows programming of the canned cycle to continue in the editing window.

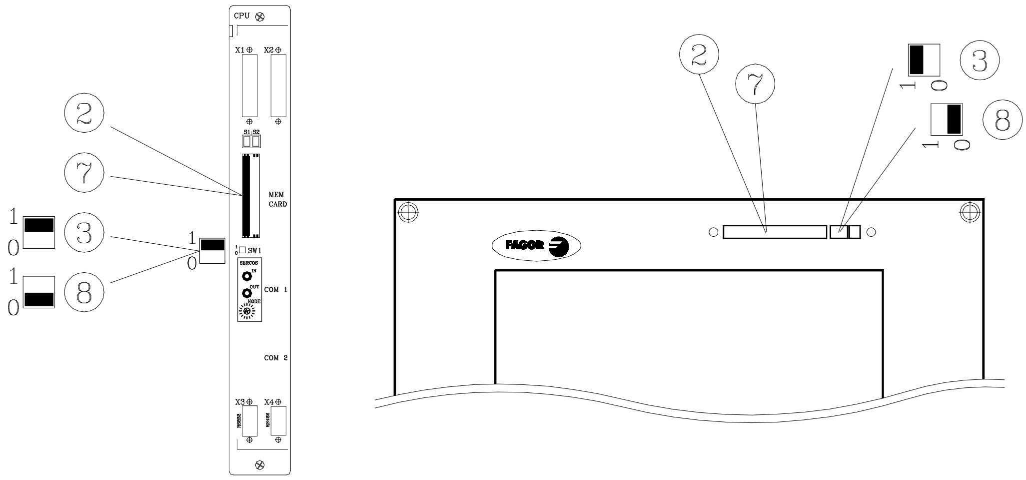
The help menu can also be abandoned after pressing the ESC key, to return to the previous operating option, or the MAIN MENU key to return to the main menu.

2.2 SOFTWARE UPDATE

Use the slot occupied by the "Memkey Card"

1- Turn the CNC off

2.- Remove the "Memkey Card" and insert the "Memory Card" that contains the new software version.



3.- Set the SW1 switch to "1".

4- Turn the CNC on.

The screen will show the software updating page with the following information:

Installed version and New version

Checksum of the installed version and that of the new one.

5.- Press the [Update software] softkey

The CNC will display the various stages of the software updating process and their status.

When done with the updating process, the CNC will display a new screen with the steps to follow.

6.- Turn the CNC off

7.- Remove the "Memory Card" and insert the "Memkey Card".

8.- Set the SW1 switch to “0”.

9- Turn the CNC on. The software version is now updated.

Notes:

With the memory card that contains the software version, the CNC CANNOT execute anything.

If the CNC is turned on with the "Memkey Card" in and the SW1 switch set to "1", the CNC does not come on, but its data is NOT affected

3 *EXECUTE / SIMULATE*

The EXECUTE operating mode allows the execution of part programs in automatic mode or in single block mode.

The SIMULATE operating mode allows the simulation of part-programs in automatic or single block mode.

When selecting one of these operating modes, one must indicate the location of the part-program to be executed or simulated.

The part program may be stored in the CNC's internal RAM memory, in the "Memkey Card", in PC connected through serial line 1 or 2, or in the hard disk (HD module).

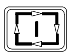
After pressing one of these softkeys, the CNC displayes the corresponding part-program directory.

The program may be selected by:

- Keying in its number and pressing [ENTER] or
- Positioning the cursor of the scren over the desired program and pressing [ENTER].

When wished to SIMULATE a part-program, the CNC will request the type of simulation to be carried out as shown on the next page.

The executing or simulating conditions (fist block, type of graphics, etc.) may be set before executing or simulating the part-program. These conditions may also be modified if the execution or simulation is interrupted.

To execute or simulate a part-program, press 

Note: To switch to JOG mode once executed or simulated a part program (or a section of it), the CNC will maintain the machining conditions (type of movement, feedrates, etc.) selected while executing or simulating it.

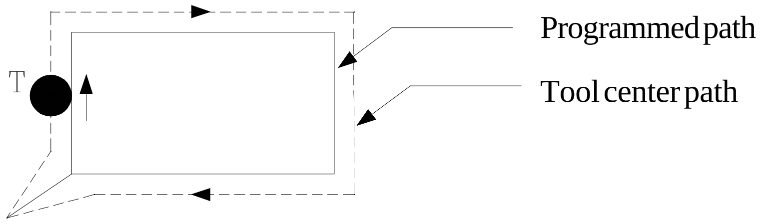
The executing or simulating conditions (fist block, type of graphics, etc.) that may be set before executing or simulating the part-program are:

THEORETICAL PATH

- It ignores tool radius compensation (functions G41, G42) thus showing the graphic representation of the programmed path.
- It does not output the M, S, T function to the PLC.
- It does not move the machine axes or start the spindle.

G FUNCTIONS

- It takes into account tool radius compensation (functions G41, G42) thus showing the graphic representation of the path for the tool center.



- It does not output the M, S, T functions to the PLC.
- It does not move the machine axes or start the spindle.

G, M, S, T FUNCTIONS

- It takes into account tool radius compensation (functions G41, G42) thus showing the graphic representation of the path for the tool center.
- It outputs the M, S, T functions to the PLC.
- It does not move the machine axes or start the spindle.

RAPID

- It takes into account tool radius compensation (functions G41, G42) thus showing the graphic representation of the path for the tool center.
- It outputs the M, S, T functions to the PLC.
- It starts the spindle if it has been programmed.
- The axes are moved at maximum feedrate allowed F0 regardless of the programmed F value and it can be varied using the Feedrate Override Switch.

RAPID [S=0]

- It takes into account tool radius compensation (functions G41, G42) thus showing the graphic representation of the path for the tool center.
- It does not start the spindle.
- It does not output the M functions associated with the spindle when operating in open loop (rpm): M3, M4, M5, M41, M42, M43 and M44.
- It does output to the PLC the M function associated with spindle orientation (M19) when operating in closed loop.
- It outputs to the PLC the rest of the functions M, S, T.
- The axes, "C" axis included, are moved at maximum feedrate F0 regardless of the programmed F value and it can be varied using the Feedrate Override Switch.

	Graphics	Axis movement	Spindle control	Outputs M, S, T functions to the PLC	Outputs functions M3-4-5 M41-42-43-44
Theoretical path	Programmed	No	No	No	No
G functions	Tool center	No	No	No	No
G, M, S, T functions	Tool center	No	No	Yes	Yes
Rapid	Tool center	Yes	Yes	Yes	Yes
Rapid [S=0]	Tool center	Yes	No	Yes	No

The executing or simulating conditions (initial block, type of graphics, etc.) that may be set before or while executing or simulating a part-program are:

BLOCK SELECTION

It allows selecting the block in which the execution or the simulation of the program will start.

STOP CONDITION

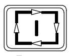
It allows selecting the block in which the execution or the simulation of the program will stop.

DISPLAY SELECTION

It allows the display mode to be selected.

MDI

It allows any type of block (ISO or high level) to be edited with programming assistance by means of softkeys.

Once a block has been edited and after pressing the  key (cycle start), the CNC will execute this block without leaving this operating mode.

TOOL INSPECTION

Once the execution of the program has been interrupted, this option allows the tool to be inspected and changed should this be necessary.

GRAPHICS

This option carries out a graphic representation of the part during the execution or simulation of the selected part program.

It also allows selecting the type of graphic, the area to be displayed, the viewpoint and graphic parameters.

SINGLE BLOCK

Allows the part program to be executed one block at a time or continuously.

3.1 BLOCK SELECTION AND STOP CONDITION

The CNC will start to execute the required block from the first line of the program and will finish it when one of the program end functions M02 or M30 is executed.

If it is required to modify one of these conditions the **BLOCK SELECTION** and **STOP CONDITION** functions must be used.

BLOCK SELECTION

With this option it is possible to indicate the beginning block of the selected program execution or simulation. This cannot be used when the CNC is already executing or simulating the selected program.

When this option is selected, the CNC will show the selected program since the initial block must always belong to this program.

The operator must select with the cursor the block where the execution or simulation of the program will be started.

To do this, the cursor can be moved line by line with the up and down arrow keys or page by page with the page-up and page-down keys.

The "find" softkey options are also available:

<u>BEGINNING</u>: By pressing this key, the cursor will position at the first line of the program.

<u>END</u>: By pressing this key, the cursor will position at the last line of the program.

<u>TEXT</u>: With this function it is possible to search for a text or character sequence starting at the current cursor position.

When this softkey is pressed, the CNC requests the character sequence to be found.

Once this text has been keyed in, press the "**END OF TEXT**" softkey and the cursor will position over the first occurrence of the keyed text.

The found text will be highlighted and it will be possible to continue (by pressing "**ENTER**") with the search all along the program or quit by pressing either the "**ESC**" key or "**ABORT**" softkey.

The search can be done as many times as it is desired. Once searched to the end of the program, it will continue the search from the beginning.

When quitting the search mode, the cursor will be positioned at the last matching text found.

<u>LINE NUMBER</u>: After pressing this key, the CNC will request the number of the line to be found. Key in the desired line number and press **ENTER**. The cursor will, then, be positioned at the desired line.

Once the desired starting block is selected, press **ENTER** to validate it.

STOP CONDITION

With this option it is possible to indicate the final execution or simulation block of the selected program. This cannot be used when the CNC is already executing or simulating the selected program.

When selecting this option, the CNC will show the following softkey functions:

PROGRAM SELECTION

This option will be used when the final execution or simulation block belongs to another program or to a subroutine resident in another program.

The CNC shows the part-program directory of the RAM memory. Use the cursor to select the desired program and press ENTER.

Then, carry out the BLOCK SELECTION as described next.

BLOCK SELECTION

Use the cursor to select the last program block to be executed.

Use the up and down arrow keys or page by page with the page-up and page-down keys.

The "find" softkey options are also available:

BEGINNING: By pressing this key, the cursor will position at the first line of the program.

END: By pressing this key, the cursor will position at the last line of the program.

LINE NUMBER: After pressing this key, the CNC will request the number of the line to be found. Key in the desired line number and press **ENTER**. The cursor will, then, be positioned at the desired line.

Once the desired final block has been selected, press **ENTER** to validate it.

NUMBER OF TIMES

This function will be used to indicate that the execution or simulation of the selected program must stop after executing the “end block” a specific number of times.

When selecting this function, the CNC will request the number of times to be executed or simulated.

If a canned cycle or a call to a subroutine has been selected as the end block of the program, the CNC will stop after executing the complete canned cycle or the indicated subroutine.

If the selected block has a number of block repetitions, the program will stop after doing all the repetitions indicated.

3.2 DISPLAY

With this option, it is possible to select the most appropriate display mode at any time even during execution or simulation of a part program.

The display modes available at the CNC and which can be selected with softkeys are:

STANDARD
POSITION
PART PROGRAM
SUBROUTINES
FOLLOWING ERRORS
USER
EXECUTION TIMES

All the display modes have a window at the bottom of the CRT which shows the history with the conditions in which machining is being done. The information shown is as follows:

F and **%** Programmed feedrate and selected feedrate OVERRIDE %.

When Feed-hold is active, the feedrate value appears highlighted.

S and **%** Programmed spindle speed and selected spindle OVERRIDE %

T Number of active tool.

D Number of active tool offset.

NT Number of the next tool

This field will be displayed when having a machining center and it will show the tool being selected but which is waiting for the execution of the M06 to make it active.

ND Tool offset number corresponding to the next tool.

This field will be displayed when having a machining center and it will show the tool being selected but which is waiting for the execution of the M06 to make it active.

S RPM Real speed of the spindle in RPM.

When working in M19 this indicates the position of the spindle in degrees.

G All displayable G functions which are active.

M All active M functions.

PARTC Parts counter. It indicates the number of consecutive parts executed with the same part-program.

Every time a new program is selected, this variable is reset to "0".

With this CNC variable (PARTC) it is possible to modify this counter from the PLC, from the CNC program and via DNC.

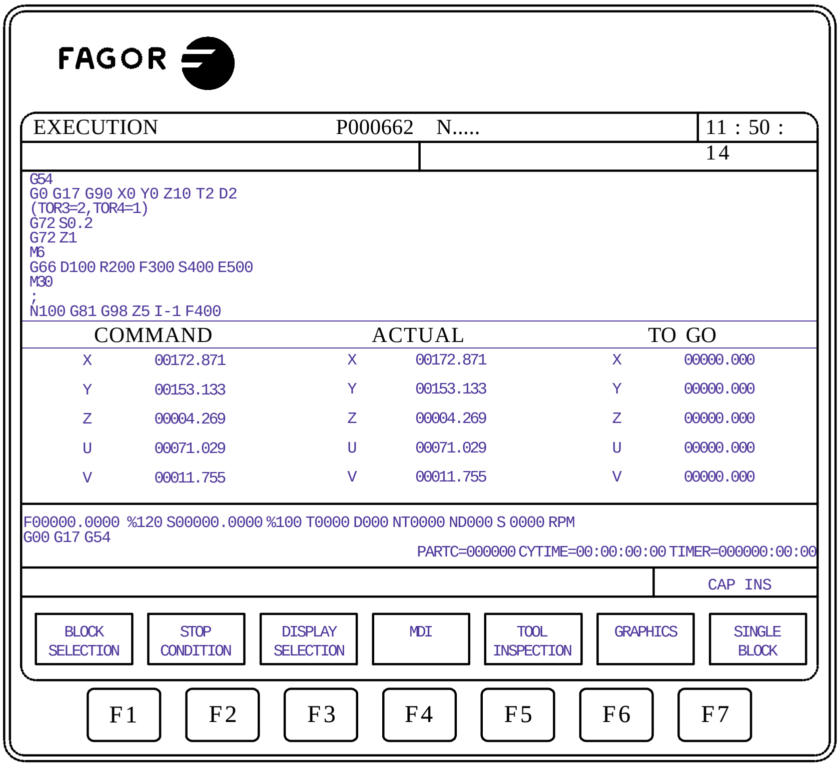
CYTIME Time elapsed during the execution of the part in “hours : minutes : seconds : hundredths of a second” format.

Every time a part-program execution starts, even when repetitive, this variable is reset to "0".

TIMER Time indicated by the PLC-enabled clock in “hours: minutes : seconds” format.

3.2.1 *STANDARD DISPLAY MODE*

This display mode is assumed by default on power-up and after the key sequence SHIFT-RESET and it shows the following fields or windows:



MD031

* A group of program blocks. The first of them is the block being executed.

* The axis coordinates, in real or theoretical values according to the setting of the "THEODPLY" machine parameter and the format defined with the axis machine parameter "DFORMAT".

Each axis is provided with the following fields:

COMMAND. Indicates the programmed coordinate or position value which the axis must reach.

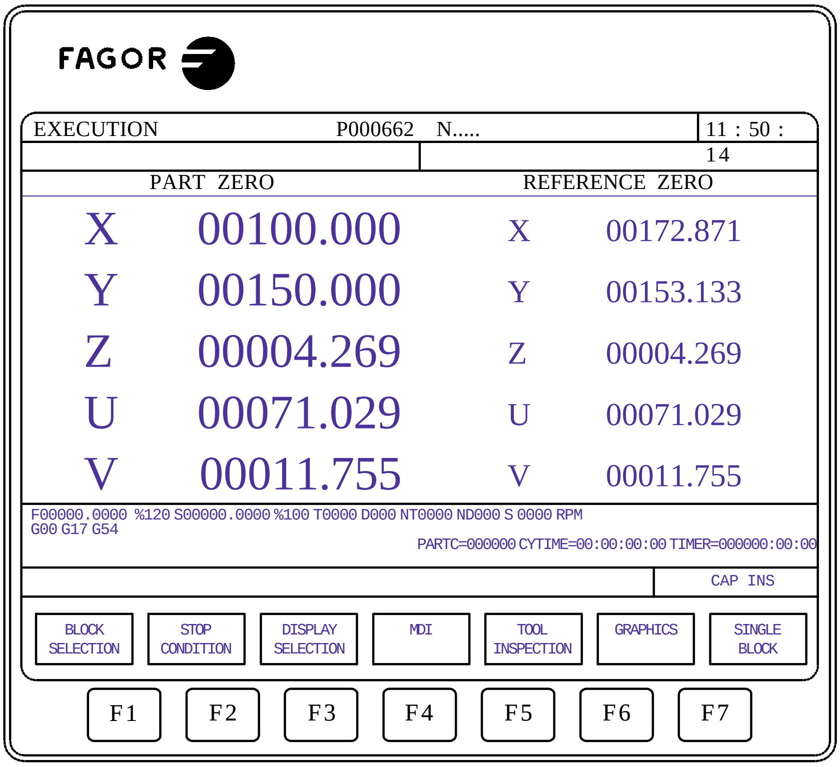
ACTUAL. Indicates the actual (current) position of the axis.

TO GO. Indicates the distance which is left to run to the programmed coordinate.

3.2.2 POSITION DISPLAY MODE

This display mode shows the position values of the axes.

This display mode shows the following fields or windows:



MD032

* The axis coordinates, in real or theoretical values according to the setting of the “THEODPLY” machine parameter and the format defined with the axis machine parameter “DFORMAT”.

Each axis has the following fields:

<u>PART ZERO</u> This field shows the real axis position with respect to part zero.

<u>MACHINE ZERO</u> This field shows the real axis position with the respect to machine reference zero (home).

3.2.3. PART PROGRAM DISPLAY MODE

Displays a page of program blocks among which the block being executed is highlighted.

3.2.4. SUBROUTINE DISPLAY MODE

This display mode shows information regarding the following commands:

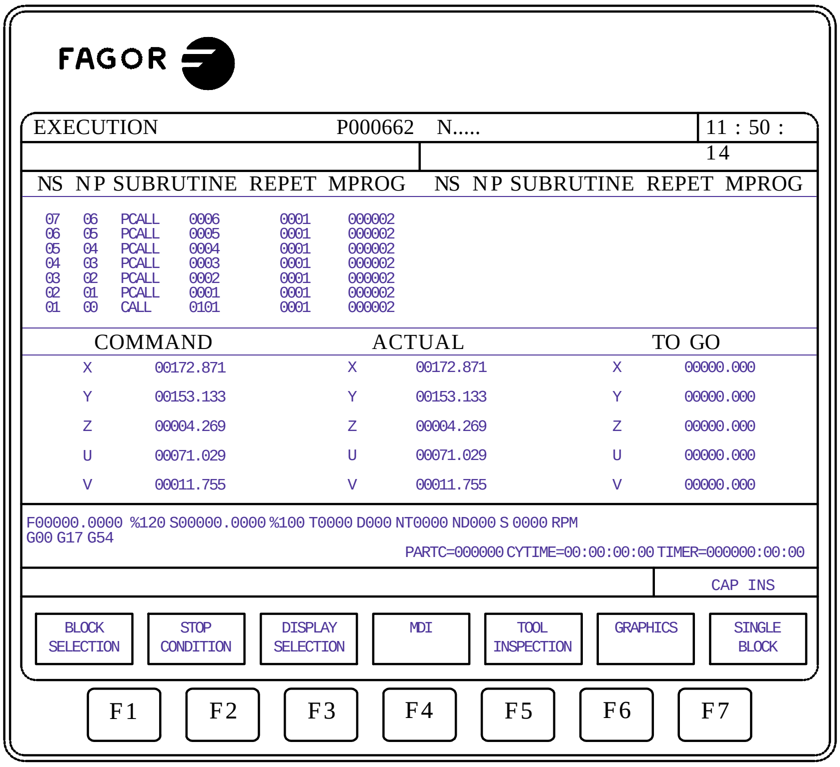
(RPT N10,N20)	This function executes the program section between blocks N10 thru N20.
(CALL 25)	This function executes subroutine number 25.
G87 ...	This function the corresponding canned cycle.
(PCALL 30)	This function executes subroutine 30 in a local parameter level.

When this mode is selected, the following must be considered:

The CNC allows the definition and usage of subroutines which can be called upon from a main program or from another subroutine and this can, in turn, call upon a second one and so forth up to 15 nesting levels (each subroutine call represents a nesting level).

When the machining canned cycles: G66, G68, G69, G81, G82, G83, G84, G85, G86, G87, G88 and G89 are active, they use the sixth nesting level of local parameters.

This display mode shows the following fields or windows:



MD033

* Information on the subroutines which are active.

<u>NS</u> Indicates the nesting level (1-15) which the subroutine occupies.

<u>NP</u> Indicates the level of local parameters (1-6) in which the subroutine is executed.

<u>SUBROUTINE</u> Indicates the type of block which has caused a new nesting level.

Examples: (RPT N10,N20) (CALL 25) (PCALL 30) G87

<u>REPT</u> Indicates the number of times which remain to be executed.

For example, if (RPT N10, N20) N4 is programmed and is the first time that it is being executed, this parameter will show a value of 4.

<u>M</u> If an asterisk is shown (*) this indicates that a Modal subroutine is active in this nesting level, and this is executed after each movement.

<u>PROG</u> Indicates the program number where the subroutine is defined.

* The axis coordinates, in real or theoretical values according to the setting of the "THEODPLY" machine parameter and in the format determined by the axis machine parameter "DFORMAT".

  Each axis is provided with the following fields:

  COMMAND. Indicates the programmed coordinate or position which the axis must reach.

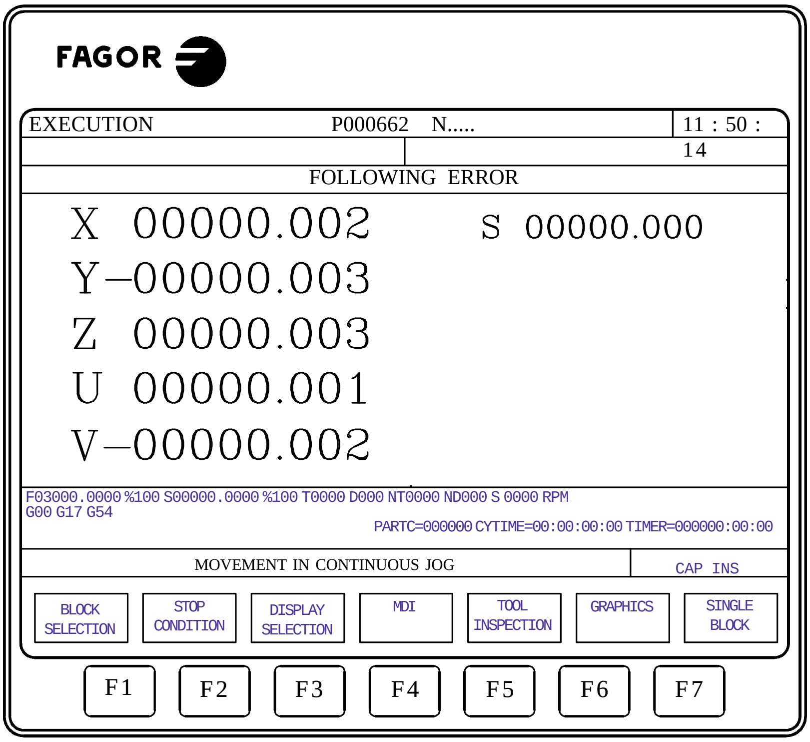
  ACTUAL. Indicates the actual (current) position of the axis.

  TO GO. Indicates the distance which is left to run to the programmed coordinate.

3.2.5 FOLLOWING ERROR DISPLAY MODE

This display mode shows the following error (difference between the theoretical value and the real value of their position) of the axes and the spindle.

Also, when having the tracing option, this mode shows, to the right of the screen, a window with the values corresponding to the tracing probe.



mp1603

The display format is determined by the axis machine parameter “DFORMAT”.

The correction factors of the probe do not depend on the work units.

The display format for the probe deflections on each axis (X, Y, Z) as well as the total deflection "**D**" is set by axis machine parameter "**DFORMAT**".

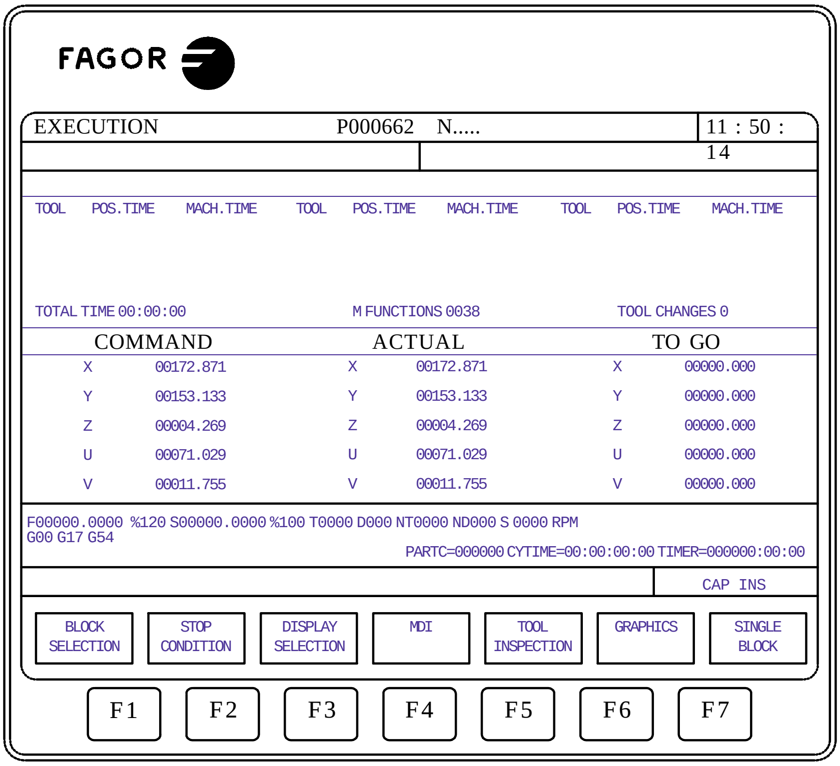
3.2.6 USER DISPLAY MODE

This option will execute the program which is selected by means of the general machine parameter “USERDPLY” in the user channel.

To quit this mode and return to the previous menu, press **ESC**.

3.2.7 EXECUTION TIME DISPLAY MODE

This option is available while simulating a part-program and it will display the following fields or windows:



MD033

* A display window shows the estimated program execution time at 100% of the programmed feedrate.

  This display area shows the following information:

  The time each tool (TOOL) takes to execute the positioning moves (POS.TIME) as well as the machining moves (MACH.TIME) indicated in the program.

  The "TOTAL TIME" required to execute the complete program.

  The "M FUNCTIONS" being executed in the program.

  The number of "TOOL CHANGES" performed during the execution of the program.

* The position values for the axes of the machine.

  It must be borne in mind that the display format for the axes is established by machine parameter "**DFORMAT**" and that real or theoretical position values will be shown depending on the setting of machine parameter "**THEODPLY**".

  Each axis has the following fields:

  COMMAND. Indicates the programmed coordinate or position which the axis must reach.

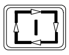
  ACTUAL. Indicates the actual (current) position of the axis.

  TO GO. Indicates the distance which is left to run to the programmed coordinate.

3.3 MDI

This function is not available in the SIMULATION mode. Besides, if a program is being executed, it must be interrupted in order to access this function.

It is possible to execute any block (ISO or high level) and it provides information on the corresponding format via the softkeys.

Once the block has been edited and after the  key has been pressed the CNC will execute this block without quitting this operating mode.

3.4 *TOOL INSPECTION*

This function is not available in the SIMULATION mode. Besides, if a program is being executed, it must be interrupted in order to access this function.

This operating mode allows all the machine movements to be controlled manually, and enabling the axis control keys on the Operator Panel (X+, X-, Y+, Y-, Z+, Z-, 4+, 4-, etc.).

Also, the CNC will show the softkeys to access the CNC tables, edit and execute a block in MDI as well as repositioning the axes of the machine to the position from where this function was called.

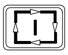
One of the ways to make the tool change is as follows:

* Move the tool to the required tool change position

This move may be made by jogging the axes from the operator panel or in MDI.

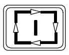
* Gain access to CNC tables (tools. Tool offsets, etc.) in order to find another tool with the similar characteristics.

* Select, in MDI, the new tool as the active one.

* Make the tool change 

This operation will be performed depending on the type of tool changer used. It is possible to execute the tool change in MDI in this step.

* Return the axes to the position where the tool inspection began (REPOSITIONING).

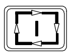
* Continue executing the program ()

Note: If during tool inspection, the spindle is stopped, the CNC will restart it in the same turning direction (M3 or M4) while repositioning.

The CNC offers the following options by means of softkeys:

MDI

Allows to edit blocks in ISO or high level (except those associated with subroutines) providing information on the corresponding format by means of softkeys.

Once the block has been edited and after the  key has been pressed the CNC will execute this block without quitting this operating mode.

TABLES

Allows access to any of the CNC tables associated with part programs (Zero offsets, Tool offsets, Tools, Tool magazine, Global and Local Parameters).

Once the desired table has been selected, all editing commands will be available for its verification and modification.

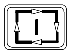
In order to return to the previous menu the **ESC** key must be pressed.

REPOSITIONING.

Positions the axes at the point where tool inspection started.

Once this option is selected, the CNC will show the axes to be repositioned and will request the order in which they will move.

The “PLANE” softkey will appear for the main plane movements and another softkey for each one of the rest of the axes to be repositioned.

Once repositioning has been completed the  key is pressed to continue with the execution of the rest of the program.

3.5 GRAPHICS

With this function it is possible to select the type of graphic to be used as well as to define all the parameters for the corresponding graphic display.

To do so, the CNC must NOT be executing or simulating a part program; otherwise, it must be interrupted.

Once the type of graphics has been selected and its parameters defined, this function can be accessed even during the execution or simulation of a part program should the type of graphic or any graphic parameters be changed

After selecting this function, the CNC will display the following softkey options:

* Type of graphic
* Display area
* Zoom
* Point of view
* Graphic parameters
* Clear Screen
* Deactivate graphics

One of the different ways that could be used to define graphics is the following:

1.- Define the DISPLAY AREA. It will depend on the dimensions of the part and its coordinate values will be referred to the part zero being currently active .

2.- Select the TYPE OF GRAPHICS to be displayed.

3.- Define the VIEWPOINT to be used. This option is available in types of graphics such as 3D and SOLID.

4.- Select the drawing colors to be used by means of the GRAPHIC PARAMETERS.

Once the part-program execution or simulation has been started, it is possible to interrupt it and define another type of graphic or select another graphic display area by means of the ZOOM option.

3.5.1 TYPE OF GRAPHICS

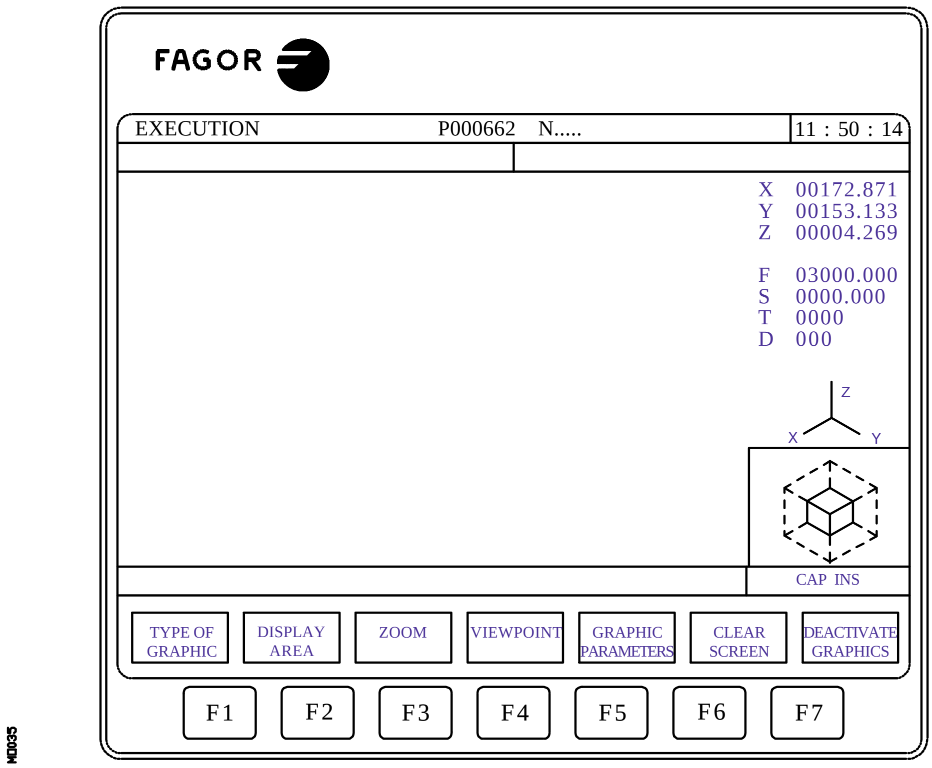
This CNC offers two types of graphics: line and solid graphics. They both are totally independent from each other in such a way that an execution or simulation performed in either one does not affect the other.

The CNC will show all the possible softkey options in order to select one of them.

The type of graphic will remain active until another type is selected or graphics are deactivated (with its corresponding softkey) or the CNC is turned off.

Every time a type of graphic is selected, the CNC recovers all the graphic conditions (zoom, graphic parameters and display area) which were active during the last type of graphic selected.

The selected type of graphics will display the following information to the right of the screen:



* The current real axes position. The tool position values will indicate the position of the tool tip.

* The axes feedrate (F) and the spindle speed (S) currently selected.

* The active tool (T) and tool offset (D).

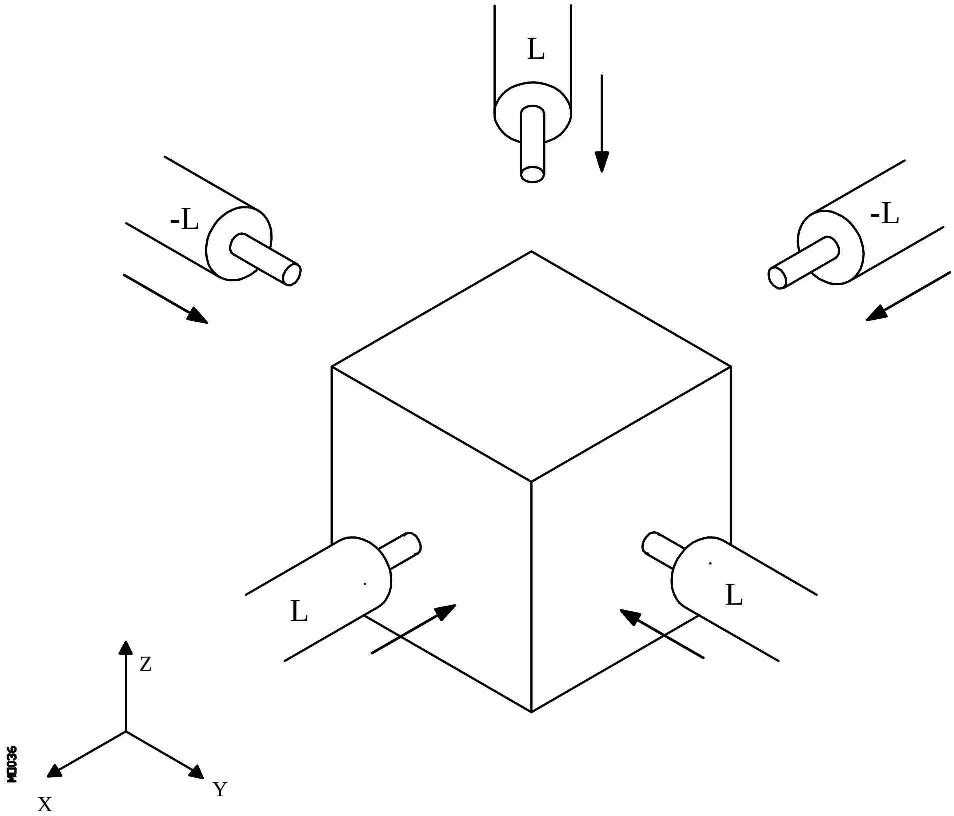
* The point of view used for the graphic display. It is defined by the X, Y, Z axes and it can be modified by means of the VIEWPOINT softkey.

* Two cubes or rectangles depending on the type of point of view selected.

  The cube, whose sides are colored, indicates the graphic area currently selected and the one drawn only with lines shows the size of display area being selected.

  When the point of view shows a single cube side or when the selected type of graphics corresponds to one of the XY, XZ or YZ planes, the CNC will display two rectangles indicating the graphic area (colored rectangle) and the display area being selected (non-colored rectangle).

This CNC will display all machining operations performed with the tool along either the X, Y or Z axis except when the tool is along the Z axis and the part is being machined on its negative side (in the -Z to +Z direction).



When simulating a part-program, the CNC analyzes the value assigned to the tool length in the corresponding tool offset.

If this value is positive, the graphic display is performed on the positive side of the part. (in the + to - direction) and if negative, it will be performed on the negative side of the part (in the - to + direction).

It must be borne in mind that the CNC will assume a value of L0 as positive. Also, if no tool has been defined during execution or simulation, the CNC will take L0 and R0 as default values.

LINE GRAPHICS

This type of graphics draws the tool path on the selected planes (XY, XZ, YZ) by means of color lines.

The possible types of line graphics are:

3D Displays a three-dimensional view of the tool path.

XY,XZ,YZ Display the tool path on the selected plane.

COMBINED VIEW This option divides the screen in four quadrants displaying in them the XY, XZ, YZ and 3D views simultaneously.

The generated graphics is lost in the following circumstances:

* When clearing the screen (softkey: CLEAR SCREEN).
* When deactivating graphics (softkey: DEACTIVATE GRAPHICS).
* When selecting a new type of solid graphic (top view or solid)

SOLID GRAPHICS

This type of graphics offer the same information in two different ways: as a three-dimensional solid (SOLID) or as a section view of the part (SECTION VIEW).

When simulating or executing a program in any of these modes, it is possible to display its graphics in either mode.

The section view is usually drawn faster than the solid view, therefore, it is recommended to first run the program in section view and then switch to solid graphics. The end result will be the same.

The graphic generated after executing or simulating a program is lost in the following instances:

* When clearing the screen (softkey CLEAR SCREEN).
* When deactivating the graphics (softkey DEACTIVATE GRAPHICS).
* When selecting a new line graphics (3D, XY, XZ, YZ, Combined).

SECTION VIEW

This option displays a section view of the part on the XY plane drawn in different shades of gray which indicate the depth of the part.

The other plane views are also shown (XZ and YZ) which correspond to the sections indicated by the vertical and horizontal indicator lines.

These vertical and horizontal indicator lines can be moved left and right or up and down respectively by means of the corresponding arrow keys.

These indicator lines can be moved at any time even while executing or simulating the part-program and the CNC will display “live” the new sections corresponding to the new indicator line positions.

Once the execution or simulation has finished or it has been interrupted, the CNC redraws the section view in order to achieve a better color definition and better sense of depth.

This type of graphics will not show the machining operations performed with the tool positioned along the X or Y axis but only when positioned along the Z axis. However, when switching to SOLID, afterwards, all machining operations will be shown.

SOLID

This option shows a three-dimensional block which will be “machined” as the part-program is being run.

If no tool has been selected while executing or simulating the part-program, the CNC will assume a default tool offset value of L0, R0. With these values, the CNC will only show the programmed tool path and the block will not be “machined” since the tool is assumed to have no radius (R0).

The screen refresh is done periodically depending on the simulation speed and always from left to right regardless of the movement direction of the tool.

It must be borne in mind that when executing or simulating a new program (other than the current one), it will be “machined” over the existing “already-machined” block. However, a new “unmachined” block can be obtained by deleting the screen with the CLEAR SCREEN softkey.

3.5.2 DISPLAY AREA

In order to use this option, the CNC must not be executing or simulating a part-program. If so, it must be interrupted.

With this option it is possible to define the display area by assigning the desired values to maximum and minimum coordinates for each axis. These coordinate values must be referred to part zero.

This maximum and minimum coordinate assignment will be done in the windows displayed to the right of the screen which show their current values.

Use the up and down arrow keys to select the desired field whose value is to be changed.

Once the desired values for all the desired fields have been keyed in, press **ENTER** to validate them.

To quit this mode without making any changes, press **ESC**.

While SOLID GRAPHICS or SECTION VIEW is selected, it must be borne in mind that if a new display area is defined, the CNC will reset the graphic representation returning to its initial status, "unmachined".

In linear graphics (3D, XY, XZ, YZ, combined) there is a softkey [optimum area] which redefines the display area that contains, in all planes, all the tool paths already executed.

Every time a new display area is defined, it redraws the machining executed up to that point. If the number of points to be redrawn exceeds the amount of memory reserved for it, only the last points will be redrawn and the older ones will be lost.

In solid graphics, the image will only be redrawn when having a Power PC card.

In certain applications such as punch presses, when only using XY graphics, it is recommended to set "minimum Z = 0" and "maximum Z = 0.0001". This way, the top view will only show the XY plane (the XZ and YZ planes will not be shown).

3.5.3 ZOOM

In order to use this option, the CNC must not be executing or simulating a part-program. If so, it must be interrupted.

With this option, it is possible to enlarge or reduce the graphics display area. It cannot be used in either COMBINED VIEW or SECTION VIEW types of graphics.

When selecting this option, the CNC will show a window superimposed on the current graphics and another one over the drawing at the lower right-hand side of the screen. These new windows indicate the new display area being selected.

Use the [zoom +] and [zoom-] keys to either enlarge or reduce the size of the new display area and the arrow keys to move the zoom window around to the desired location on the screen.

By pressing the softkey [INITIAL VALUE], it assumes the values set by means of [DISPLAY AREA]. The CNC shows that value, but it does not quit the zoom mode.

Once the new display area has been defined, press **ENTER** to validate the new values.

Press **ESC** to quit this ZOOM mode without making any changes to the initial values.

Every time a Zoom is carried out, it redraws the machining executed up to that point. If the number of points to be redrawn exceeds the amount of memory reserved for it, only the last points will be redrawn and the older ones will be lost.

In solid graphics, the image will only be redrawn when having a Power PC card.

3.5.4 VIEWPOINT

In order to use this option, the CNC must not be executing or simulating a part-program. If so, it must be interrupted.

This option can be used with any three-dimensional graphics (3D, COMBINED VIEW or SOLID) and it allows to change the point of view (perspective) of the part by shifting the X, Y and Z axes.

When selecting this option, the CNC will highlight the current viewpoint on the right-hand side of the screen.

Use the right and left arrow keys to rotate the XY plane around the Z axis up to 360°.

Use the up and down arrow keys to tilt the Z axis up to 90°.

Once the new viewpoint has been selected, press **ENTER** to validate it.

If SOLID GRAPHICS was selected before or it is selected again, the CNC will refresh the screen showing the same part but from the new viewpoint (with new perspective).

When the selected type of graphics is 3D or COMBINED VIEW, the CNC will maintained the current drawing. The new viewpoint will be applied when executing the next blocks. These blocks will be drawn over the existing graphics. However, the screen can be cleared by using the CLEAR SCREEN softkey in order to start drawing with an “unmachined” part.

To quit this mode without making any changes, press **ESC**.

3.5.5 *GRAPHIC PARAMETERS*

This function can be used any time, even during part program execution or simulation:

With this function it is possible to modify the simulation speed and the colors used to draw the tool paths.

The modifications made to any parameter are immediately assumed by the CNC and can be made during the execution or simulation of the part program.

The softkey options displayed by the CNC are:

SIMULATION SPEED

With this option it is possible to modify the percentage of the speed used by the CNC to execute the part programs in the simulation modes.

The CNC will display a window at the top right-hand side of the screen indicating the current % of simulation speed.

This value can be modified by using the right and left arrow keys. Once the desired value is selected, press **ENTER** to validate the new value.

Press **ESC** to quit this function without making any changes to this field.

It is also possible to change the simulation speed while it is redrawing after a zoom. This lets you check the machining of a particular operation.

PATH COLORS

With this option it is possible to modify the colors used to draw the various tool paths in the execution and simulation modes. They can only be used in line graphics XZ. The available parameters are:

The color for representing rapid moves
The color for representing path without compensation
The color for representing path with compensation
The color for representing threading
The color for representing canned cycles

The CNC will show a series of windows for the definition of graphics parameters.

Among the various colors to choose from, there is a black or "transparent" one. If this one is chosen for a particular path, this path will not be displayed on the screen.

If any of them is to be modified, first select the corresponding window using the up and down keys and then use right and left arrow keys to select the desired color.

Once the desired colors have been selected, press **ENTER** to validate the new choices or **ESC** to ignore the changes and leave this function with the original values intact.

SOLID COLORS

With this option it is possible to modify the colors used in the three-dimensional solid graphics. These colors will be considered when in execution or simulation and will only be used in SOLID graphics mode. The available parameters are the following:

Color for the external X side
Color for the external Y side
Color for the external Z side
Color for the internal X side, machined side
Color for the internal Y side, machined side
Color for the internal Z side, machined side

The CNC will show to the right of the screen a series of windows to select these parameters indicating as well the colors currently selected.

Among the various color choices, the black one indicates that the machining operations done with this color will not be shown graphically (invisible).

To modify any of these parameters, select the corresponding field by using the up and down arrow keys and use the right and left arrow keys to select the color within the desired field or window.

Once the desired colors for the desired solid sides have been selected, press **ENTER** to validate them.

Press **ESC** to quit this color selection mode without making any changes to the original settings.

3.5.6 CLEAR SCREEN

In order to use this function, no part program may be in execution or simulation. If this is the case, it must be interrupted.

Erases the screen or graphic representation shown.

If the solid graphic mode is selected, it will return to its initial status showing the unmachined part.

3.5.7 DEACTIVATE GRAPHICS

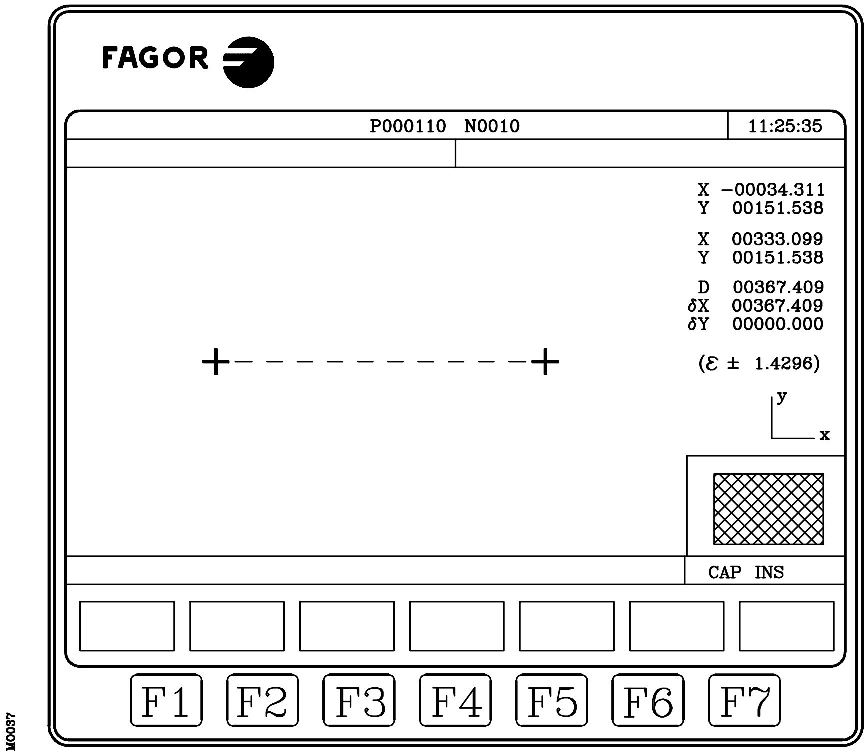
It allows the graphic representation to be deactivated at any time, even during execution or simulation of a part program.

To activate this function again, the “GRAPHICS” softkey must be pressed again. To do this, the CNC must not be executing or simulating a part program. If this is the case, it must be interrupted.

3.5.8 MEASURE

To use this function, a "Line Graphics" (planes XY, XZ or YZ) must be selected and the CNC must not be executing or simulating the part-program. If it is, it must be interrupted.

Once this function is selected, the CNC shows the following information on the screen:



The center of the CRT shows a dotted line with two cursors, the section to be measured. Also, the right-hand side of the screen shows:

* The coordinates of those two cursors with respect to part-zero.

* The distance "D" between them and the components of this distance along the axes of the selected plane " δ X" and " δ Y".

* The cursor step " ε " corresponding to the selected display area. It is given in the work units, millimeters or inches.

The CNC shows the selected cursor and its coordinates in red.

To select the other cursor, press the "+" or "-" key. The CNC shows the new selected cursor and its coordinates in red.

To move the selected cursor, use the up, down, right and left arrow keys.

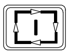
Also, with the keystroke sequences: Shift-Up arrow, Shift-Down arrow, Shift-Right arrow and Shift-Left arrow, it is possible to move the cursor to the corresponding end.

To quit this command and return to the graphics menu, press [**ESC**]

Also, if [key] is pressed, the CNC exits this work mode and returns to the graphics menu.

3.6 SINGLE BLOCK

When actuating on this option, the CNC toggles between single block mode and continuous run mode. This function can be used at any time, even during the execution or simulation of a part program.

If the single block mode is selected, the CNC will only execute one line of the program every time the  is pressed.

The upper window of the screen will show the selected mode of operation. If continuous execution, no message will appear and if SINGLE BLOCK, it will display the message: SINGLE BLOCK.

4. EDIT

This operating mode will be used to edit, modify or look at a part-program stored in the CNC's RAM memory.

To edit a part-program stored in the "Memkey Card" (CARD A) or in the hard disk (HD), it must be previously copied into RAM memory.

To edit a part-program, enter the program number (up to 6 digits) from the keyboard or by selecting it with the cursor from the CNC's part-program directory and then pressing **ENTER**.

Move the cursor on the screen line by line with the “up and down” arrow keys or page by page with the “page up” and “page down” keys.

Once the program number has been entered, the CNC will display the softkeys for the following options:

EDIT *(See section 4.1)*
To edit new lines in the selected program.

MODIFY *(See section 4.2)*
To modify an existing line of the program.

FIND *(See section 4.3)*
To search a string of characters within a program.

REPLACE *(See section 4.4)*
To replace a string of characters with another.

DELETE BLOCK *(See section 4.5)*
To delete a block or group of blocks.

MOVE BLOCK *(See section 4.6)*
To move a block or group of blocks within a program.

COPY BLOCK *(See section 4.7)*
To copy a block or group of blocks to another program position.

COPY TO PROGRAM *(See section 4.8)*
To copy a block or group of blocks into a different program.

INCLUDE PROGRAM *(See section 4.9)*
To insert the contents of another program into the one currently selected.

EDITOR PARAMETERS *(See section 4.10)*
To select the editing parameters (automatic numbering and axes for Teach-in editing).

4.1 EDIT

With this option it is possible to edit new lines or blocks of the selected program.

Select with the cursor the block after which the new ones will be added and press the softkey corresponding to one of the available editing modes.

CNC LANGUAGE .... *(See section 4.1.1)*
The program is edited in ISO code or high level language.

TEACH-IN .... *(See section 4.1.2)*
The machine is jogged to the desired position and, then, the new axis position may be assigned to the block.

INTERACTIVE .... *(See section 4.1.3)*
Editing mode assisted by the CNC.

PROFILES .... *(See section 4.1.4)*
To edit a new profile
After defining the known profile data, the CNC generates its corresponding ISO-coded program.

PROFILE SELECTION
To modify an existing profile.
The CNC requests the first and last blocks of the profile.
Once they are both defined, the CNC will show the corresponding graphics.
Section 4.1.4 describes how to operate with the profile.

USER
When selecting this option, the CNC will execute, in the user channel, the customizing program selected by general machine parameter "USEREDIT". *(See section 4.1.1)*
This is edited in ISO-code or high level language.

4.1.1 EDITING IN CNC LANGUAGE

A program will be edited block by block and each block can be written either in ISO code or high level language or it can be just a program comment.

Once this option has been selected, the softkeys will change colors and they will appear over white background showing the information corresponding to the type of editing possible at that point.

Also, editing help will be available at any time by just pressing the **HELP** key. To quit this help mode, press **HELP** again.

If ESC is pressed while editing a block, the block editing mode is abandoned and the block currently being edited will not be added to the program.

Once the block has been edited, press **ENTER.** This new block will be added to the program after the one indicated by the cursor.

The cursor will position over the new edited block and the editing area (window) will be cleared so another block can be written.

To quit the block editing mode, press **ESC** or **MAIN MENU.**

4.1.2 *TEACH-IN EDITING*

It is basically identical to the previous option (editing in CNC language), except what regards the programming of position coordinate values.

This option shows the current position values of each one of the axes of the machine.

It permits to enter the axes position values from the CNC keyboard (as when editing in CNC language) or, also, use the TEACH-IN editing format as described next.

* Jog the machine axes with the jogging keys or with the electronic handwheel up to the desired position.

* Press the softkey corresponding to the axis to be defined.

* The CNC will assign to this axis its current physical position as the program position value.

Either position value programming methods can be used at any time while defining a block.

When the block being edited has no information (empty editing area or window), the **ENTER** key may be pressed in which case the CNC will generate a new block with the current position values of the axes.

This block will be added automatically to the program and it will be inserted after the block indicated by the cursor.

The cursor will position over the new edited block and the editing area will be cleared so another can be written.

When the position values of <u>all</u> the axes are <u>not</u> to be programmed in this fashion, the CNC permits to select the desired axes. To do this, in this operating mode and within the “EDITOR PARAMETERS” option there is a soft key for “**TEACH-IN AXES**”

4.1.3 *INTERACTIVE EDITOR*

This editor leads the operator through the program editing process by means of questions he/she will answer.

This type of editing offers the following advantages:

* No knowledge of the CNC programming language is required.

* The CNC only admits the data it is requesting, thus no erroneous data can be entered.

* The programmer has, at all times, the appropriate programming aide by means of screens and messages.

When selecting this option, the CNC displays in the main window, a series of graphic options selectable by softkey.

If the selected option has more menus, the CNC will keep showing new graphic options until the desired one is selected.

From this moment, the information corresponding to this option will appear in the main window and it will start requesting the data necessary to program it.

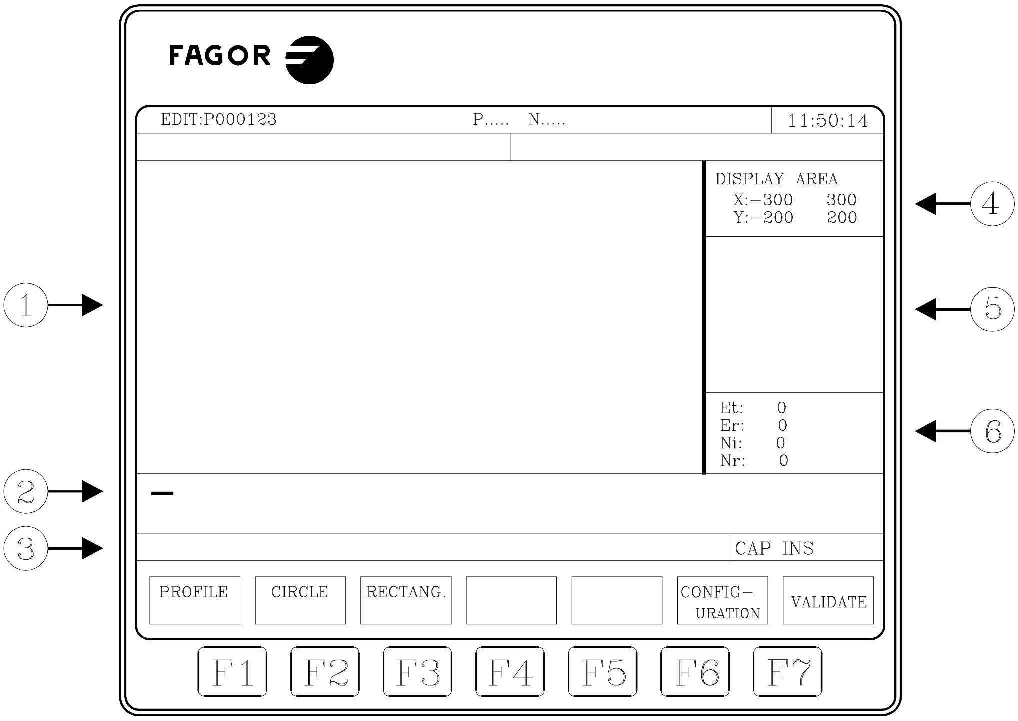
As the requested data is entered, the editing window will show, in CNC language, the block being edited.

The CNC will generate all necessary blocks and it will add them to the program once the editing of this option is done and it will insert them after the one indicated by the cursor.

The main window will show again the graphic options corresponding to the main menu being possible to continue editing the program.

4.1.4 *PROFILE EDITOR*

When selecting this option, the CNC displays the following fields or windows:



1.- Window showing the graphic representation of the profile being edited.

2.- Editing window showing the new generated block in CNC language.

3.- Area for editing messages.

4.- Display area

Indicates the area of the plane shown in the graphic representation of the profile. Indicated by the maximum and minimum position values of each axis.

The way to select this display are is described later on.

5.- Display area for the profile section currently selected for editing or modifying.

It may be the starting block, straight line, a clockwise arc or a counterclockwise arc.

6.- Display area for additional information. It shows a series of parameters for internal use and whose meanings are:

Et : Total elements of the profile
Ec : Complete elements
Ni : Number of data entered
Nr : Number of required data

4.1.4.1 OPERATION WITH THE PROFILE EDITOR

Several profiles may be edited without quitting the profile editor. To edit a profile, proceed as follows:

1.- Select a point of the profile as its beginning point.

2.- Break the profile into straight and curve sections.

If the profile has corner roundings, chamfers, tangential entries or exits, take one of the following actions:

- Treat them as individual sections when having enough information to define them.

- Ignore them when defining the profile and, once done defining the whole profile, select the corners showing those characteristics and enter the corresponding radius value.

CONFIGURATION

Use the [abscissa axis] and [ordinate axis] softkeys to select the editing plane.

The Autozoom function indicates whether the CNC recalculates the graphics display area or not when the edited lines go beyond it.

PROFILE

For editing any profile.

CIRCLE

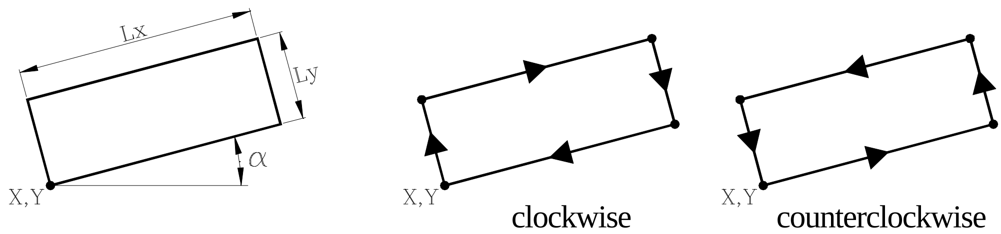
For a quick circular profile definition. If the starting point (X,Y) is not defined, the CNC assumes one.

The [Profile Direction] softkey indicates whether the profile is programmed clockwise or counterclockwise. This data is very important for later modifications and profile intersection. Every time this softkey is pressed, the text at the top of the middle right window changes.

STRAIGHTANGLE

For a quick straight angular profile definition.

The [Profile Direction] softkey indicates whether the profile is programmed clockwise or counterclockwise. This data is very important for later modifications and profile intersection. Every time this softkey is pressed, the text at the top of the middle right window changes.



A straight angular profile is defined with a single command, but the CNC internally breaks into 4 straight segments.

4.1.4.2 PROFILE EDITING

When pressing the [PROFILE] softkey, the CNC requests the starting point of the profile. To define it, use the corresponding softkeys.

For example, if when working in the XY plane the new desired starting point is (20,50):
[X] 20 [ENTER]
[Z] 50 [ENTER]

The values may be set by means of a numeric constant or by means of any expression. Examples:

X 100
X 10 * cos 45
X 20 + 30 * sin 30
X 2 * (20 + 30 * sin 30)

Once the starting point has been set, press the [VALIDATE]

The CNC will show a filled circle in the graphics area to indicate the starting point of the profile.

Also, the softkeys will show the following options:

[STRAIGHT LINE] To edit a straight section.

[CLOCKWISE ARC] To edit a clockwise arc.

[COUNTERCLOCKWISE ARC] To edit a counterclockwise arc.

[CORNERS] To insert roundings, chamfers, tangential entries and exits.

[MODIFY] To modify the starting point.
Modify any profile element, even the type of element (straight line, clockwise or counterclockwise arc)
Insert a new element (straight line or arc) in any position of the profile.
Delete any profile element.
Add a new additional text to any section of the profile.
Modify the display area.

[NEW PROFILE] To edit a new profile.

[FINISH] It must be pressed when all the sections of the profile have been defined.
It must be indicated whether the edited profile or profiles must be saved or not.
The CNC quits the profile editor and adds to the program the ISO code corresponding to the profile just edited .

4.1.4.3 *DEFINITION OF A STRAIGHT SECTION*

When pressing the [STRAIGHT LINE] softkey, the CNC displays the data shown on the right margin of this page.

DISPLAY AREA

X: -300 300
Y: -200 200

STRAIGHT LINE

X1: 50.000
Y1: 60.000
X2:
Y2:
? :

TANGENCY: NO

Et: 0
Er: 0
Ni: 2
Nr: 2

X1, Y1 Coordinates of starting point of the line. They cannot be modified because they correspond to the last point of the previous section.

X2, Y2 Coordinates of the end point of the section.

? Angle of the line referred to the abscissa axis.

TANGENCY Indicates whether the line to be drawn is tangent to the previous section or not.

All these parameters need not be defined, but all the known ones should be defined.

To define a parameter, press the corresponding softkey, key in the desired value and press [ENTER].

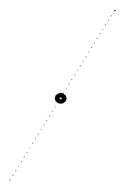
The value may be defined by a numeric constant or by any expression.

Examples:
X 100
X 10 * cos 45
X 20 + 30 * sin 30
X 2 * (20 + 30 * sin 30)

Once all known parameters are set, press the [VALIDATE] softkey and the CNC will show the defined section, if possible.

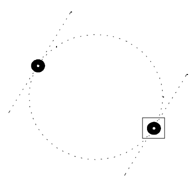
If there is not enough data to show the section, the CNC will show a dotted line indicating its orientation.

Example
X1=0
Y1=0
X2
Y2
? = 60



If there are more than one possibility, all the possible options will be shown and the desired one (framed in red) must be selected using the right and left arrow keys.

Example
X1
Y1
X2
Y2
? = 60
TANGENCY = YES



Use the up and down arrow keys to choose whether all the possible options are shown or only the one framed in red.

Once the desired option is selected, press [ENTER] for the CNC to assume it.

4.1.4.4 DEFINITION OF A CIRCULAR SECTION

When pressing the [CLOCKWISE ARC] or [COUNTERCLOCKWISE ARC] softkey, the CNC displays the data shown on the right margin of this page.

DISPLAY AREA	
X: -300	300
Y: -200	200
CLOCKWISE ARC	
X1:	50.000
Y1:	60.000
X2:	
Y2:	
XC:	
YC:	
RA	
TANGENCY: NO	
Et:	0
Er:	0
Ni:	2
Nr:	2

X1, Y1 Coordinates of the starting point of the arc. They cannot be modified because they correspond to the last point of the previous section.

X2, Y2 Coordinates of the end point of the arc.

XC, YC Coordinates of the arc center.

XC, YC Radius of the arc.

TANGENCY Indicates whether the arc to be drawn is tangent to the previous section or not.

All these parameters need not be defined, but all the known ones should be defined.

To define a parameter, press the corresponding softkey, key in the desired value and press [ENTER].

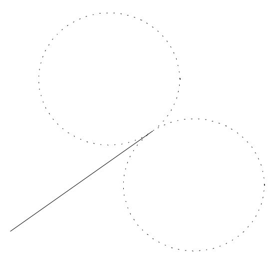
The value may be defined by a numeric constant or by any expression.

Examples:
X 100
X 10 * cos 45
X 20 + 30 * sin 30
X 2 * (20 + 30 * sin 30)

Once all known parameters are set, press the [VALIDATE] softkey and the CNC will show the defined section, if possible.

If there are more than one possibility, all the possible options will be shown and the desired one (framed in red) must be selected using the right and left arrow keys.

Example
X1 = 40
Y1 = 30
X2
Y2
XC
YC
RA = 20
TANGENCY = YES



Use the up and down arrow keys to choose whether all the possible options are shown or only the one framed in red.

If there is not enough data to show the section, the CNC waits for more data in order to solve the profile.

Once the desired option is selected, press [ENTER] for the CNC to assume it.

4.1.4.5 CORNERS

When selecting this option, the CNC shows the following option softkeys:

Rounding	For rounding any corners of the profile.
Chamfer	For adding chamfers at any corner of the profile.
Tangential Entry	To add a tangential tool entry when machining.
Tangential Exit	To add a tangential tool exit at the end of the machining operation.

When selecting one of these, one of the corners of the profiles will appear highlighted.
To select another corner of the same profile, use the up/down and left/right arrow keys.
To select a corner of another profile, use the [page up] and [page down] keys.

To define the rounding, enter the rounding radius and press [ENTER].

To define the chamfer, enter the chamfer radius and press [ENTER].

To define the tangential entry, enter the radius of the path that the tool has to follow when doing a tangential entry and press [ENTER].

To define the tangential exit, enter the radius of the path that the tool has to follow when doing a tangential exit and press [ENTER].

To quit the CORNER mode, press [ESC].

4.1.4.6 MODIFY

When selecting this option, the CNC shows the following softkey options:

Starting Point	To modify the starting point of the profile.
Modify element	To modify any element of the profile, even the type of element (straight lines, clockwise or counterclockwise arcs).
Insert element	To insert a new element (straight line or arc) in any position of the profile.
Delete element	To delete any element of the profile.
Additional Text	To add additional text to any section of the profile.
Configuration	To add a new editing plane or redefine the Autozoom option.
Display area	To change the display area.

When selecting one of these options, one of the profile elements will be highlighted.

To select another element of the same profile, use the up/down and left/right arrow keys.

To select an element of another profile, use the [page up] and [page down] keys.

Starting point

- Select the starting point of the desired profile. The CNC shows the values used to define it.
- Modify the desired values and press the [VALIDATE] softkey.
  If it is the starting point of a "Circular profile" or of a "rectangular profile", it modifies whatever is necessary to leave the profile as it is.

Modify element

- Select the desired element. The CNC shows the values used to define it.
- It is possible to: change the type of section (straight or arc), redefine the existing data, define a new data or delete an existing one.
  A "Circular profile" is treated as a single element and a "rectangular profile" may be treated as a whole profile or each element separately.
- To delete data, press the softkey that defines it and press [ESC]
- Once the element has been modified, press the [VALIDATE] softkey. The CNC recalculates the new profile.

Insert element

- Select the point, or corner, after which the new one is to be inserted.
- Select the type of section (straight or arc), define it and press the [VALIDATE] softkey.
- The CNC recalculates the new profile.

Delete element

- Select the element to be deleted and confirm the command.
  When deleting a circular profile, the whole profile is deleted. When deleting a rectangular profile it is possible to delete the whole profile or each element separately.
- The CNC recalculates the new profile.

Additional text

- Select the desired element. The CNC shows the ISO code corresponding to that section in the editing area.
- Add the desired text. Functions F, S, T, D, M or program comments may be added.
- Press the [VALIDATE] softkey.

Display area

When selecting this option, the following softkey options are shown:

- [Zoom +] to enlarge the image on the screen.
- [Zoom -] to reduce the image on the screen.
- [Optimum area] to show the full profile on the screen.
- The display area may be moved around with the [left arrow], [right arrow], [up arrow] and [down arrow] keys.
- Press the [VALIDATE] softkey. The CNC updates the values indicated in the upper right-hand window (DISPLAYED AREA).

To quit the MODIFY mode, press [ESC].

4.1.4.7 FINISH

This softkey must be pressed once all the sections of the profile have been defined.

The CNC will try to calculate the requested profile by previously solving all the unknowns.

If it finds several possibilities for certain sections, the CNC will show them for each section and the desired option (framed in red) will have to be chosen using the right and left arrow keys.

Once the whole profile has been solved, the CNC will show the code of the part program currently being edited.

The ISO-coded program for the edited profile is contained between these lines:

```
;************************** START **********************
;************************** END **********************
```

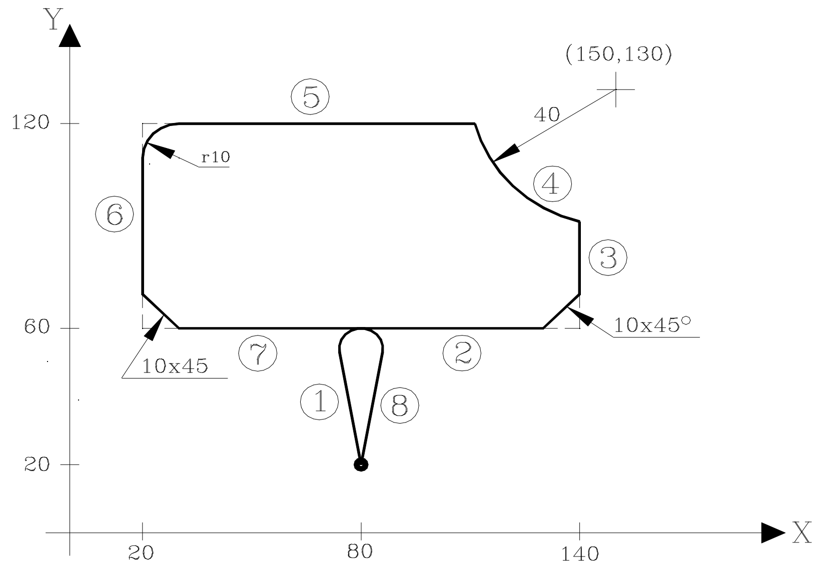
If a profile cannot be solved due to lack of data, the CNC will issue the corresponding error message.

Warning:

When pressing the [FINISH] softkey, the CNC quits the profile editor and adds to the program the ISO-code corresponding to the profile just edited.

To quit the profile editor without changing the part-program, press [ESC] and the CNC will request confirmation of this command.

4.1.4.8 EXAMPLES OF PROFILE DEFINITION



Profile definition without rounding, chamfers, tangential entries or exits.

Abscissa and ordinate of the starting point		X = 80 Y = 20
Section 1	STRAIGHT LINE	X=80 Y=60
Section 2	STRAIGHT LINE	X=140 Y=60
Section 3	STRAIGHT LINE	? =90
Section 4	CLOCKWISE ARC	XC=150 YC=130 Radius = 40
		The CNC shows the possible intersections between sections 3 and 4. Select the correct one.
Section 5	STRAIGHT LINE	X=20 Y=120 ? =180
		The CNC shows the possible intersections between sections 4 and 5. Select the correct one.
Section 6	STRAIGHT LINE	X=20 Y=60
Section 7	STRAIGHT LINE	X=80 Y=60
Section 8	STRAIGHT LINE	X=80 Y=20

Adapt the image to the screen

Select the DISPLAY AREA option and press the [OPTIMUM AREA] softkey.

Definition of roundings, chamfers and tangential entries and exits.

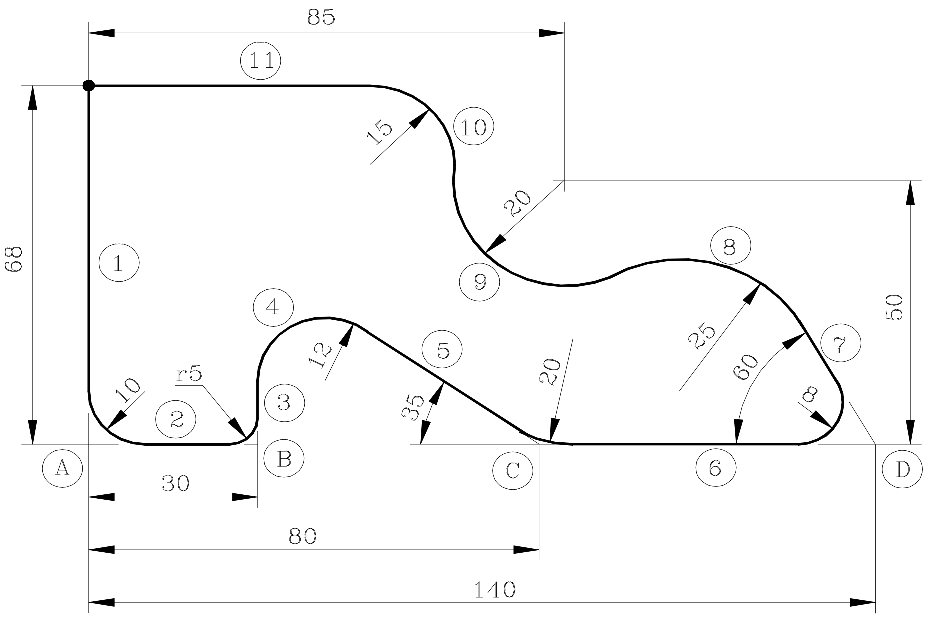
Select the MODIFY option and define:

CHAMFER	Select corner 2-3 and press ENTER. With Radius = 10
ROUNDING	Select corner 5-6 and press ENTER. With Radius = 10
CHAMFER	Select corner 6-7 and press ENTER With Radius = 10
TANGENTIAL ENTRY	
	Select corner 1-2 and press ENTER. With Radius = 5
TANGENTIAL EXIT	
	Select corner 7-8 and press ENTER. With Radius = 5

Press ESC to quit the Modify option.

End of the editing process

Press the [FINISH] softkey. The CNC quits the profile editing mode and the shows the ISO-coded program that has been generated.



Profile definition without rounding

Abscissa and ordinate of the starting point X = 0 Y = 68
Section 1 STRAIGHT LINE X=0 Y=0
Section 2 STRAIGHT LINE X=30 Y=0
Section 3 STRAIGHT LINE ? =90
Section 4 CLOCKWISE ARC RA=12 Tangent = Yes
Section 5 STRAIGHT LINE X=80 Y=0 ? =-35 Tangent = Yes
The CNC shows the possible solutions for section 4. Select the correct one.
Section 6 STRAIGHT LINE X=140 Y=0
Section 7 STRAIGHT LINE ? =120
Section 8 COUNTERCLOCKWISE ARC RA=25 Tangent = Yes
Section 9 CLOCKWISE ARC XC=85 YC=50 RA=20 Tangent = Yes
The CNC shows the possible solutions for section 8. Select the correct one.
Section 10 COUNTERCLOCKWISE ARC RA=15 Tangent = Yes
Section 11 STRAIGHT LINE X=0 Y=68 ? =180 Tangent = Yes
The CNC shows the possible solutions for section 10. Select the correct one.

Adapt the image to the screen

Select the DISPLAY AREA option and press the [OPTIMUM AREA] softkey.

Rounding definition

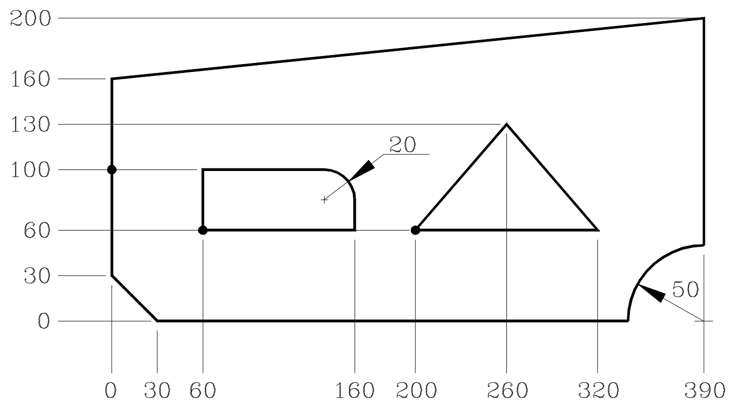
Select the MODIFY option and define:
ROUNDING Select the A corner and press ENTER With Radius = 10
ROUNDING Select the B corner and press ENTER With Radius = 5
ROUNDING Select the C corner and press ENTER With Radius = 20
ROUNDING Select the D corner and press ENTER With Radius = 8
Press ESC to quit the Modify option.

End of the editing process

Select the FINISH softkey. The CNC quits the profile editing mode and shows the ISO coded program that has been generated.

Example of how to define a profile and modify it later:

Configuration
Abscissa axis: X   Ordinate axis: Y   Autozoom: Yes   Validate



Profile (outside profile)

Starting point		X 0	Y 100	Validate	
Straight	X 0	Y 0	Validate		
Straight	X 340	Y 0	Validate		
Clockwise arc		Xf 390	Yf 50	R 50	Validate
	(choose the right arc)				
Straight	X 390	Y 200	Validate		
Straight	X 0	Y 160	Validate		
Straight	X 0	Y 100	Validate		

New Profile (rectangle)

Rectangle   X 60   Y 60   Lx 100   Ly 40   Validate

New Profile (triangle)

Profile

Starting point		X 200	Y 60	Validate
Straight	X 320	Y 60	Validate	
Straight	X 260	Y 130	Validate	
Straight	X 200	Y 60	Validate	

Corners (roundings and chamfers)

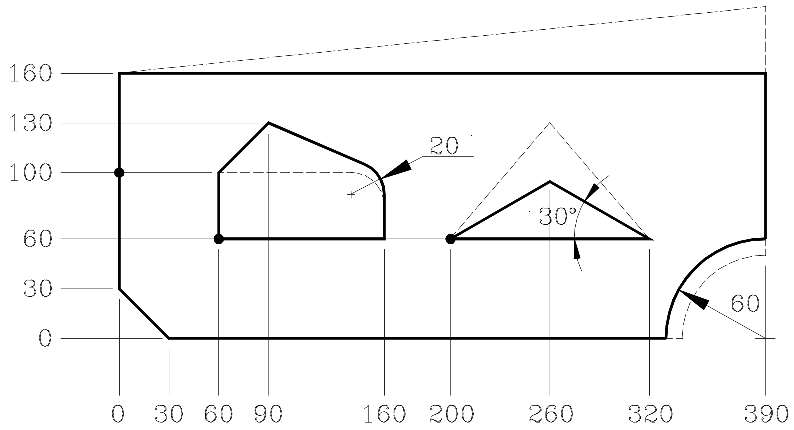
Chamfer

Select the first profile with the page up/down keys.
Select the lower left corner with the up&down and left/right arrow keys   Enter
Chamfer 30
Escape

Rounding

Select the second profile (rectangle) the upper right corner   Enter
Radius 20
Escape
Escape

Modify



(modify first profile)

Modify element
Select the lower line on the first profile Enter
Straight X 330 Validate (choose arc)

Modify - Modify element
Select the arc of the lower right corner Enter
Clockwise arc Yf 60 R 60 Validate (choose arc)

Modify - Modify element
Select right line Enter
Straight Y 160 Validate

(modify second profile)

Modify - Insert element
Select the second profile
Select the theoretical upper right corner Enter
Straight X 90 Y 130 Validate

(modify third profile)

Modify - Modify element.
Select the right side of the triangle on the third profile Enter
Straight Y Escape (to delete) Angle 150 Validate

Finish.

4.2 MODIFY

This option permits modifying the contents of a selected program block.

Before pressing this softkey, select with the cursor the block to be modified .

Once this option is selected, the softkeys will change their color showing their type of modifying option over a white background.

Also, it is possible to get more editing assistance by pressing **HELP**. Press **HELP** again to exit the editing assistance mode.

By pressing **ESC**, the information corresponding to that block and which was shown in the editing area will be cleared. It will then be possible to modify its contents again.

To quit the block modifying mode, press **CL** or **ESC** to clear the editing window and then press **ESC** again. This way, the selected block will not be modified.

Once the block contents have been modified, press **ENTER** so the new contents replace the old ones.

4.3 FIND

This option is used to find a specific text within the selected program.

When selecting this option, the softkeys will show the following options:

BEGINNING This softkey positions the cursor over the first program block which is then selected quitting the "find" option.

END This softkey positions the cursor over the last program block which is then selected quitting the "find" option.

TEXT With this function it is possible to search a text or character sequence starting from the block indicated by the cursor.

When this key is selected, the CNC requests the character sequence to be found.

When the text is defined, press the "END OF TEXT" softkey and the cursor will be positioned over the first occurrence of that text.

The search will begin at the current block.

The text found will be highlighted being possible to continue with the search or to quit it.

Press **ENTER** to continue the search up to the end of the program. It is possible to search as many times as wished and when the end of the program is reached, it will start from the first block.

Press the "EXIT" softkey or the **ESC** key to quit the search mode. The cursor will be positioned where the indicated text was found last.

LINE NUMBER After pressing this key, the CNC requests the number of the block to be found. After keying in the desired number and pressing **ENTER**, the cursor will position over that block which will then be selected quitting the search mode.

4.4 REPLACE

With this function it is possible to replace a character sequence with another throughout the selected program.

When selecting this option, the CNC requests the character sequence to be replaced.

Once the text to be replaced is indicated, press the "WITH" softkey and the CNC will request the character sequence which will replace the previous one.

Once this text is keyed in, press the "END OF TEXT" softkey and the cursor will be positioned over the first occurrence of the searched text.

The search will begin at the current block.

The found text will be highlighted and the following softkey options will appear:

<u>REPLACE</u> Will replace the highlighted text and will continue the search from this point to the end of the program.

If no more occurrences of the text to be replaced are found, the CNC will quit this mode.

If another occurrence of the text is found, it will be highlighted showing the same "replacing" or "not replacing" options.

<u>DO NOT REPLACE</u> Will not replace the highlighted text and will continue the search from this point to the end of the program.

If no more occurrences of the text to be replaced are found, the CNC will quit this mode.

If another occurrence of the text is found, it will be highlighted showing the same "replacing" or "not replacing" options.

<u>TO THE END</u> This function will automatically replace all the matching text from the current block to the end of the program without offering the option of not replacing it.

<u>ABORT</u> This function will not replace the highlighted text and it will quit the "find and replace" mode.

4.5 DELETE BLOCK

With this function it is possible to delete a block or group of blocks.

To delete only one block, just position the cursor over it and press **ENTER**.

To delete a group of blocks, indicate the first and last blocks to be deleted. To do so, follow these steps:

* Position the cursor over the first block to be deleted and press the “INITIAL BLOCK” softkey.

* Position the cursor over the last block to be deleted and press the “FINAL BLOCK” softkey.

  If the last block to be deleted is also the last one of the program, it can also be selected by pressing the “TO THE END” softkey.

* Once the first and last blocks are selected, the CNC will highlight the selected blocks requesting confirmation to delete them.

4.6 MOVE BLOCK

With this option it is possible to move a block or group of blocks by previously indicating the first and last blocks to be moved. To do so, follow these steps:

* Position the cursor over the first block to be moved and press the “INITIAL BLOCK” softkey.

* Position the cursor over the last block to be moved and press the “FINAL BLOCK” softkey.

If the last block to be moved is also the last one of the program, it can also be selected by pressing the “TO THE END” softkey.

To move only one block, the “initial block” and the “final block” will be the same one.

* Once the first and last blocks are selected, the CNC will highlight the selected blocks requesting confirmation to move them.

Then, indicate the block after which this group of blocks must be placed.

* Press the “START OPERATION” softkey to carry out the move.

4.7 COPY BLOCK

With this option it is possible to copy a block or group of blocks by previously indicating the first and last blocks to be copied. To do so, follow these steps:

* Position the cursor over the first block to be copied and press the "INITIAL BLOCK" softkey.

* Position the cursor over the last block to be copied and press the "FINAL BLOCK" softkey.

  If the last block to be copied is also the last one of the program, it can also be selected by pressing the "TO THE END" softkey.

  To copy only one block, the "initial block" and the "final block" will be the same one.

* Once the first and last blocks are selected, the CNC will highlight the selected blocks requesting confirmation to copy them.

  Then, indicate the block after which this group of blocks must be placed.

* Press the "START OPERATION" softkey to carry out this command.

4.8 COPY TO PROGRAM

With this option it is possible to copy a block or group of blocks of one program into another program.

When selecting this option, the CNC will request the number of the destination program where the selected block or blocks are to be copied. After entering the program number press **ENTER**.

Next, indicate the first and last blocks to copy by following these steps:

* Position the cursor over the first block to be copied and press the "INITIAL BLOCK" softkey.

* Position the cursor over the last block to be copied and press the "FINAL BLOCK" softkey.

  If the last block to be copied is also the last one of the program, it can also be selected by pressing the "TO THE END" softkey.

  To copy only one block, the "initial block" and the "final block" will be the same one.

* Once the first and last blocks are selected, the CNC will highlight the selected blocks and will execute the command.

  If the destination program already exists, the following options will be displayed:

* Write over the existing program. All the blocks of the destination program will be erased and will be replaced by the copied blocks.

* Append (add) the copied blocks behind the ones existing at the destination program.

* Abort or cancel the command without copying the blocks.

4.9 INCLUDE PROGRAM

With this option it is possible to include or merge the contents of another program into the one currently selected.

Once this option is selected, the CNC will request the number of the source program to be merged. After keying in that number press **ENTER**.

Next, indicate with the cursor the block after which the source program will be included.

Finally, press the "START OPERATION" softkey to execute the command.

4.10 EDITOR PARAMETERS

With this option it is possible to select the editing parameters used in this operating mode.

The options or parameters available are described here and they are selected by softkeys.

4.10.1 AUTONUMBERING

With this option it is possible to have the CNC automatically number (label) the blocks after the one being edited.

Once this option is selected, the CNC will display the “ON” and “OFF” softkeys to either activate or deactivate this function.

Once this function is activated, the following options will appear on the CRT:

STEP After pressing this softkey, Enter the desired numbering step between two consecutive blocks and press **ENTER**.

The default value is 10.

STARTING After pressing this softkey, Enter the starting block number to be used on the next block to be edited.

The default value is 0.

When setting both parameters, select the STEP first and then the STARTING block number.

Example: STEP = 12, STARTING= 56;
generated blocks: N56, N68, N80,...

Warning:

This function will not number the already existing blocks.

4.10.2 AXES SELECTION FOR TEACH-IN EDITING

Remember that in the TEACH-IN editing mode, the following feature is available:

When the block being edited has no information (editing area empty), the "**ENTER**" key can be pressed. In this case, the CNC will generate a new block with the current position values of the axes.

The option described here, permits the selection of the axes whose position values will be automatically entered in said block.

After pressing the "**TEACH-IN AXES**" softkey, the CNC shows all the axes of the machine.

The operator must eliminate, pressing the corresponding softkeys, the axis or axes not desired. Every time a softkey is pressed, the CNC will eliminate the corresponding axis displaying only the selected ones.

To end this operation, press "**ENTER**".

The CNC will assume from now on and whenever editing in TEACH-IN, the selected axes. To change those values, access this option again and select the new axes.

5. *JOG*

This mode of operation will be used whenever the manual control of the machine is desired.

Once this mode of operation is selected, the CNC allows the movement of all the axes by means of the axes control keys (X+, X-, Y+, Y-, Z+, Z-, 4+, 4-) located on the operator panel, or by means of the electronic handwheel (if available).

This mode of operation offers the following softkey options: 

With the **MDI** option it is possible to modify the machining conditions (type of moves, feedrates, etc.) being selected. Also, the CNC will maintain the ones selected in this mode when switching to **"EXECUTION"** or **"SIMULATION"** modes.

This operating mode offers the following softkey options:

REFERENCE SEARCH

With this option it is possible to perform a home search on the desired axis or axes.

The CNC offers two ways to search the machine reference (home):

* Using the subroutine associated with function G74. The number of this subroutine will defined by the general machine parameter "REFSUB".

* By selecting the axis or axes to be referenced.

Once the Reference search function is selected, the CNC will show a softkey for each axis and the softkey "ALL".

If the "ALL" softkey is selected, the CNC will highlight (in reverse video) the names of all axes and after pressing the key, it will execute the subroutine associated with G74.

On the other hand, to search the reference anywhere from one to all axes at once (without executing the associated subroutine), the softkeys corresponding to those axes must be pressed.

After pressing each softkey, the CNC will highlight the name of the selected axis.

If an unwanted axis has been selected, press **ESC** to cancel that selection and return to select "REFERENCE SEARCH".

Once all the desired axes have been selected, press .

The CNC will start the home search by moving all selected axes at once until the home reference switches for all axes are pressed and, from then on, the CNC will continue the home search one axis at a time.

Warning:

When searching home using the "ALL" softkey, the CNC will maintain the part zero or zero offset active at the time. However, if the axes have been selected one by one, the CNC will assume the "home" position as the new part zero.

PRESET

With this function it is possible to preset the desired axis position value.

Once this option is selected, the CNC will show the softkey corresponding to each axis.

After pressing the softkey of the corresponding axis to be preset, the CNC will request the position value to be preset with.

Press **ENTER** after the value has been keyed in so the new value is assumed by the CNC.

TOOL CALIBRATION

With this function it is possible to calibrate the length of the selected tool by using a part of known dimensions for this purpose.

Before pressing this softkey, the tool to be calibrated must be selected.

The tool calibration will be performed on the selected axis by means of the G15 function as longitudinal axis (by default: the Z axis).

When using a probe for tool calibration, the following machine parameters must be properly set: **"PRBXMIN", "PRBXMAX", "PRBYMIN", PRBYMAX", "PRBZMIN" , "PRBZMAX"** and **"PRBMOVE"**.

Tool calibration without a probe

Follow these steps:

* Press the softkey corresponding to the axis to be calibrated.

* The CNC will request the position value of the known part at the touch point.

  Once this value has been keyed in, press **ENTER** for this value to be assumed by the CNC.

* Jog the tool with the jog-keys (X+, X- Y+, Y-, Z+, Z-, 4+, 4-) until touching the part.

* Press the "**LOAD**" softkey corresponding to this axis.

  The CNC will perform the necessary calculations and it will assign the new value to the selected tool length offset.

Tool calibration with a probe

It may be done in two ways, as described in "calibration without a probe" or as follows:

* Press the softkey which indicates the direction of the tool calibration along the longitudinal axis.

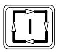
* The CNC will move the tool at the feedrate indicated by the machine parameter for that axis "**PRBFEED**" until touching the probe.

  The maximum distance the tool can move is set by machine parameter "**PRBMOVE**".

* When the tool touches the probe, the CNC stops the axis and, after making the pertinent calculations, it will assign the new tool length value to its corresponding offset.

MDI

With this function it is possible to edit and execute a block (ISO or high-level) providing the necessary information by means of softkeys.

Once the block has been edited, press  to execute it without leaving this operation mode.

Warning:

When searching home "G74", the CNC will maintain the part zero or zero offset active at the time.

USER

When selecting this option, the CNC will execute, in the user channel, the program whose number is indicated in the general machine parameter "USERMAN".

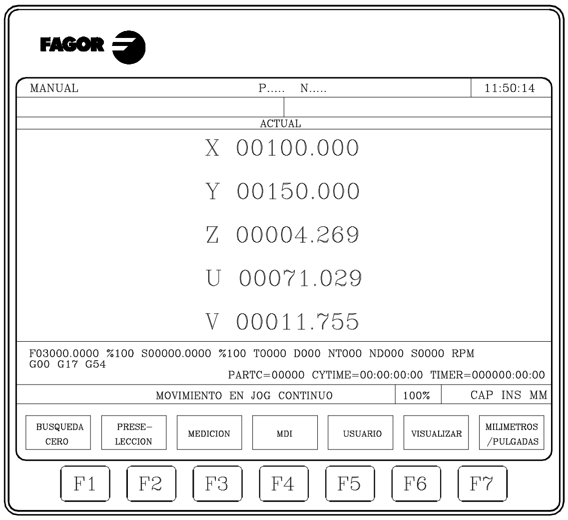
To quit its execution and return to the previous menu, press **ESC**.

DISPLAY

The available display modes are:

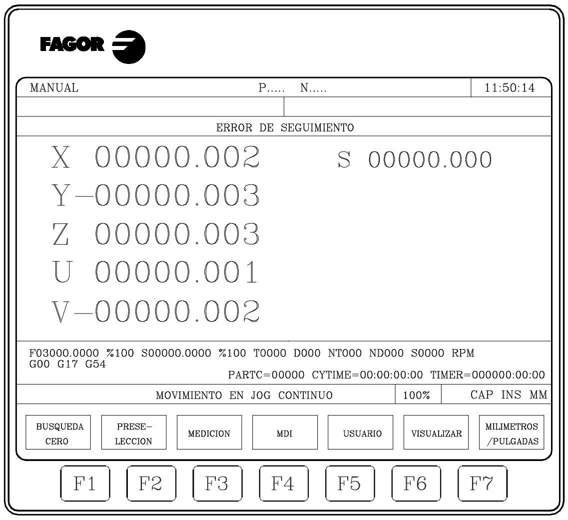
Actual

Shows the current real position of the axes referred to part zero.



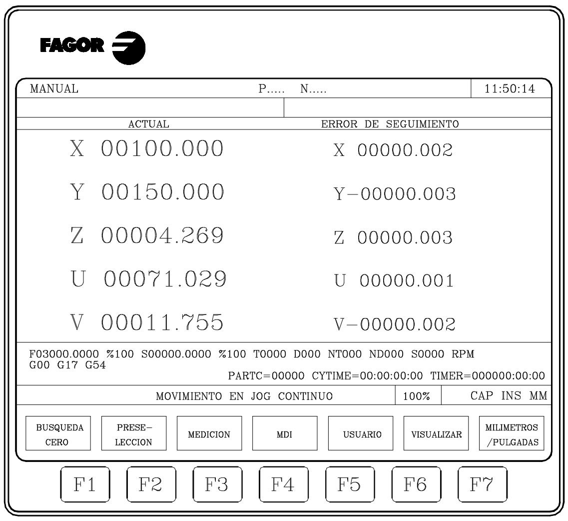
Following error

Difference between the real and the theoretical position of each axis and the spindle.



Actual and following error

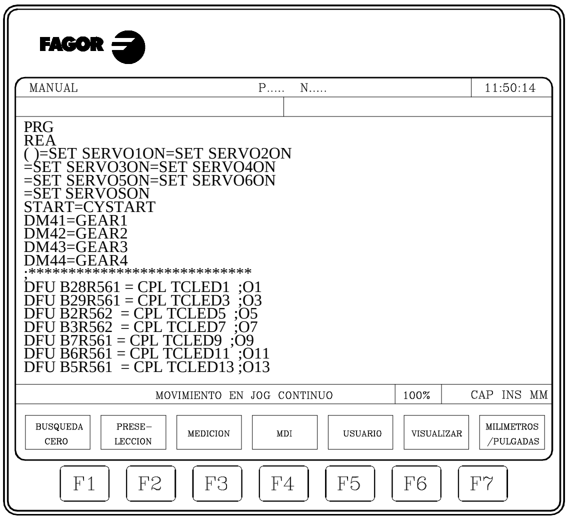
It shows the real position of the axes and their following error.



PLC

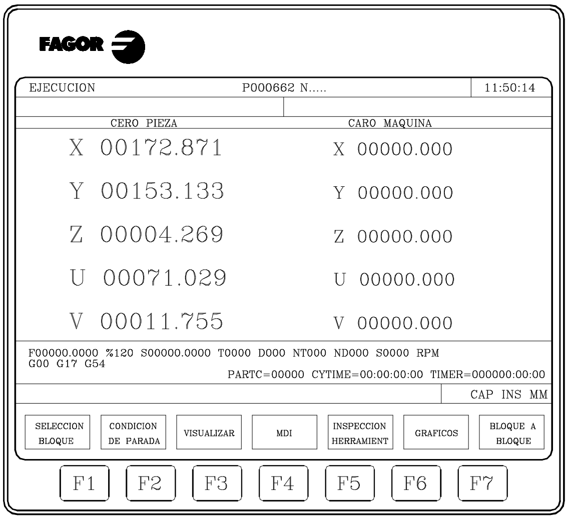
Access the PLC monitoring mode.

Refer to the PLC chapter, monitoring, to know how to use it.



Position

It shows the real position of the axes referred to part zero and to machine zero (home)



MM/INCHES

This softkey toggles the display units for the linear axes from millimeters to inches and vice versa.

The lower right-hand window will indicate which units are selected at all times.

Note that this switching obviously does not affect the rotary axes which are shown in degrees.

5.1 JOGGING THE AXES

5.1.1 CONTINUOUS JOG

Once the % override of the jogging feedrate (indicated by axis-machine parameter “JOGFEED”) has been selected with the switch at the Operator Panel , press the jog keys corresponding to the desired axis and to the desired jogging direction (X+, X-, Y+, Y-, Z+, Z-, 4+, 4- etc.).

The axes can be jogged one at a time and in different ways depending on the status of the general logic input “LATCHMAN”:

* If the PLC sets this mark low, the axes will be jogged while pressing the corresponding Jog key.

** If the PLC sets this mark high, the axes will be jogged from the time the corresponding Jog key is pressed until the key is pressed or another jog key is pressed. In this case, the movement will be transferred to the axis corresponding to the new jog key.

If while jogging an axis, the key is pressed, the axis will move at the feedrate established by machine parameter “G00FEED” for this axis as long as this key stays pressed. When releasing this key, the axis will recover the previous feedrate (with its override %).

5.1.2 INCREMENTAL JOG

It allows to jog the selected axis in the selected direction an incremental step selected by the Feedrate Override switch and at the feedrate indicated by machine Parameter for that axis “JOGFEED”.

The available positions are: 1, 10, 100, 1000 and 10000 corresponding to display resolution units.

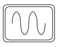
Example:

Display format: 5.3 in mm or 4.4 in inches

Switch position	Movement
1	0.001 mm or 0.0001 inch
10	0.010 mm or 0.0010 inch
100	0.100 mm or 0.0100 inch
1000	1.000 mm or 0.1000 inch
10000	10.000 mm or 1.0000 inch

The maximum permitted step is 10 mm or 1 inch regardless of the selected display format (for example: 5.2 in mm or 4.3 in inches).

After selecting the desired incremental move at the switch, if a jog key is pressed (X+, X-, Y+, Y-, Z+, Z-, 4+, 4- etc.), the corresponding axis will move the selected distance in the selected direction.

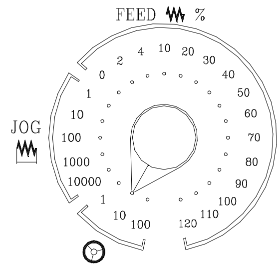
If while jogging an axis, the  key is pressed, the axis will move at a feedrate established by machine parameter “G00FEED” for this axis as long as this key stays pressed. When releasing this key, the axis will recover the previous feedrate (with its override %).

5.1.3 *MOVEMENT BY MEANS OF ELECTRONIC HANDWHEEL*

The various handwheel configurations are:

General handwheel	It can be used to jog any axis one by one. Select the axis and turn the handwheel to move it.
Individual handwheel:	It replaces the mechanical handwheels. Up to 3 handwheels can be used (one per axis). It only moves the axis it is associated with.

To move any of them, turn the switch to any of the handwheel positions.



Positions 1, 10 and 100 indicate the multiplying factor being applied besides the internal x4 to the feedback pulses supplied by the electronic handwheel.
For example, if the manufacturer has set a distance of 0.100 mm or 0.0100 inches per handwheel turn, thus:

Switch position	Distance per turn	
1	0.100 mm	or 0.0100 inch
10	1.000 mm	or 0.1000 inch
100	10.000 mm	or 1.0000 inch

Warning:

When operating with individual handwheels and depending on how fast the handwheel is turned and on the selected handwheel switch position, the CNC might be demanded to move the axis faster than the maximum permitted. In that case, the CNC will move the axis the indicated distance but it will limit the axis speed to that maximum value.

There are 3 operating modes with handwheels:

Standard handwheel:	With the general handwheel, select the axis to be moved and turn the handwheel. With individual handwheels, turn the handwheel associated with the axis to be moved.
Path handwheel:	For chamfering and rounding corners. 2 axes are moved along a selected path (chamfer or rounding) by moving a single handwheel. This feature must be activated via PLC. The general handwheel is assumed as the "path handwheel" or the individual handwheel associated with the X axis.
Feed handwheel	To control the feedrate of the machine. This feature must be activated via PLC.

5.1.3.1 STANDARD HANDWHEEL MODE

With the general handwheel proceed as follows:

1.- Select the axis to be jogged.
Press one of the JOG keys of the axis to be jogged. The selected axis will be highlighted.

When using a FAGOR handwheel with an axis selector button, the axis may be selected as follows:

Push the button on the back of the handwheel. The CNC select the first axis and it highlights it.

When pressing the button again, the CNC selects the next axis and so on in a rotary fashion.

To deselect the axis, hold the button pressed for more than 2 seconds.

2.- Jog the axis

Once the axis has been selected, it will move as the handwheel is being turned and in the direction indicated by it.

With individual handwheels:

Each axis will move as the corresponding handwheel is being turned according to the switch position and in the direction indicated by it.

With several simultaneous handwheels:

The machine may have a general handwheel and up to 3 individual handwheels associated with each axis.
The individual handwheels have priority over the general handwheel. So, if an individual handwheel is moving, the general handwheel will be ignored.

5.1.3.2 PATH HANDWHEEL MODE

With this feature, it is possible to jog two axes at the same time along a linear path (chamfer) or circular path (rounding) with a single handwheel.

The CNC assumes as the path handwheel the general handwheel or, when this one is missing, the one associated with the X axis.

This feature must be handled by the PLC.

To activate or cancle the "Path handwheel" work mode, act upon the logic CNC input "MASTRHND" M5054,

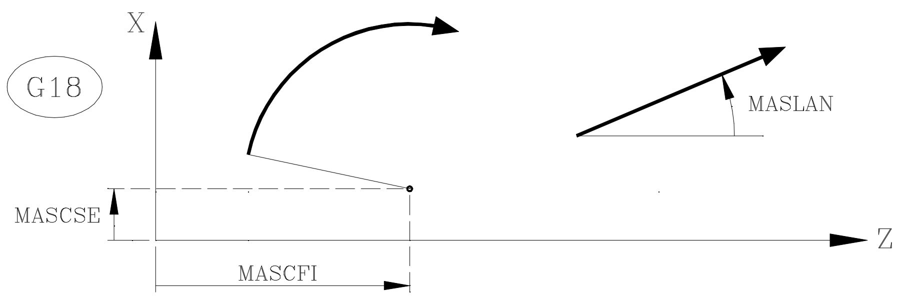
M5054 = 0 Normal handwheels
M5054 = 1 Path Handwheel mode ON.

To indicate the type of movement, act upon the logic CNC input "HNLINARC" M5053,

M5053 = 0 Straight line (linear path)
M5053 = 1 Arc

For a linear path, indicate the angle of the path at the MASLAN variable (value in degrees between the linear path and the firs axis of the plane).

For an arc, indicate the coordinates of the arc center at the MASCFI and MASCSE variables (for the first and second axes of the main plane).



Variables MASLAN, MASCFI and MASCSE can read and written from the CNC, DNC and the PLC .

Simultaneous handwheels

When selecting the Path Handwheel mode, the CNC behaves as follows:

- If there is a General Handwheel, it will be the one working in Path handwheel mode. The individual handwheels, if any, will remain associated with the corresponding axes.
- If there is no General Handwheel, the individual handwheel associated with the X axis then works in Path Handwheel mode.

5.1.3.3 FEED HANDWHEEL

Usually, when making a part for the first time, the machine feedrate is controlled by means of the feedrate override switch.

From this version on, it is also possible to use the machine handwheels to control that feedrate. This way, the machining feedrate will depend on how fast the handwheel is turned.

To do this, proceed as follows:

Inhibit all the feedrate override switch positions from the PLC.
Detect how far the handwheel is turned (reading of pulses received)
Set the corresponding feedrate from the PLC depending on the pulses received from the handwheel.

The following CNC variables return the number of pulses the handwheel has turned.

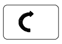
HANPF shows the number of pulses of the 1st handwheel.
HANPS shows the number of pulses of the 2nd handwheel.
HANPT shows the number of pulses of the 3rd handwheel.
HANPFO shows the number of pulses of the 4th handwheel.

Example: The machine has a button to activate and deactivate this feature (feed handwheel) and the feedrate control is carried out with the second handwheel.

```
CY1
R101=0                              Resets the register containing the previous
handwheel reading
END

PRG
DFU I71 = CPL M1000                 Every time the button is pressed, mark M1000 is
inverted
M1000 = MSG1                        If the feature is active, a message is displayed.
NOT M1000                           If the feature is not active
= AND KEYDIS4 $FF800000 KEYDIS4     enables all the positions of the feedrate
override switch
= JMP L101                          and goes on with program execution
If the feature is active
DFU M2009                           and a leading edge (up flank) occurs at the clock
mark M2009
= CNCRD(HANPS,R100,M1)              We read the number of handwheel pulses contained
in R100

= SBS R101 R100 R102                calculates the number of pulses received from the
last reading
= MOV R100 R101                     updates R101 for the next reading
= MLS R102 3 R103                   calculates in R103 the proper % of feedrate
override
= OR KEYDIS4 $7FFFFF KEYDIS4        inhibits all the other positions of the feedrate
override switch
CPS R103 LT 0 = SBS 0 R103 R103     ignores the handwheel turning direction
CPS R103 GT 120 = MOV 120 R103      Limits the maximum feedrate override to 120%.
DFU M2009                           With the leading edge (up flank) of the clock mark
M2009
= CNCWR(R103,PLCFRO,M1)             set the calculated feedrate override
(PLCFRO=R103)

L101
END
```

5.2 *MANUAL CONTROL OF THE SPINDLE*

It is possible to control the spindle by means of the following Operator-Panel keys without the need to execute M03, M04 or M05.

 is similar to executing M03. It starts the spindle clockwise and it displays M03 in the history of machining conditions.

 is similar to Executing M04. It starts the spindle counter-clockwise and it displays M04 in the history of machining conditions.

[STOP] is similar to executing M05. It stops the spindle.

[+] and [−] vary the programmed spindle speed between the % set in spindle machine parameters “MINSOVR” and “MAXSOVR with incremental steps set in spindle machine parameter “SOVRSTEP”.

It is recommended to define the spindle speed before selecting the turning direction in order to avoid an abrupt start.

6. TABLES

In order to select a new tool, tool offset or zero offset, it is necessary that those values be previously stored at the CNC.

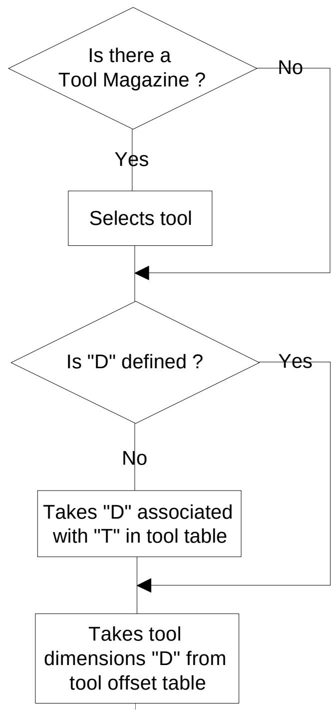
Zero Offset table. It must be defined. It indicates each axis offset for each zero offset.
Tool Offset table. It must be defined. It stores the dimensions of each tool.
Tool Table. It must be defined. It indicates for each tool, the family it belongs to, the offset associated to it, its nominal and real life spans, etc.
Tool Magazine Table. It must be defined. It indicates the position each tool occupies in the tool magazine.
Global and local parameter table. It does not need to be defined. It is updated by the CNC.

When selecting a tool (T) or a tool offset (D), the CNC acts as follows:

If the machine has a tool magazine, the CNC looks up the "tool magazine table" to know the position of the desired tool and of the selected tool.

If "D" has not been defined, it looks up the "Tool table" to know the tool offset "D" associated with it.

Examines the "Tool Offset Table" and it assumes the tool dimensions corresponding to the "D" offset.



It is recommended to save the tables in the "Memkey Card" (CARD A) or out to a peripheral device or PC.

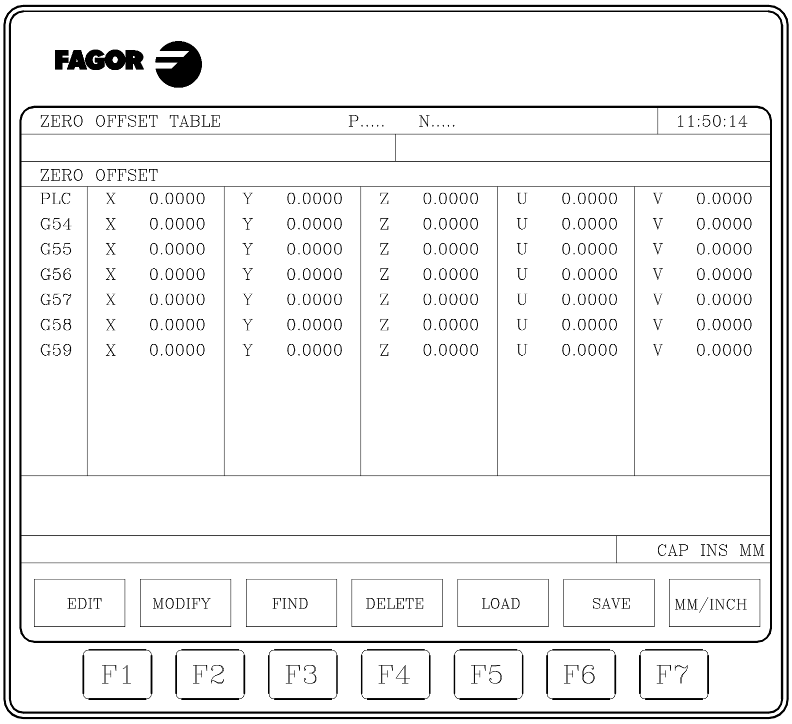
When accessing the TABLES operating mode, the CNC shows all the tables saved into the "Memkey Card" (CARD A).

If when turning the CNC on, a table is found to be damaged, it checks whether that table is stored in "CARD A" or not.
If it is stored in "CARD A", it asks whether to make a copy of it or not.
If it is not stored in "CARD A", it asks whether to reset to default values or not.

Note: When copying one of the following tables from "CARD A", the CNC is automatically reset.
Zero offsets, M functions, Tools, Tool magazines, Tool offsets, Geometry, Leadscrew error compensation, Cross compensation.

6.1 *ZERO OFFSET TABLE*

This table stores the offset of each axis.



The end of the chapter describes how to edit the table. The possible zero offsets are

PLC. Additive zero offset defined by PLC.

It is used, among others, to compensate for possible deviations due to machine dilatation.

These values are set from the PLC and from the part-program, by means of high level variables "PLCOF(X-C)".

The CNC always adds these values to the zero offset currently active.

G54 through G57. Absolute zero offsets.

They can also be modified from the PLC and from the part-program, by means of high level variables "ORG(X-C)".

In order for one of these absolute zero offsets to be active, it must be selected at the CNC using its corresponding G code (G54, G55, G56 or G57).

G58 and G59. Incremental zero offsets.

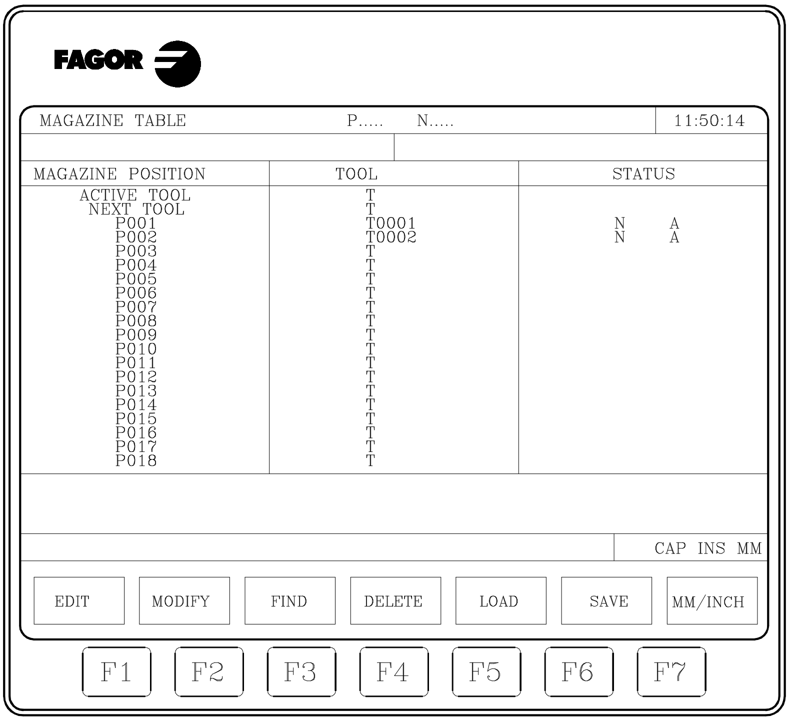
They can also be modified from the PLC and from the part-program, by means of high level variables "ORG(X-C)".

In order for one of these incremental zero offsets to be active, it must be selected at the CNC using its corresponding G code (G58 or G59).

The new incremental zero offset will be added to the absolute zero currently selected.

6.2 *TOOL MAGAZINE TABLE*

This table contains information about the tool magazine indicating all the tools of the magazine and their position in it.



The end of this chapter describes how to edit the table.

Magazine position

Besides indicating each position in the magazine, it indicates the active tool and the one selected for the following operations.
The next tool will be placed in the spindle after executing auxiliary function M06.

Tool

It indicates the number of the tool occupying that position.
The empty positions appear with the letter "T" and the canceled ones with the characters T****.

Status

The first letter indicates the tool size and the second one its status.

The size depends on the number of pockets it occupies in the magazine.

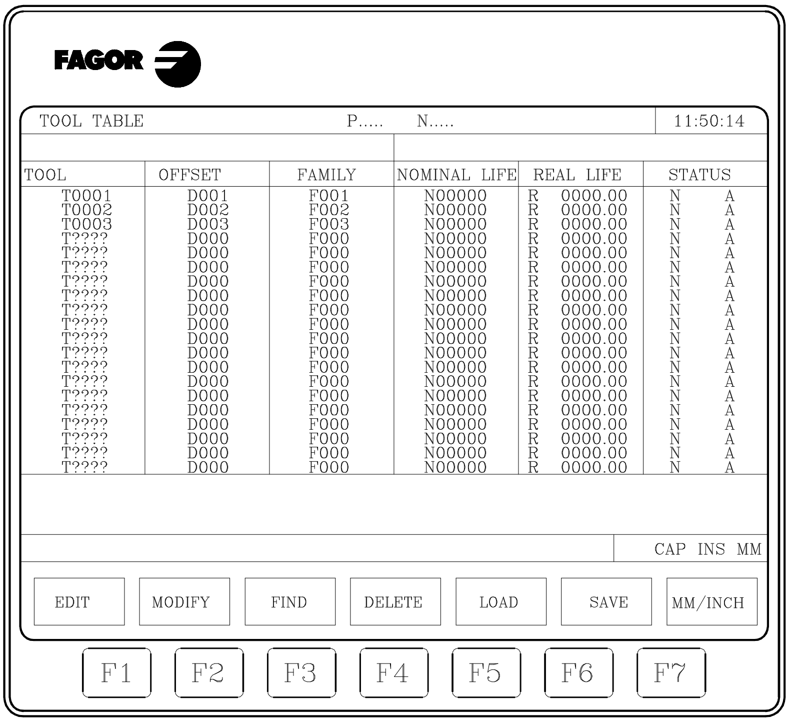
N = Normal (family 0-199)   S = Special (family 200-255)

The tool status is defined as following:

A = Available
E = Expired ("real life" greater than "nominal life")
R = Rejected by the PLC

6.3 TOOL TABLE

This table stores information about the tools available indicating the type of tool offset associated with them, their family, etc.



TOOL	OFFSET	FAMILY	NOMINAL LIFE	REAL LIFE	STATUS
T0001	D001	F001	N00000	R 0000.00	N A
T0002	D002	F002	N00000	R 0000.00	N A
T0003	D003	F003	N00000	R 0000.00	N A
T????	D000	F000	N00000	R 0000.00	N A
T????	D000	F000	N00000	R 0000.00	N A
T????	D000	F000	N00000	R 0000.00	N A
T????	D000	F000	N00000	R 0000.00	N A
T????	D000	F000	N00000	R 0000.00	N A
T????	D000	F000	N00000	R 0000.00	N A
T????	D000	F000	N00000	R 0000.00	N A
T????	D000	F000	N00000	R 0000.00	N A
T????	D000	F000	N00000	R 0000.00	N A
T????	D000	F000	N00000	R 0000.00	N A
T????	D000	F000	N00000	R 0000.00	N A
T????	D000	F000	N00000	R 0000.00	N A
T????	D000	F000	N00000	R 0000.00	N A
T????	D000	F000	N00000	R 0000.00	N A
T????	D000	F000	N00000	R 0000.00	N A
T????	D000	F000	N00000	R 0000.00	N A
T????	D000	F000	N00000	R 0000.00	N A

The end of this chapter describes how to edit the table. Each tool has the following data fields:

Offset number associated with the tool.

Every time a tool is selected, the CNC will assume its dimensions as they appear in the tool offset table for the specified tool offset.

Family code.

It will be used when having an automatic tool changer and it will allow replacing the worn out tool with another one with similar characteristics.

There are two types of families:

* Those for normal tools whose codes are between 0 and 199.

* Those for special tools (which occupy more than one magazine pocket), whose numbers are between 200 and 255.

Every time a new tool is selected, the CNC checks whether it is worn out ("real life" greater than "nominal life"). If that is the case, it will not select it, but it will select another one of the same family, instead.

If while machining a part, the PLC "asks" the CNC to reject the current tool (by activating the logic input “TREJECT”), the CNC will display the message "rejected" in the "STATUS" field and it will replace it with the next tool of the same family that appear in the tool table.

This change will take place the next time that tool is selected.

Nominal tool life.

It indicates the machining time (in minutes) or the number of operations that that tool is calculated to last.

Real tool life.

It indicates the machining time (in minutes) or the number of operations already carried out by that tool.

Tool status.

It indicates the size of the tool and its status:

The tool size depends on the number of pockets it takes in the magazine and it is defined as follows:

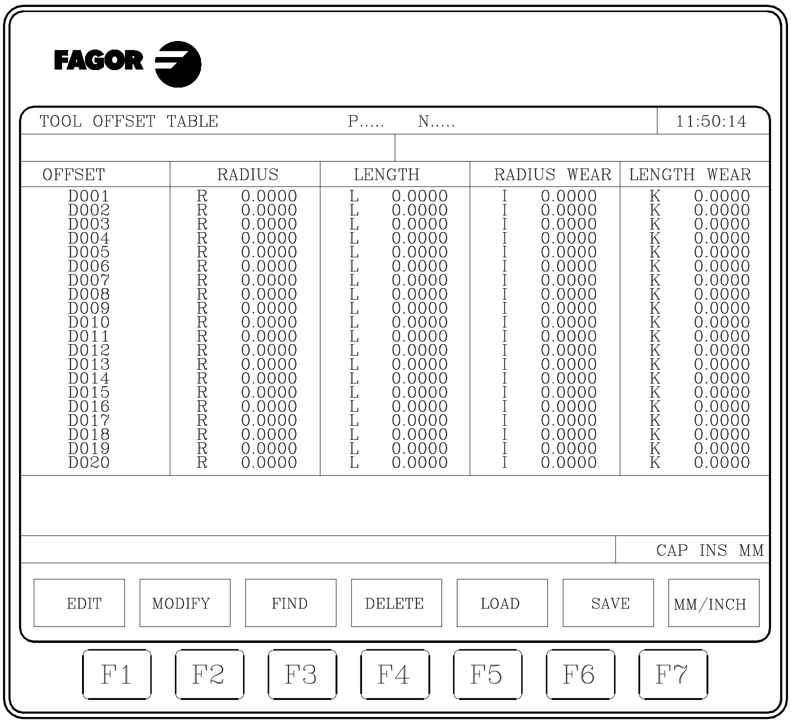
N = Normal (family 0-199)
S = Special (family 200-255)

The tool status is defined as follows:

A = Available
E = Expired ("real life" greater than "nominal life")
R = Rejected by the PLC

6.4 *TOOL OFFSET TABLE*

This table stores the dimensions of each tool.



OFFSET	RADIUS	LENGTH	RADIUS WEAR	LENGTH WEAR
D001	R 0.0000	L 0.0000	I 0.0000	K 0.0000
D002	R 0.0000	L 0.0000	I 0.0000	K 0.0000
D003	R 0.0000	L 0.0000	I 0.0000	K 0.0000
D004	R 0.0000	L 0.0000	I 0.0000	K 0.0000
D005	R 0.0000	L 0.0000	I 0.0000	K 0.0000
D006	R 0.0000	L 0.0000	I 0.0000	K 0.0000
D007	R 0.0000	L 0.0000	I 0.0000	K 0.0000
D008	R 0.0000	L 0.0000	I 0.0000	K 0.0000
D009	R 0.0000	L 0.0000	I 0.0000	K 0.0000
D010	R 0.0000	L 0.0000	I 0.0000	K 0.0000
D011	R 0.0000	L 0.0000	I 0.0000	K 0.0000
D012	R 0.0000	L 0.0000	I 0.0000	K 0.0000
D013	R 0.0000	L 0.0000	I 0.0000	K 0.0000
D014	R 0.0000	L 0.0000	I 0.0000	K 0.0000
D015	R 0.0000	L 0.0000	I 0.0000	K 0.0000
D016	R 0.0000	L 0.0000	I 0.0000	K 0.0000
D017	R 0.0000	L 0.0000	I 0.0000	K 0.0000
D018	R 0.0000	L 0.0000	I 0.0000	K 0.0000
D019	R 0.0000	L 0.0000	I 0.0000	K 0.0000
D020	R 0.0000	L 0.0000	I 0.0000	K 0.0000

The end of this chapter describes how to edit the table. Each offset has a number of fields containing the tool dimensions. These fields are:

Tool radius.

Tool length.

Tool radius wear

The CNC will add this value to the nominal radius to calculate the real tool radius (R+I).

Tool length wear.

The CNC will add this value to the nominal length to calculate the real tool length (Z+K).

How to edit these values will be described later on. They can also be modified from the PLC and from the part-program by means of the high level variables associated with the tools.

6.5 GLOBAL AND LOCAL PARAMETER TABLES

The CNC has two types of general purpose variables:

Local parameters P0-P25 (7 levels)
Global parameters P100-P299.

The CNC updates the parameter tables after carrying out the operations indicated in the block in preparation. This operation is always carried out before executing the block. Therefore, the values shown in the table may not coincide with those of the block being executed.

When quitting the Execution mode after interrupting program execution, the CNC updates the parameter tables with the values corresponding to the block that was being executed.

In the global and local parameter tables, the values of the parameters may be displayed in decimal (4127.423) or in scientific notation (0.23476 E-3).

The CNC generates a new nesting level of local parameters every time parameters are assigned to a subroutine. Up to a maximum of 6 nesting levels of local parameters are possible.

Machining canned cycles G66, G68, G69, G81, G82, G83, G84, G85, G86, G87, G88 and G89 use the sixth nesting level of local parameters when they are active.

To access the different local parameter tables, the corresponding level must be indicated (0 through 6).

While programming in high level, local parameters may be referred to as P0-P25, or as **A-Z**, "A" being the same as "P0" and "Z" the same as "P25".

To do that, the local parameter tables show the letter associated to them, in brackets, next to the parameter number. In the tables, the parameter can only referred to as P0-P25, letters are not allowed.

6.6 *HOW TO EDIT TABLES*

The screen may be scrolled up and down line by line using the up/down arrow keys or page by page using the page up/down keys.

There are several ways to edit or modify a line which will be described next.

Once the user has selected any of those options, a editing area is available on the screen which may be scrolled up and down using the up/down arrow keys. On the other hand, with the up arrow key, the cursor may be placed over the first character of the editing window and, with the down arrow key over the last character.

EDIT

Once this option has been selected, the softkeys change color appearing over a white background and they show the information corresponding to the type of editing that may be done.

On the other hand, more information on the editing commands can be obtained at any time by pressing [HELP]. To quit this help mode, press [HELP] again.

Press [ESC] to quit the editing mode and maintain the table with the previous values.

Once the editing is done, press [ENTER]. The values assigned will be entered into the table.

MODIFY

Once this option has been selected, the softkeys change color appearing over a white background and showing the information corresponding to each field.

On the other hand, more information on the editing commands can be obtained at any time by pressing [HELP]. To quit this help mode, press [HELP] again.

By pressing [ESC], the information shown in the editing area is deleted. From this point on, the selected line may be edited again.

To quit the "modify" option, the information appearing in the editing area must be deleted by pressing [CL] or [ESC] and then [ESC]. The table will keep its previous values.

Once the modification is over, press [ENTER]. The new values assigned will be entered into the table.

FIND

Once this option has been selected, the softkeys will show the following options:

BEGINNING When pressing this softkey, the cursor is placed over the first line of the table that can be edited.

END When pressing this softkey, the cursor is placed over the last line of the table.

ZERO OFFSET, TOOL OFFSET, TOOL, POSITION, PARAMETER
When pressing one of these softkeys, the CNC requests the field number to be found. Once the field has been defined, press [ENTER].

The CNC searches for the requested field and places the cursor over it (when found).

DELETE

When deleting a line, the CNC sets all the fields to "0".

To delete a line, indicate its number and press [ENTER]

To delete several lines, indicate the beginning, press the [UP TO] softkey, indicate the last line to be deleted and press [ENTER]

To delete all the lines, press the "ALL" softkey. The CNC requests confirmation of the command.

INITIALIZE

It deletes all the data of the table by setting them all to "0". The CNC requests confirmation of the command.

LOAD

The tables may be loaded from the "Memkey Card" (CARD A) or a peripheral device or a PC through the two serial communications lines (RS232C or RS422).

The transmission starts after pressing the corresponding softkey. When using a serial line, the receptor must be ready before starting the transmission.

To interrupt the transmission, press the "ABORT" softkey.

If the length of the table received does not coincide with the current table length, the CNC will act as follows:

If the table received is shorter than the current one, the received lines are modified and the rest remain with their previous values.

If the table received is longer than the current one, all the lines of the table are modified and when detecting that there is no more room, the CNC will issue the corresponding error message

SAVE

The tables may be saved into the "Memkey Card" (CARD A) or out to a peripheral device or PC through the two serial lines (RS232C or RS422).

The transmission starts after pressing the corresponding softkey. When using a serial line, the receptor must be ready before starting the transmission.

To interrupt the transmission, press the "ABORT" softkey.

MM/INCHES

It toggles the display units for the data. The lower right-hand side window shows the units selected (MM/INCH).

7. UTILITIES

In this operating mode, one can access the programs stored in the CNC's RAM memory, in the "Memkey Card" (CARD A), in the hard disk (HD) and in external devices (through the serial lines 1 and 2).

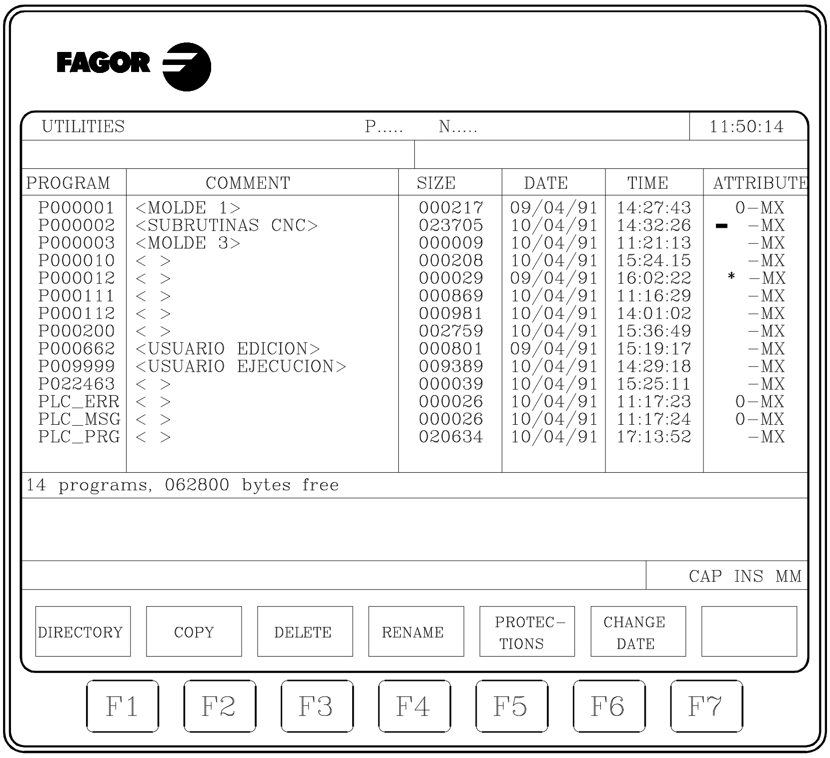
They can be deleted, renamed or their protection changed. It is also possible to make copies within the same device or from one to another.

7.1 DIRECTORY

To access the program directory of the CNC's RAM memory, the "Memkey Card" (CARD A), the hard disk (HD) and of the external devices (through serial lines 1 and 2).

The subroutine directory of the CNC can also be accessed.

Program directory.
By default, the CNC shows the program directory of the RAM memory, to view another directory, press the corresponding softkey.



On each directory, the CNC shows all the programs visible (not hidden) to the user, that is:

- Part programs
- Customizing programs
- The PLC program (PLC_PRG)
- The PLC error file (PLC_ERR)
- The PLC message file (PLC_MSG)

The program directory has the following definition fields:

Program
It shows the number when it is a part-program or a customizing program and the corresponding mnemonic when it is a PLC program, the PLC error file or the PLC message file.

Comment
Any program may have a comment associated with it for its identification.
The comments may be defined when editing the program or in this operating mode using the Rename option as described later on.

Size
It indicates, in bytes, the size of the program text. It must be borne in mind that the actual size of the program is slightly greater because this field does not include the space occupied by some variables used internally (header, etc.).

The date and the time when the program was edited (last changed)

Attributes
They show information about the source and usefulness of each program. The attributes are defined in this operating mode by means of the Protections option as described later on.

***** The program is running, either because it is the main program or because it contains a subroutine which has been called upon from that program or from another subroutine.

O The program was created by the machine manufacturer.

H The program is hidden and cannot be displayed in any directory.

Since a hidden program can nevertheless be edited or deleted if its number is known, it is recommended to remove the "Modifiable" attribute to prevent it from being edited or deleted.

M The program may be modified. In other words, it may be edited, copied, etc.

If a program does not have this attribute, the operator cannot see or modify its contents.

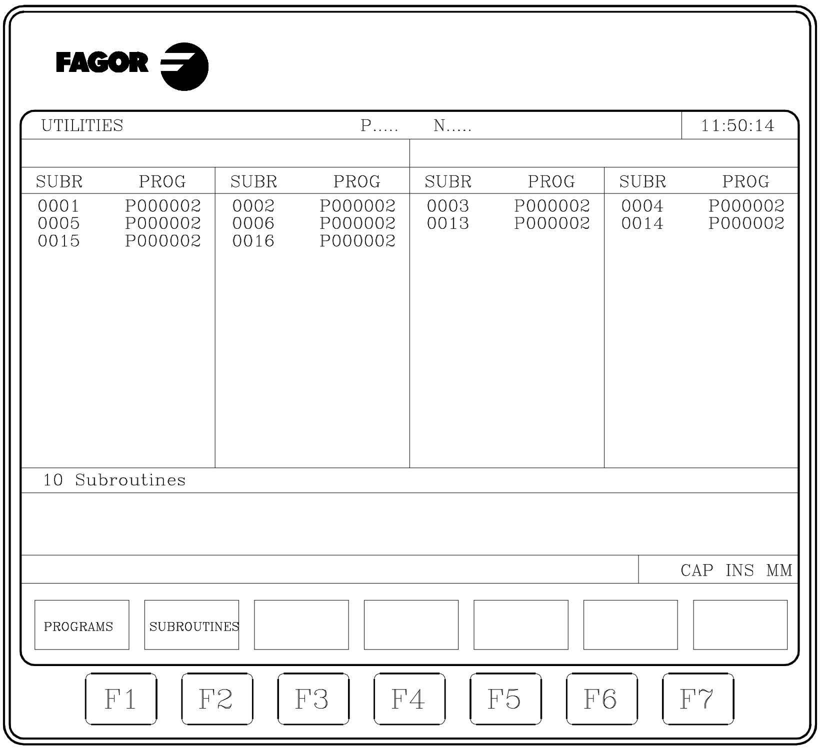
X Indicates that the program may be executed.

A program not having this attribute cannot be executed by the operator.

Only the attributes currently selected will be shown, the ones not selected will appear as "-".

Example: O—X Indicates that the program was created by the manufacturer, it will be displayed in the directory (not hidden), it cannot be modified, but it may be executed.

Subroutines directory.



It lists all the subroutines defined in the part programs of the CNC ordered from the smallest one to the largest one.

Also, next to the subroutines, it displays the number of the program where it has been defined.

If the program containing the subroutine has the "hidden" attribute assigned to it, that program number will appear as P??????.

7.1.1 DIRECTORY OF THE EXTERNAL DEVICES

When accessing the directory of an external device through the serial lines, that directory is shown in DOS format.

The [CHANGE DIR] softkey lets the user select the work directory of the PC to operate with from the CNC.

This operation does not change the work directory that was selected to operate with from the PC.

In other words, when working via DNC, it is possible to select a work directory at the PC and another PC directory at the CNC.

This new feature is available from DNC50 version 5.1 on.

7.2 COPY

To copy programs in the same directory or between directories of different devices.

To make the copy, proceed as follows:

- Press the COPY softkey.
- Indicate where the program or programs to be copied are located.
  CNC's RAM memory, "Memkey Card" (CARD A), Hard disk (HD) o external devices (serial lines).
- Indicate the number of the program to be copied.
  Select the program with the arrows and press [ENTER] or key in its number.
- To copy several programs, press the softkeys: "To the end" or "To". For "To" indicate the number of the last program to be copied.
- Press the IN softkey.
- Indicate the destination of the copy.
  CNC's RAM memory, "Memkey Card" (CARD A), Hard disk (HD) or external devices (serial lines).
- When copying a single program, a different number may be selected for the target program.
- Press [ENTER].

Example Copy program 200103, from the CNC's RAM memory out to the "Memkey Card" with the number 14

COPY (MEMORY) P200103 IN (CARD A) P14 ENTER

Copy from program 102100 to the end, from CNC's RAM memory into the "Memkey Card".

COPY (MEMORY) P102100 (TO THE END) IN (CARD A) ENTER

If a program with the same number already exists, the CNC will display a warning message. On the other hand, if that program is in execution, the CNC will display a message indicating that it is not possible.

Two subroutines may not have the same name in RAM memory. To make a copy and change the name of the copied subroutine, write the subroutine defining block as a comment **before** making the copy.

7.3 DELETE

To delete programs in the same directory or between directories of different devices.

To delete a program, proceed as follows:

- Press the DELETE softkey.
- Indicate where the program or programs to be deleted are located.
  CNC's RAM memory, "Memkey Card" (CARD A), Hard disk (HD) o external devices (serial lines).
- Indicate the number of the program to be deleted.
  Select the program with the arrows and press [ENTER] or key in its number.
- To delete several programs, press the softkeys: "To the end" or "To". For "To" indicate the number of the last program to be deleted.
- Press [ENTER].

Example: To delete program 200103 from the "Memkey Card"

DELETE (CARD A) P200103 ENTER

To deleted from programa 123123 to program123456, from the CNC's RAM memory:

DELETE (MEMORY) P123123 (TO) P123456 ENTER

Only programs that can be modified ("M" attribute) can be deleted.

7.4 RENAME

To rename or assign a new comment to a program stored in the CNC's RAM memory, "Memkey Card" (CARD A), or in the Hard Disk (HD).

To rename a program, proceed as follows:

- Press the [RENAME] softkey.

- Indicate the number of the program to be renamed.
  If it is in another directory, press the corresponding softkey.
  Select the program with the arrow keys and press [ENTER] or key in its number and press the [TO] softkey.

The files associated with the PLC (program, messages and errors) are always referred to with their associated mnemonics. Therefore, only their comment may be renamed.

If there is a program with the same number, the CNC will issue a warning message and it will offer the chance to modify the command.

Examples:

To change the name of program 200103 from the "Memkey Card"

RENAME (CARD A) P200103 TO NEW NUMBER P12 ENTER

to change the comment of program 100453 from the CNC

RENAME (MEMORY) P100453 TO NEW COMMENT "Test" ENTER

7.5 PROTECTIONS

To prevent certain programs from being manipulated and restrict access to the operator to certain CNC commands.

It is possible to protect programs stored in the CNC's RAM memory, in the "Memkey card" (CARD A) or in the Hard Disk (HD).

USER PERMISSIONS

Lets the operator see those CNC programs that have been created by the operator and sets their attributes.

To modify the attributes of a program, proceed as follows:

- Press the [USER PERMISSION] softkey
- Indicate the program number
  If it is in another directory, press the corresponding softkey.
  Select the program with the arrow keys, or key in its number and press [ENTER].
- Press the following softkeys

F2	to change the (H) attribute	hidden program
F3	to change the (M) attribute	modifiable program
F4	to change the (X) attribute	executable program

- Press [ENTER].

OEM PERMISSION

Lets see all the programs stored at the CNC whether they are created by the OEM or by the operator and set their attributes.

To modify the attributes of a program, proceed as follows:

- Press the [OEM PERMISSION] softkey
- Indicate the program number
  If it is in another directory, press the corresponding softkey.
  Select the program with the arrow keys, or key in its number and press [ENTER].
- Press the following softkeys:

F1	to change the (O) attribute	OEM program
F2	to change the (H) attribute	hidden program
F3	to change the (M) attribute	modifiable program
F4	to change the (X) attribute	executable program

- Press [ENTER].

PASSWORDS

Lets define each of the passwords that the operator must key in before accessing the various CNC commands.

General access password (MASTERPSW)
It is requested when trying to access this password option.
(Utilities mode / Protections / Passwords).

OEM password (OEMPSW)
It is requested when trying to access OEM permissions
(Utilities mode / Protections / OEM permissions).

User password (USERPSW)
It is requested when trying to access user permissions
(Utilities mode / Protections / User permissions).

PLC access password (PLCPSW)
It is requested in the following cases:

- When compiling the PLC program.
- When trying to change the status of a resource or execute a program execution controlling command.

To protect the PLC program, the PLC message program and the PLC error program, change their attributes so they are "not modifiable".

Customizing password (CUSTOMPSW)
It is requested when trying to access the Customizing mode or when attempting to erase an OEM screen.

Machine parameter access password (SETUPPSW)
It is requested when trying to access the options to modify the table values (Edit, Modify, Initialize, Delete and Load) except for tables of the serial lines which are not protected.

To change or delete the passwords, use the following softkeys:

Change password.
Select the desired password and enter the new one.

Delete password.
Lets delete (eliminate) one of several codes from the table.

- To delete a password, indicate its number and press [ENTER].
- To delete several passwords (they must be in a row), indicate the number of the first one to be delete, press the "UPTO" softkey, indicate the number of the last one to be deleted and press [ENTER].
- To delete a password, indicate its number and press [ENTER].

Clear all.
Lets delete all the passwords. The CNC will request confirmation of the command and it will delete them after pressing [ENTER].

7.6 CHANGE DATE

Lets change the system date and time.

First, the date will be shown as day/month/year (12/04/1998). After changing it, press [ENTER] to validate it. If it is not to be changed, press [ESC].

Next, the time will be shown as hours/minutes/seconds (08/30/00). After changing it, press [ENTER] to validate it. If it is not to be changed, press [ESC].

8. *DNC*

Each softkey of this operating mode shows the following information:

CNC	Number of the program and line that was executing the last time an execution error came up or there was a power outage.
DNC	Information and statistics of the DNC communication lines.
SERCOS	Information and statistics of the CAN communication.
CAN	Information and statistics of the SERCOS communication.

8.1 CNC

This screen shows the number of the line that was being executed last time an execution error or a power outage occurred.

The CNC shows the program number and line number that was executing as well as where the program is stored.

On the other hand, if that program called upon a subroutine and the CNC was executing it, it will display:

The subroutine number, the program containing its definition and the line or block of the subroutine being executed.

Example:

Device	Program	Line number	Subroutine
CARD A	000012	7	
MEMORY	001000	15	0033

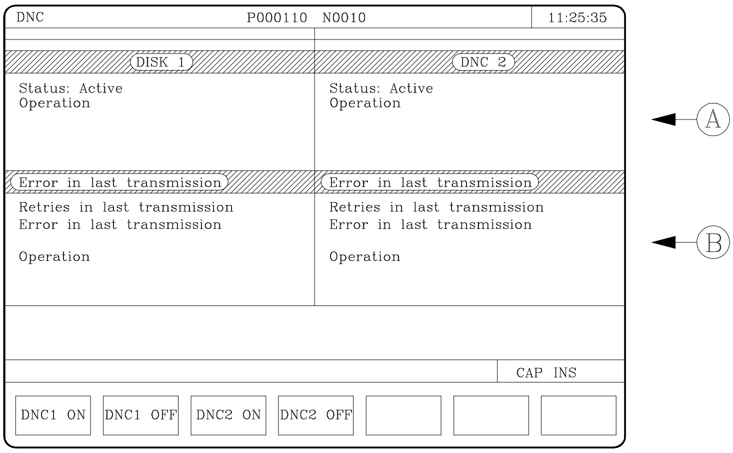
Indicates that the CNC was executing line 7 of program 12 of CARD A.

That program line called to subroutine 15 and it was executing its line number 33. That subroutine is defined (contained) in program 1000 which is stored in the CNC's RAM memory.

8.2 DNC

With this CNC, it is possible to access this operating mode when at least one of the serial lines (RS232C or RS422) is set to work in the DNC mode or to communicate with the FAGOR Floppy Disk Unit.

When accessing this mode, the CNC shows the following screen:



The left side of the screen corresponds to serial line 1 and the right side to serial line 2.

In the example of the figure above, serial line 1 is used to communicate with a Fagor Floppy Disk Unit; and serial line 2 to communicate via DNC.

The upper area, A, indicates:

* The status of the serial line: Active / Inactive.

* The type of operation in progress:
  Sending program / Receiving program / Sending directory /
  Receiving directory / etc.

The lower area, B, indicates the last operation and the type of error occurred if any.

Also, the following softkeys appear at the bottom of the screen for each serial line currently activated by machine parameter:

DNC ON Activates the DNC mode in the corresponding serial channel.

DNC OFF Deactivates the DNC mode which is active in the corresponding serial channel.

The activation/deactivation of this operating mode is made dynamically, therefore, if, when deactivating the DNC mode, you are transmitting via this channel, the CNC aborts the transmission and deactivates the DNC.

Irrespective of this operating mode, the OEM can set by machine parameter whether the DNC mode will be active or not on power-up.

8.3 SERCOS

The CNC allows access to this operating mode when the SERCOS bus has been defined.

It is especially designed for the Technical Service Department.

It displays information and statistics on the incidences occurred in SERCOS communication.

The data with a green background must have a "0" value if everything is fine.

Pressing the "LOG FILE" softkey displays a history of the errors occurred up to that instant, their error number and their description.

Use the "SAVE" softkey to save the error history into a program.

8.4 CAN

The CNC allows accessing this operating mode when the CAN bus has been defined.
It is especially designed for the Technical Service Department.
It shows information and statistics of the incidences occurred in CAN communication.

The main screen shows:

Information of each node

Node ID	Node identifier
Reported inputs	Number of inputs detected
Reported outputs	Number of outputs detected
Rx errors	Reception errors
Tx errors	Transmission errors
Lost messages	
Stage	Communication status stage. 5 = OK

CNC information

Stage	Communication status stage
CAN speed	
Rx errors	Reception errors
Tx errors	Transmission errors
Lost messages	
Number of retries	
Status	For Fagor Service Department
Input cycle	For Fagor Service Department

The data on a green background must have a value of "0" in order to be OK.

Pressing the LOG FILE softkey displays a history of the errors that came up with their date, time, error number and description.

Use the "SAVE" softkey to save the error history into a program.

Press the VERSION softkey to display a screen that shows:
The software version installed in each node: SW version, Date and Checksum.
The hardware version for the node: HV version
The CAN software version available at the CNC: SW version and Checksum.

There are softkeys to update or copy the CNC's CAN software in each Fagor module. This operation may be carried out either module by module or in all of them at the same time.

9. *PLC*

In this mode of operation it is possible to access the PLC to check its operation or the status of the various PLC variables. It also allows editing and analyzing the PLC program as well as the PLC message file and error file.

The accessible programs associated with the PLC are:

The PLC program (PLC_PRG)
The PLC error file (PLC_ERR)
The PLC message file (PLC_MSG)

The PLC program (PLC_PRG) may be edited at the front panel or copied from the "Memkey Card" (CARD A) or from a peripheral device or PC.

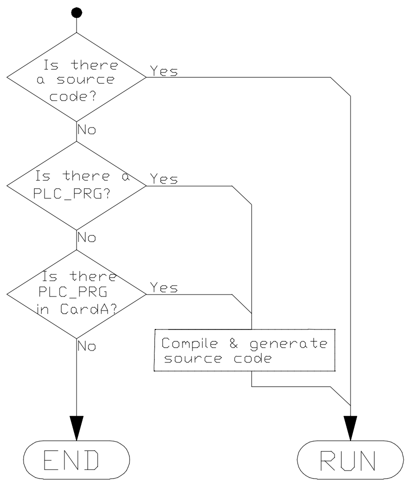
The PLC program (PLC_PRG) is stored in the internal CNC memory with the part-programs and it is displayed in the program directory (utilities) together with the part-programs.

Before executing the PLC_PRG program, it must be compiled.
Once it is done compiling, the CNC requests whether the PLC should be started or not.

To make the operator life easier and avoid new compilations, the source code generated at each compilation is stored in memory.

After power-up, the CNC acts as follows:

- Runs the executable program stored in memory.
- If there isn't one, it compiles the PLC_PRG program already in memory and runs the resulting executable program.
- If there isn't one, it looks for it in the "Memkey Card" (CARD A)
- If it isn't in the CARD A either, it does nothing. Later on, when accessing the Jog mode, Execution mode, etc. the CNC will issue the corresponding error message.



Once the program has been compiled, it is not necessary to keep the source program (PLC_PRG) in memory because the PLC always executes the executable program.

Once the proper performance of the PLC has been verified, it is a good idea to save it into the "Memkey Card" (CARD A) using the instruction SAVE PROGRAM (as described later on).

9.1 *EDIT*

Once this option is selected, indicate with the corresponding softkey the PLC program to be edited.

The PLC program (PLC_PRG)
The PLC error file (PLC_ERR)
The PLC message file (PLC_MSG)

The cursor can be moved line by line with the "up and down" arrow keys or page by page with the "page up" and "page down" keys.

The cursor position or line number will be displayed in a white window inside the communications window (bottom of the screen) next to the CAP/INS indicator window.

This operating mode offers various options which are described next.

Once any of these functions is selected, the CNC shows an editing area on the CRT where the cursor may be moved by using the up/down and right/left arrow keys. Also, the up-arrow key positions the cursor over the first character of the editing area and the down-arrow key positions the cursor over the last character.

EDIT

With this option it is possible to edit new lines or blocks of the selected program.

Before pressing this softkey, the block after which the new ones will be added must be selected with the cursor.

The program will be edited (written) a block at a time and each block can be written in ISO language, High Level language or it can be just a program comment.

Once this option is selected, the softkeys will change their color showing their type of editing option over a white background.

Also, it is possible to get more editing assistance by pressing **HELP**. Press **HELP** again to exit the editing assistance mode.

Press the **ESC** key to exit the block editing mode when writing a block and this block will not be added to the program.

Once the block has been edited, press **ENTER** to add it to the program behind the block previously indicated by the cursor.

The cursor will be positioned at the new block (just edited) and the editing window (area) will be cleared In order to edit a new block.

Press **ESC** or **MAIN MENU** to quit the block editing mode.

MODIFY

This option permits modifying the contents of a selected program block.

Before pressing this softkey, select with the cursor the block to be modified .

Once this option is selected, the softkeys will change their color showing their type of modifying option over a white background.

Also, it is possible to get more editing assistance by pressing **HELP**. Press **HELP** again to exit the editing assistance mode.

By pressing **ESC**, the information corresponding to that block and which was shown in the editing area will be cleared. It will then be possible to modify its contents again.

To quit the block modifying mode, press **CL** or **ESC** to clear the editing window and then press **ESC** again. This way, the selected block will not be modified.

Once the block contents have been modified, press **ENTER** so the new contents replace the old ones.

FIND

This option is used to find a specific text within the selected program.

When selecting this option, the following options will appear:

BEGINNING This softkey positions the cursor over the first program block which is then selected quitting the "find" option.

END This softkey positions the cursor over the last program block which is then selected quitting the "find" option.

TEXT With this function it is possible to search a text or character sequence starting from the block indicated by the cursor.

When this key is selected, the CNC requests the character sequence to be found.

When the text is defined, press the "END OF TEXT" softkey and the cursor will be positioned over the first occurrence of that text.

The search will begin at the current block.

The text found will be highlighted being possible to continue with the search or to quit it.

Press **ENTER** to continue the search up to the end of the program. It is possible to search as many times as wished and when the end of the program is reached, it will start from the first block.

Press the "EXIT" softkey or the **ESC** key to quit the search mode. The cursor will be positioned where the indicated text was found last.

LINE NUMBER After pressing this key, the CNC requests the number of the block to be found. After keying in the desired number and pressing **ENTER**, the cursor will position over that block which will then be selected quitting the search mode.

REPLACE

With this function it is possible to replace a character sequence with another throughout the selected program.

When selecting this option, the CNC requests the character sequence to be replaced.

Once the text to be replaced is indicated, press the "WITH" softkey and the CNC will request the character sequence which will replace the previous one.

Once this text is keyed in, press the "END OF TEXT" softkey and the cursor will be positioned over the first occurrence of the searched text.

The search will begin at the current block.

The found text will be highlighted and the following softkey options will appear:

REPLACE Will replace the highlighted text and will continue the search from this point to the end of the program.

If no more occurrences of the text to be replaced are found, the CNC will quit this mode.

If another occurrence of the text is found, it will be highlighted showing the same "replacing" or "not replacing" options.

DO NOT REPLACE Will not replace the highlighted text and will continue the search from this point to the end of the program.

If no more occurrences of the text to be replaced are found, the CNC will quit this mode.

If another occurrence of the text is found, it will be highlighted showing the same "replacing" or "not replacing" options.

TO THE END This function will automatically replace all the matching text from the current block to the end of the program without offering the option of not replacing it.

ABORT This function will not replace the highlighted text and it will quit the "find and replace" mode.

DELETE BLOCK

With this function it is possible to delete a block or group of blocks.

To delete only one block, just position the cursor over it and press **ENTER**.

To delete a group of blocks, indicate the first and last blocks to be deleted. To do so, follow these steps:

* Position the cursor over the first block to be deleted and press the "INITIAL BLOCK" softkey.

* Position the cursor over the last block to be deleted and press the "FINAL BLOCK" softkey.

  If the last block to be deleted is also the last one of the program, it can also be selected by pressing the "TO THE END" softkey.

* Once the first and last blocks are selected, the CNC will highlight the selected blocks requesting confirmation to delete them.

MOVE BLOCK

With this option it is possible to move a block or group of blocks by previously indicating the first and last blocks to be moved. To do so, follow these steps:

* Position the cursor over the first block to be moved and press the "INITIAL BLOCK" softkey.

* Position the cursor over the last block to be moved and press the "FINAL BLOCK" softkey.

  If the last block to be moved is also the last one of the program, it can also be selected by pressing the "TO THE END" softkey.

  To move only one block, the "initial block" and the "final block" will be the same one.

* Once the first and last blocks are selected, the CNC will highlight the selected blocks requesting confirmation to move them.

  Then, indicate the block after which this group of blocks must be placed.

* Press the "START OPERATION" softkey to carry out the move.

COPY BLOCK

With this option it is possible to copy a block or group of blocks by previously indicating the first and last blocks to be copied. To do so, follow these steps:

* Position the cursor over the first block to be copied and press the “INITIAL BLOCK” softkey.

* Position the cursor over the last block to be copied and press the “FINAL BLOCK” softkey.

  If the last block to be copied is also the last one of the program, it can also be selected by pressing the “TO THE END” softkey.

  To copy only one block, the “initial block” and the “final block” will be the same one.

* Once the first and last blocks are selected, the CNC will highlight the selected blocks requesting confirmation to copy them.

  Then, indicate the block after which this group of blocks must be placed.

* Press the “START OPERATION” softkey to carry out this command.

COPY TO PROGRAM

With this option it is possible to copy a block or group of blocks of one program into another program.

When selecting this option, the CNC will request the number of the destination program where the selected block or blocks are to be copied. After entering the program number press **ENTER**.

Next, indicate the first and last blocks to copy by following these steps:

* Position the cursor over the first block to be copied and press the "INITIAL BLOCK" softkey.

* Position the cursor over the last block to be copied and press the "FINAL BLOCK" softkey.

  If the last block to be copied is also the last one of the program, it can also be selected by pressing the "TO THE END" softkey.

  To copy only one block, the "initial block" and the "final block" will be the same one.

* Once the first and last blocks are selected, the CNC will highlight the selected blocks and will execute the command.

  If the destination program already exists, the following options will be displayed:

* Write over the existing program. All the blocks of the destination program will be erased and will be replaced by the copied blocks.

* Append (add) the copied blocks behind the ones existing at the destination program.

* Abort or cancel the command without copying the blocks.

INCLUDE PROGRAM

With this option it is possible to include or merge the contents of another program into the one currently selected.

Once this option is selected, the CNC will request the number of the source program to be merged. After keying in that number press **ENTER**.

Next, indicate with the cursor the block after which the source program will be included.

Finally, press the "START OPERATION" softkey to execute the command.

9.2 COMPILE

With this option it is possible to compile the PLC source program "PLC_PRG".

The PLC program must be stopped in order to compile it, otherwise, the CNC will "ask" if it is desired to stop it.

Once the source program compiled, the CNC will generate the executable PLC program (object program).

If while compiling, some errors are detected, the CNC will not create the object program and the detected errors (up to 15) will appear on the screen.

If the errors do not affect the proper program execution (such as non-referenced labels, etc.), the CNC will display the corresponding warning messages but it will generate the object program.

After a successful compilation, the CNC will "ask" whether the PLC program must be started or not.

9.3 MONITORING

With this option it is possible to display the PLC program and analyze the status of the different PLC resources and variables.

Once this option has been selected, the CNC will show the source program that corresponds to the executable program (object) even when that program (source) has been deleted or modified at the CNC.

The CNC will also display all the variable consultations at logic level 1 (including those not being executed) and the actions whose conditions are met.

To display the program from a specific line on, press the "**L**" key followed by that line number and then press **ENTER**.

The operator can move the cursor around the CRT a line at a time with the up/down arrow keys and a page at a time with the page-up and page-down keys.

The various monitoring options available are described next.

Once any of the those options has been selected, the operator has an editing window where the cursor may be moved with the right and left arrow keys. The up arrow will position the cursor over the first character of the editing window and the down arrow over the last one.

MODIFY THE STATUS OF THE RESOURCES

The CNC has the following instructions to modify the status of the different PLC resources.

I 1/256 = 0/1	Alters the status (0/1) of the indicated input. For example: I120 = 0, sets input I120 to 0.
I 1/256.1/256 = 0/1	Alters the status (0/1) of a the indicated group of inputs. For example: I100.103 = 1, sets inputs I101, I102 and I103 to 1.
O 1/256 = 0/1	Alters the status (0/1) of the indicated output. For example: O20 = 0, sets output O20 to 0.
O 1/256.1/256 = 0/1	Alters the status (0/1) of the indicated group of outputs. For example: O22.25= 1 sets outputs O22 thru O25 to 1.
M 1/5957 = 0/1	Alters the status (0/1) of the indicated mark. For example: M330 = 0 sets Mark M330 to 0.
M 1/5957.1/5957 = 0/1	Alters the status (0/1) of the indicated group of marks. For example: M400.403=1 sets marks M400 thru M403 to 1.
TEN 1/256 = 0/1	Alters the status (0/1) of the ENABLE input of the indicated timer. For example: TEN12 = 1, sets the Enable input of timer T12 to 1.

TRS 1/256 = 0/1	Alters the status (0/1) of the RESET input of the indicated timer. For example: TRS2 = 0 sets the reset input of timer T2 to 0.
TGn 1/256 n = 0/1	Alters the status (0/1) of the trigger input "TGn" of the indicated timer (1 thru 256) assigning the desired time constant (n) to it. For example: TG1 22 1000 sets the trigger input 1 of timer T22 to one and it assigns a time constant of 1000 (10 seconds).
CUP 1/256 = 0/1	Alters the status (0/1) of the UP count input of the indicated counter. For example: CUP 33 = 0 sets the status of the UP input of counter C33 to 0.
CDW 1/256 = 0/1	Alters the status (0/1) of the DOWN count input of the indicated counter. For example: CDW 32 = 1 sets the status of the UP input of counter C32 to 1.
CEN 1/256 = 0/1	Alters the status (0/1) of the enable input of the indicated counter. For example: CEN 12 = 0, sets the enable input of counter 12 to 0.
CPR 1/256 n = 0/1	Alters the status (0/1) of the preset input of the indicated counter (1 thru 256). The counter will be preset with the value "n" if an up flank is produced with this instruction. For example: CPR 10 1000 =1 sets the preset input of counter C10 to 1 and also, if an up flank has occurred (being previously set to 0), the counter will be preset with a value of 1000.
C 1/256 = n	Presets the count of the indicated counter to the "n" value. For example: C42 = 1200 sets the count of counter C42 to 1200.
B 0/31 R 1/559 = 0/1	Alters the status (0/1) of the indicated bit (0/31) of the indicated register (1/559). For example: B5 R200 = 0 sets Bit 5 of register R200 to 0.
R 1/559 = n	Assigns the "n" value to the indicated register. For example: R 303 = 1200 assigns the value of 1200 to register R303.
R 1/559.1/559 = n	Assigns the "n" value to the indicated register group. For example: R234.236 = 120 assigns the value of 120 to registers R234, R235 and R236.

It must be borne in mind that when referring to a single resource, it is possible to do it using its corresponding mnemonic.

For example: /STOP=1 is interpreted by the CNC as M5001=1

CREATE WINDOW

This CNC allows the possibility of creating windows to display the status of the various PLC resources.

These windows will be shown overlapping the PLC program and the information displayed in them will be updated dynamically.

The options "MODIFY WINDOW", "ACTIVE WINDOW" and "ACTIVATE SYMBOLS" allow the manipulation of these windows.

Every time a new window is created, the CNC will assign 2 data lines to it in order to display the status of the desired resources.

There are two types of windows which can be selected with softkeys.

WINDOW TO DISPLAY TIMERS AND REGISTERS

This window is divided into two sections, one to display Timers and the other one to display Registers.

Timer. It will show one timer per line showing the following information for each one of them:

TG	Indicates the logic status of the active trigger input.
M	Indicates the status of the timer: "S" means stopped, "T" means timing and "D" means disabled.
TEN	Indicates the logic status of the Enable input.
TRS	Indicates the logic status of the Reset input.
T	Indicates the logic status of the status output of the timer.
ET	Indicates the elapsed time.
TO	Indicates the remaining time.

Key in the command **T 1/256** or **T 1/256.1/256** to request the data on a timer or group of timers and then press **ENTER**.

Register. It will display one register per line showing the following information fields for each of them:

HEX	Indicates the hexadecimal value of its contents.
DEC	Indicates the decimal value of its contents (with sign).

Key in **R 1/559** or **R 1/559.1/559** to request information on one or more registers and, then, press **ENTER**.

WINDOW TO DISPLAY COUNTERS AND BINARY DATA

This window is divided into two sections, one to display Counters and the other one to display Binary Data.

Counter. It will display one counter per line showing the following information fields for each of them:

CEN	Indicates the logic status of the ENABLE input.
CUP	Indicates the logic status of the UP COUNT input.
CDW	Indicates the logic status of the DOWN COUNT input.
CPR	Indicates the logic status of the PRESET input.
S	Indicates the status of the counter. “1” when its internal count is 0 and 0 for all other cases.
C	Indicates its count value.

Key in **C 1/256** or **C 1/256.1/256** to request information on one or more counter and, then, press **ENTER**.

Binary Data. It will show one data line per resource or group of resources requested.

The instructions available to request information of the various resources are:

I 1/256 or I 1/256.1/256	It shows the status of the selected input or group of inputs.
O 1/256 or O 1/256.1/256	It shows the status of the selected output or group of outputs.
M 1/5957 or M 1/5957.1/5957	It shows the status of the selected mark or group of marks.
B 0/31 R 1/559	It shows the status of the selected bit of the indicated register.

When requesting the status of one or more inputs, outputs or marks, the CNC will show complete data lines even when all of them have not been requested.

When using generic denominators (I / O / M) to display resources, the CNC will display 20 of them per line and 3 when using their associated mnemonics (symbols). In the latter case, the generic denomination will be displayed when no mnemonic is associated to a resource.

When requesting the status of a register bit, the CNC will display only the requested bit on the corresponding line.

MODIFY WINDOW

With this option it is possible to manipulate the active window (the one selected) by enlarging it, reducing it, clearing it or even eliminating (closing) it.

To do so, the following softkey options are available:

ENLARGE To enlarge the size of the window by one line every time this softkey is pressed.

REDUCE To reduce the size of the window by one line every time this softkey is pressed (minimum 2 lines).

CLEAR To clear the contents of the active window.

CLOSE To close the active window, the CNC will no longer display it.

ACTIVE WINDOW

With this option it is possible to select between the PLC program and each one of the windows being displayed (timers, registers, counters and binary data) in order to operate with it.

Bear in mind that the operator can only operate with the active window.

Once the active window has been selected, it Will be possible to:

Move the cursor (if the PLC program is the one active) or shift the display area with the up and down arrow keys.

Execute any command of the “MODIFY WINDOW” option.

FIND

This option will be executed regardless of which is the active window and it offers the following searching options:

<u>BEGINNING</u> This softkey positions the cursor over the first program block which is then selected quitting the "find" option.

<u>END</u> This softkey positions the cursor over the last program block which is then selected quitting the "find" option.

<u>TEXT</u> With this function it is possible to search a text or character sequence starting from the block indicated by the cursor.

When this key is selected, the CNC requests the character sequence to be found.

The CNC will consider a text found when it is isolated by blank spaces or separators. Thus, When looking for "I1" it will not find or stop at "I12" or "I123", but only at "I1".

When the text is defined, Press the "END OF TEXT" softkey and the cursor will be positioned over the first occurrence of that text.

The search will begin at the current block.

The text found will be highlighted being possible to continue with the search or to quit it.

Press **ENTER** to continue the search up to the end of the program. It is possible to search as many times as wished and when the end of the program is reached, it will start from the first block.

Press the "EXIT" softkey or the **ESC** key to quit the search mode.

ACTIVATE / DEACTIVATE SYMBOLS

With this option it is possible to display in all available windows the symbols or mnemonics associated to the various resources.

The names of the resources may be displayed in two ways: using their generic names (I, O, M, T, C, R) by **deactivating** symbols or using their associated symbols by **activating** them.

When a resource has no mnemonic associated to it, it will always be displayed with its generic name.

This softkey will toggle between ACTIVATE SYMBOL and DEACTIVATE SYMBOL every time is pressed in order to show which option is available.

LOGIC ANALYZER

It is especially suited to help in the machine start-up and for troubleshooting errors and critical situations in signal behavior.

START PLC

When selecting this option, the CNC will start executing the PLC program from the beginning, including the CY1 cycle.

The CNC will ignore this command when it is already executing the PLC program.

FIRST CYCLE

When selecting this option, the CNC will execute only the initial cycle of the PLC program (CY1).

The CNC will ignore this command when it is already executing the PLC program.

SINGLE CYCLE

When selecting this option, the CNC will execute the main cycle of the PLC program (PRG) only once.

The CNC will ignore this command when it is already executing the PLC program.

STOP PLC

This softkey interrupts the execution of the PLC program.

CONTINUE

This softkey resumes the execution of the PLC program.

9.3.1 MONITORING WITH THE PLC IN OPERATION AND WITH THE PLC STOPPED

It must be borne in mind that the CNC initializes all physical outputs and the PLC resources on power-up, after the key sequence SHIFT-RESET and after detecting a WATCHDOG error at the PLC.

The initialization process sets all resources to "0" except those active low. They will be set to "1".

During the monitoring of the PLC program and the various PLC resources, the CNC will always show the real values of the resources.

If the PLC is on, note that a program cycle is processed in the following way:

* The PLC updates the real input values after reading the physical inputs (from the electrical cabinet).

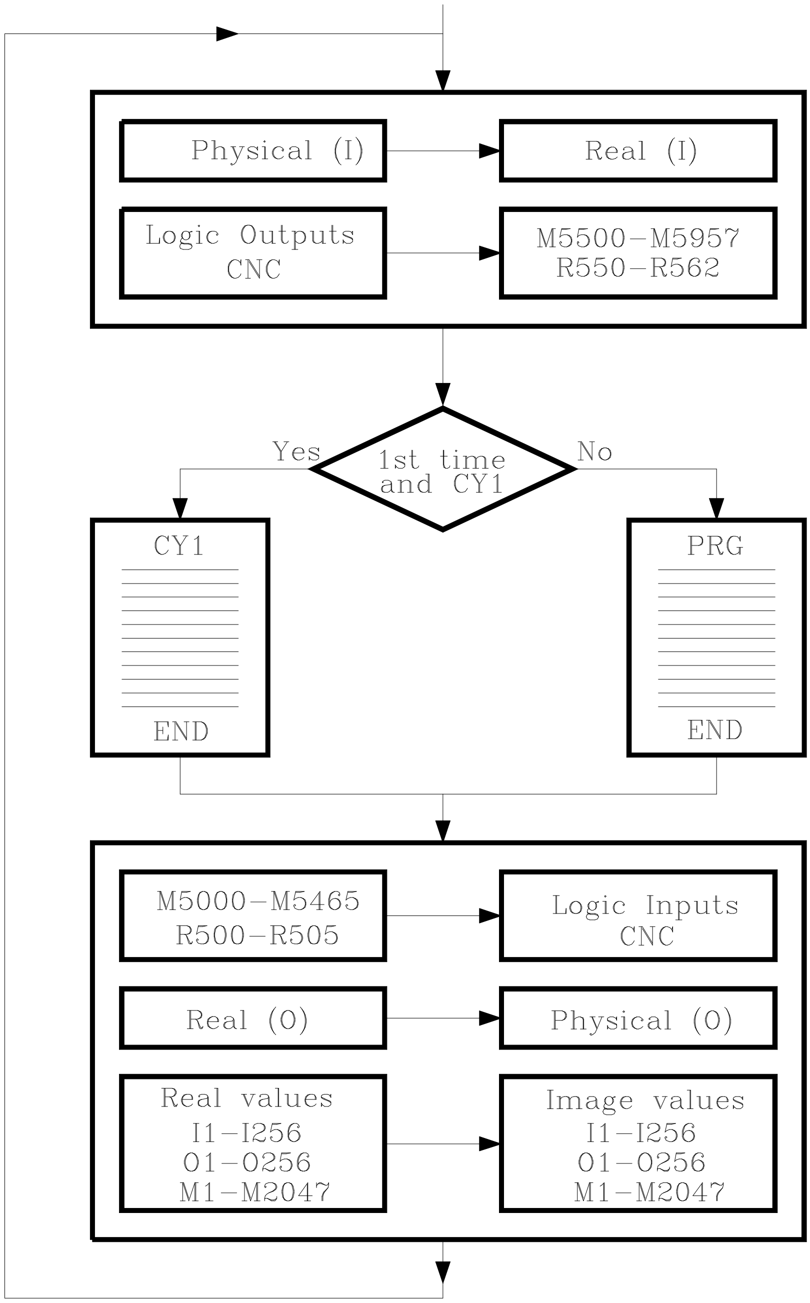
* It updates the values of resources M5000 thru M5957 and R500 thru R559 with the values of the CNC logic outputs (internal variables).

* Executes the program cycle.

* It updates the CNC logic inputs (internal variables) with the real values of resources M5000 thru M5957 and R500 thru R559.

* It assigns to the physical outputs (electrical cabinet) the real values of the corresponding "O" resources.

* It copies the real values of resources I, O, M into their own images.



If the PLC is stopped, it will work as follows:

* The real values of the "I" resources corresponding to the physical inputs will be updated every 10 milliseconds.

* The physical outputs will be updated every 10 milliseconds with the real values of the corresponding "O" resources.

* The PLC will attend to all requests and modifications of its internal variables.

9.4 ACTIVE MESSAGES

When selecting this option, the CNC will display a page (or screen) showing dynamically all the active messages generated by the PLC.

These messages will be listed by priority always starting from the one with the smallest number (highest priority).

The operator can move the cursor a line at a time with the up and down arrow keys or page by page with the page-up and page-down keys.

To delete one of the displayed messages, select it with the cursor and press the “DELETE MESSAGE” softkey.

Note that the CNC dynamically updates the active messages.

9.5 ACTIVE PAGES (SCREENS)

When selecting this option, the CNC will show the active page with the lowest number.

To delete a page or access the other active pages, the CNC will display the following softkey options:

NEXT PAGE	Press this softkey to display the next active page.
PREVIOUS PAGE	Press this softkey to display the previous active page.
CLEAR PAGE	Press this softkey to deactivate the page being displayed.

Note that the CNC dynamically updates the active pages.

9.6 SAVE PROGRAM

Press this softkey to save the PLC_PRG program into the user "Memkey Card" (CARD A).

The PLC program must be stopped before attempting to save it. If it is running, the CNC will ask whether it is desired to stop it or not.

The PLC program must be compiled, otherwise, the CNC will issue an warning message

If the PLC program is running, the CNC requests it to be stopped.

9.7 RESTORE PROGRAM

Press this softkey to restore (recover) the PLC program (PLC_PRG) from the user "Memkey Card" (CARD A) .

The PLC program must not be running any PLC program, otherwise, the CNC will ask whether it is desired to stop it or not.

After executing this instruction, the new source program recovered will replace the one that the PLC previously had. This new one must be compiled and started in order for the PLC to execute it.

9.8 RESOURCES IN USE

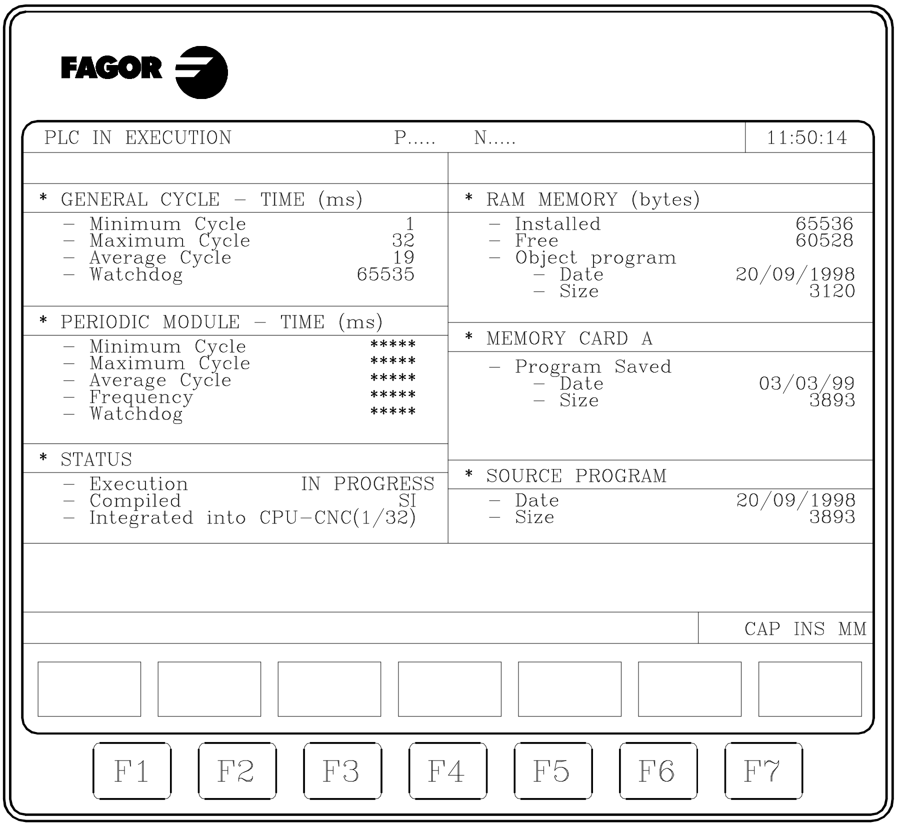
When selecting this option, the CNC will offer the softkeys to select the table of resources used in the PLC program.

The following resource tables are available:

INPUTS (I)
OUTPUTS (O)
MARKS (M)
REGISTERS (R)
TIMERS (T)
COUNTERS (C)

9.9 *STATISTICS*

This option shows the PLC memory distribution, the execution time of the various PLC modules, the PLC program status and the date when it was edited.



GENERAL CYCLE

This section shows the time (maximum, minimum and average) it takes the PLC to execute a program cycle.

This cycle includes:

* Updating the resources with the values of the physical inputs and internal CNC variables.
* Executing both the main cycle (PRG) and the periodic module.
* Updating the internal CNC variables and the physical outputs with the resource variables.
* Copying the resources into their corresponding images.

This section also shows the watchdog time selected by the PLC machine parameter “WDGPRG”.

PERIODIC MODULE

This section shows the time (maximum, minimum and average) that it takes to execute the periodic module of the PLC.

It also shows the period assigned to this module by means of the directive instruction “PE t”. This period indicates how frequently the periodic module will be executed (every “t” milliseconds).

It also shows the watchdog time for this module selected by the PLC machine parameter “WDGPER”.

STATUS

Provides information on the PLC program status indicating whether it is compiled or not and whether it is stopped or in execution.

The line: "integrated into CPU-CNC(1/32)" indicates the time that the system CPU dedicates to the PLC (1 ms for each 32 ms). This value Will defined by the PLC machine parameter “CPUTIME”.

RAM MEMORY

This section indicates the system’s RAM memory available for the exclusive use of the PLC (installed) and it also indicates how much free memory there is.

The object program (executable) is obtained when compiling the source program and is the one executed by the PLC. This section shows the date when it was generated and the RAM memory space it occupies (size).

MEMORY CARD A

This section also shows the date the PLC program (PLC_PRG) was saved into the "Memkey Card" and its size.

SOURCE PROGRAM

This section indicates the date when it was last edited and its size.

The PLC source program is stored in the CNC’s RAM memory.

TIME RESET

Pressing the “TIME RESET” softkey resets the values of the minimum, maximum and average time of the PRG and PE modules so the time is measured from that instant on.

Application examples:

- When the statistics are not exact because the time values of the first cycle of the PLC are too high.
- To know the duration of the PLC cycle from a particular moment on, for example after having changed a parameter.

9.10 LOGIC ANALYZER

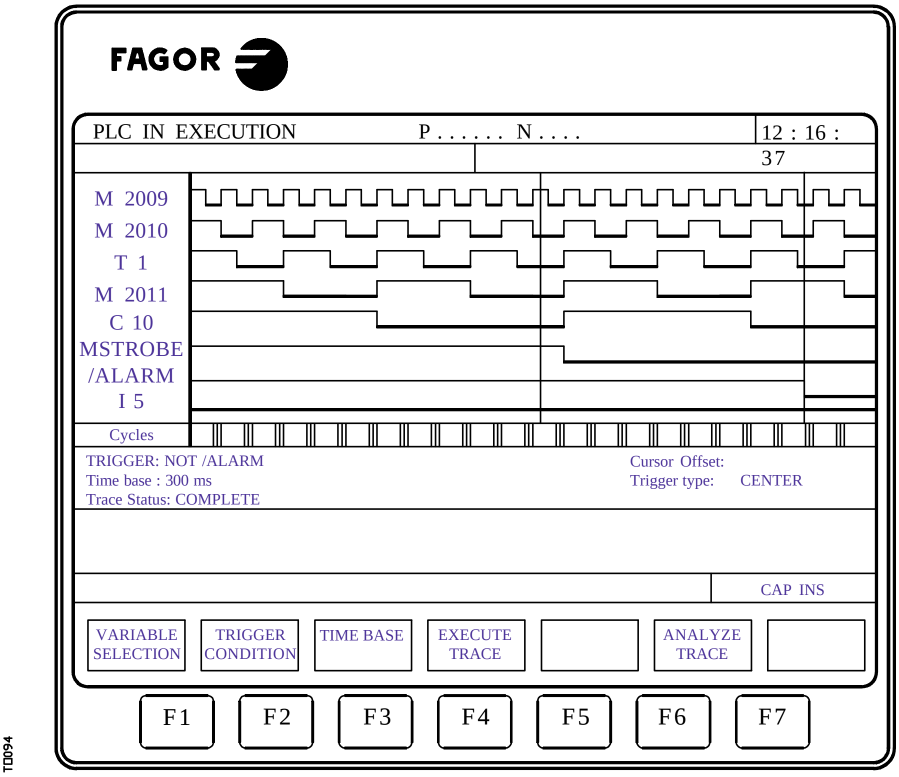
The logic analyzer is especially indicated to perform the machine setup and to determine errors and critical situations in the behavior of the various signals.

With this option it is possible to analyze the behavior of the logic signals of the PLC according to a time base and some trigger conditions established by the user.

Up to 8 signals can be monitored simultaneously. The results are displayed using a graphic interface to simplify the interpretation of the obtained data.

9.10.1 DESCRIPTION OF THE WORK SCREEN

The screen for the logic analyzer can be divided into the following display windows or areas:

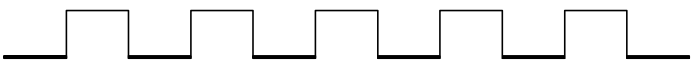


1.- Status window

It displays the graphic representation of the status of each one of the selected signals.

* The variable area shows the names or symbols of the logic signals to be analyzed.

* The status area shows the status of each variable in the shape of square waves. The line corresponding to logic level 0 is shown with a thicker line.



TD095

Also, a vertical red line is displayed to indicate the TRIGGER point and a vertical green line indicating the cursor position.

The green cursor line can be slid right and left along the trace and it can be used to measure the time difference between two of its points.

The status area is divided in several vertical sections. Each of them represents the amount of time established by the "time base" constant.

This constant determines the resolution of the logic signals and, after being defined by the user, can be modified at will. The relationship between the "time base" and the signal resolution is inversely proportional in such way that the smaller the time base, the greater the signal resolution is and vice versa.

2.- Cycle window

This window displays a series of vertical lines "|". Each one of them indicates the instant when a new PLC program cycle starts being executed.

It allows to maintain a relationship between the flow of the logic signals and the duration of each PLC execution cycle.

3.- Information window

This window provides general information about the trace being shown at the time. The shown data is the following:

Trigger	It shows the trigger condition set by the user to do the trace.
Time Base	Indicates the time base set by the user and used to show the current trace.
Trace Status	Indicates the current trace status. The shown texts and their meanings are as follows:

Empty	There is no calculated trace.
Capturing	There is one trace in progress.
Complete	One stored trace is available.

Cursor Offset	Indicates the time difference, in milliseconds, between the cursor position (green line) and the trigger position (red line).
Trigger Type	Indicates the type of trigger selected. The texts shown and their meanings are the following:

Before	The trigger is positioned at the beginning of the trace.
After	The trigger is positioned at the end of the trace.
Center	The trigger is positioned at the center of the trace.
Default	When no trigger condition has been specified.

4.- Editing window

It is the standard CNC editing window. It is used for all the processes requiring data entry.

5.- Message window

The CNC uses this window to display a warning or error message.

9.10.2 SELECTION OF VARIABLES AND TRIGGER CONDITIONS

Before requesting a trace, it is necessary to define the variables to be analyzed, the trigger type and conditions and the time base to be used to display the captured data.

To do this, the following softkey options are available: "**VARIABLE SELECTION**", "**TRIGGER CONDITION**" and "**TIME BASE**".

9.10.2.1 VARIABLE SELECTION

With this option it is possible to select up to 8 variables to be analyzed later.

It displays a cursor over the variable area and it can be slid up and down by means of the up and down arrow keys. The following softkey options will appear:

EDIT

With this option it is possible to edit a new variable or modify one of the currently defined variables.

Before pressing this softkey, we must select, with the cursor, the location where that variable will be shown.

Once this option is selected, the softkeys will change their background color to white and they will show the information corresponding to the editing type possible.

It is possible to analyze any logic signal of the PLC (I3, B1R120, TEN 3, CDW 4, DFU M200, etc.) and it can be referred to by its name or by its associated symbol.

It is also possible to analyze logic expressions, formed with one or more consultations which must follow the syntax and rules used to write the PLC equations.

M100 AND (NOT I15 OR I5) AND CPS C1 EQ 100

Although it might seem difficult to understand the processing of expressions and consultations at a logic analyzer, it should be borne in mind that it could prove very useful when it comes to finding out the status of a whole expression..

It is **not** possible to use more than 16 flank (edge) detecting instructions (DFU and DFD) among all the selected variable definitions and trigger conditions.

By pressing the **ESC** key, the variable being edited will be deleted. From this point on, that variable can be edited again.

Once the variable has been edited, press the **ENTER** key. The new variable will appear in the cursor position inside the variable area.

Only the first 8 characters of the selected variable or expression are shown even when it has more than 8.

The cursor will position at the next variable which will be shown in the editing window, thus being possible to continue editing new variables.

To quit this option, the editing area must be empty. If it is not empty, delete its contents by pressing **ESC** and then press **ESC** again.

DELETE

Use this option to delete a variable.

Before pressing this softkey, use the cursor to select the variable to be deleted.

To delete more variables, repeat these steps for each one of them.

CLEAR ALL

This option deletes all variables from the status window.

9.10.2.2 SELECTION OF TRIGGER CONDITION

The trigger condition as defined is that around which the data capture takes place. This data capture can be done before after or both before and after having met the selected trigger condition.

With this option it is possible to select the trigger type and condition of the logic analyzer. To do this, the following softkey options appear:

EDIT

With this option it is possible to edit the trigger condition around which the data capture will take place.

Once this option is selected, the softkeys will change their background color to white and they will show the information corresponding to the editing type possible.

It is possible to analyze logic expressions, formed with one or more consultations which must follow the syntax and rules used to write the PLC equations.

Examples of expressions and trigger conditions:

M100	The trigger occurs when M100 = 1
NOT M100	The trigger occurs when M100 = 0
CPS R100 EQ 1	The trigger occurs when R100 = 1
NOT I20 AND I5	The trigger occurs when the expression is true

It is **not** possible to use more than 16 flank (edge) detecting instructions (DFU and DFD) among all the selected variable definitions and trigger conditions.

By pressing the **ESC** key, the trigger condition being edited will be deleted. From this point on, that condition can be edited again.

Once the trigger condition has been edited, press ENTER. The new trigger condition will appear at the information window.

If no trigger condition has been specified, the system assumes one by default and it displays the message: "Trigger type: DEFAULT" in the information window. Besides, it will not permit the selection of any other possible types of trigger (before, center or after).

TRIGGER BEFORE

The CNC starts the data capture once (**after**) the selected trigger condition is met.

Then, once the trace has been executed, the trigger (vertical red line) will be positioned at the beginning of the trace.

TRIGGER AFTER

The CNC starts the data capture at the very instant the user selects the option to execute the trace **(before** the trigger condition is met).

The trace will be considered done when the selected trigger condition is met.

The trigger (vertical red line) will be positioned at the end of the trace.

TRIGGER CENTER

The CNC starts the data capture at the very instant the user selects the option to execute the trace.

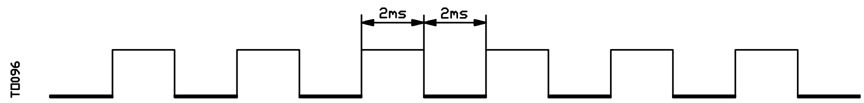
Then, once the trace has been executed, the trigger (vertical red line) will be positioned in the center of the trace.

9.10.2.3 SELECTION OF TIME BASE

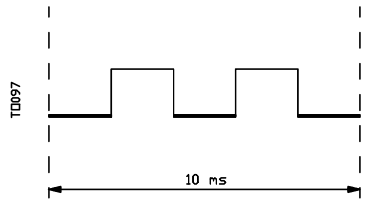
By means of this parameter, the user specifies the amount of time represented by each of vertical intervals.

Since the CRT width of these intervals is always the same, the signal resolution will be established by this time base in such way that the smaller the time base, the greater the signal resolution will be.

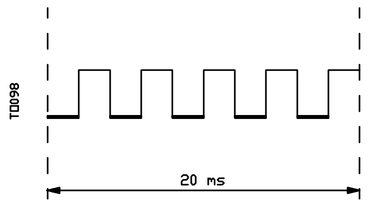
Example: Having a Mark whose status changes every 2 milliseconds.



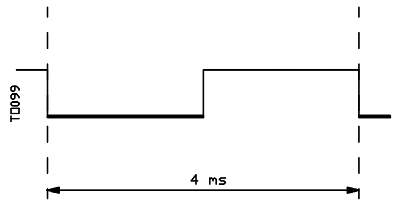
With a time base of 10 milliseconds, it will appear as follows:



With a time base of 20 milliseconds, it will appear as follows:



With a time base of 4 milliseconds, it will appear as follows:



The time base is given in milliseconds and the information window will show the selected value. By default, the CNC assumes a time base of 10 milliseconds.

It is possible to set a time base equal to the frequency of the signal to be monitored and then change it to obtain a finer signal resolution when analyzing the trace.

9.10.3 EXECUTE TRACE

Once having selected the variables and trigger conditions desired, press the "**EXECUTE TRACE**" softkey to indicate to the CNC to begin the data capture.

When the selected trigger condition is met, the trigger line displayed at the information window will change its color.

While the trace is being executed, the information window will display the message: "Trace Status: CAPTURING".

The trace will be completed when the internal memory buffer, dedicated to this function, is full or it is interrupted by pressing the "STOP TRACE" softkey. At this point, the information window will show the message: "Trace Status: COMPLETE".

9.10.3.1 DATA CAPTURE

The data capture takes place at the beginning of each cycle (PRG and PE), after reading the physical inputs and updating the marks corresponding to the CNC logic outputs and just before starting the PLC program execution.

Use this instruction to carry out another data capture while executing the PLC cycle.

This instruction permits the data capture of signals changing at frequencies greater than the cycle time as well as of those changing status during the execution of the cycle while keeping it the same at the beginning and at the end of the cycle.

Example of how to use the "**TRACE**" instruction:

```
PRG
    ________
    ________
    TRACE          ; Data capture
    ________
    ________
    TRACE          ; Data capture
    ________
    ________
    TRACE          ; Data capture
    ________
    ________
END
PE5
    ________
    TRACE          ; Data capture
    ________
END
```

The data capture in the execution of the trace in this program takes place:

- At the beginning of each **PRG** cycle
- Every time the periodic cycle (**PE**) is executed (every 5 milliseconds)
- 3 times while executing the **PRG** module.
- Once while executing the **PE** module.

This way, by means of the "**TRACE**" instruction the data capture can be done any time, especially at those program points considered more critical.

This instruction must only be used when debugging the PLC program and it should be avoided once the PLC program is fully debugged.

9.10.3.2 MODES OF OPERATION

The way the data is captured depends on the type of trigger selected. This section describes the different types of trigger being used as well as the way the data capture is done in each case.

Trigger Before The data capture begins as soon as the selected trigger condition is met, that is when the trigger line shown at the information window changes its color.

The trace will be completed when the trace buffer is full or when the user interrupts it with the "**STOP TRACE**" softkey.

If interrupted before the trigger occurs, the trace will be empty.

Trigger after The data capture begins the instant the user presses the "**EXECUTE TRACE**" softkey.

The trace will be completed when the selected trigger condition is met or it is interrupted by pressing the "**STOP TRACE**" softkey.

If interrupted before the trigger occurs, a trace will be shown with data but without the trigger position (vertical red line).

Trigger center The data capture begins the instant the user presses the "**EXECUTE TRACE**" softkey.

The CNC will enable half the trace buffer to store the data corresponding to the trace prior to the trigger and the other half for the data corresponding to the trace after the trigger.

The trace is completed when its buffer is full or when it is interrupted by pressing the "**STOP TRACE**" softkey.

If interrupted before the trigger occurs, a trace will be shown with data but without the trigger position (vertical red line).

Trigger by Default The CNC carries out this type of trace when no trigger condition has been specified.

The data capture begins the instant the "**EXECUTE TRACE**" softkey is pressed.

The trace is completed when interrupted by pressing the "**STOP TRACE**" showing a trace with data but without the trigger position (vertical red line).

9.10.3.3 *TRACE REPRESENTATION*

Once the data capture is done, the CNC will display graphically in the status window the status of the signals based on the trace calculated for the analyzed variables.

Also, a vertical red line indicating the trigger position and a vertical green line indicating the cursor position will appear superimposed on the trace.

The cursor position (vertical green line) can be slid along the trace by means of the following keys:

Left arrow Moves the cursor one pixel to the left.

While keeping this key pressed, the cursor will advance automatically one pixel at a time and increasing its speed.

If the cursor is positioned at the left end, the trace will be shifted to the right while the cursor stays in the same position.

Right arrow Moves the cursor one pixel to the right.

While keeping this key pressed, the cursor will advance automatically one pixel at a time and increasing its speed.

If the cursor is positioned at the right end, the trace will be shifted to the left while the cursor stays in the same position.

Previous page Moves the cursor one screen to the left.

Next page Moves the cursor one screen to the right.

The CNC will show at all times, in the information window, the cursor position (vertical green line) with respect to the trigger position (vertical red line). This information will appear as "Cursor Offset" and it will be given in milliseconds.

9.10.4 ANALYZE TRACE

Once the data capture is done, the CNC, besides displaying the status window, will enable the "**ANALYZE TRACE**" softkey.

With this option it is possible to position the cursor (vertical green line) at the beginning of the trace, at the end of it or at a specific point along the trace. It is also possible to change the time base for the trace or calculate the time difference between two points of the trace.

To do this, the following softkey options are available:

Find beginning The cursor will position at the beginning of the trace being shown.

Find End It will show the last section of the trace and the cursor will position at the end of it.

Find Trigger It will show the area of the trace corresponding to the trigger zone. The trigger position will appear as a vertical red line over the trace.

The CNC will execute this option when a trigger occurs while analyzing the trace.

Find Time Base When pressing this key, the CNC will request the cursor position with respect to the trigger point. This value is given in milliseconds.

For example: Having selected a "Find time base" of -1000 milliseconds, the CNC will show the trace section corresponding to 1 second prior to the trigger instant.

If no trigger occurred while analyzing the trace, the CNC will assume that the indicated position is referred to the beginning of the trace.

Calculate Times With this option it is possible to find out the time between two points of the trace. To do this, follow these steps in order to set the initial and final points of the calculation.

Position the cursor at the initial point of calculation and press the "**MARK BEGINNING**" softkey to validate it. Use the "left arrow", "right arrow", "page-up" and "page down" keys to move the cursor.

Position the cursor at the final point of calculation and press the "**MARK END**" softkey to validate it.

The CNC will display in the message window the time difference between those two points. It will be given in milliseconds.

This feature can prove very useful to calculate exactly the rise and fall times of a signal, times between two signals, times between the trigger of a signal and the beginning of a cycle, etc.

Modify Time Base This option permits the "Time Base" to be modified.

The status area is divided into several vertical sections. Each of these sections represents a time pitch determined by the "Time Base" constant.

The relationship between the "Time Base" and the signal resolution is inversely proportional in such way that the smaller the "time base", the greater the signal resolution and vice versa.

When pressing this softkey, the CNC will request the new value for the time base. This value must be given in milliseconds.

10. SCREEN EDITOR

In this operating mode, the operator can create up to 256 pages (screens) which will be stored in the "Memkey Card".

The operator can also create up to 256 SYMBOLS to be used when creating the user screens. These symbols are also stored in the "Memkey Card".

The information contained in a page or symbol cannot occupy more than 4Kb of memory. Otherwise, the CNC will issue the corresponding error message.

The user screens stored in the "Memkey Card" may be:

* Used in the screen customizing programs as described next.

* Displayed on power-up (page 0) instead of the FAGOR logo.

* Activated from the PLC.

  The PLC has 256 marks, with their corresponding mnemonics, to select the user screens. These marks are:

M4700	PIC0
M4701	PIC1
M4702	PIC2
——-	——
——-	——-
M4953	PIC253
M4954	PIC254
M4955	PIC255

  When any of these marks is set high, its corresponding screen (page) is activated.

* Used to complete the M function assistance system (screens 250-255).

  When requesting programming assistance for the auxiliary M functions by pressing the [HELP] key, the CNC will show the corresponding internal screen (page).

  When user page 250 is defined, that information will also include the symbol indicating that more help pages are available. By pressing this key, the CNC will display user screen 250.

  The CNC will keep showing that indicator as long as there are more user screens defined (250-255).

  These screens must be defined in a row always starting from page 250. If one of them is missing, the CNC will interpret that there are no more screens defined.

The user screens activated from the PLC may be displayed with the ACTIVE PAGES option of the PLC.

The various options available in this operating mode are:

* **UTILITIES** to manipulate user symbols and screens (edit, copy, delete, etc.).
* **GRAPHIC ELEMENTS** to insert graphic elements in the selected symbol or screen.
* **TEXTS** to insert texts in the selected symbol or screen.
* **MODIFICATIONS** to modify the selected symbol or screen.

10.1 UTILITIES

The various options available in this mode are:

DIRECTORY

To display the directory of user screens and symbols that are stored in the "Memkey Card" (CARD A) or in external devices through the serial lines.

Select the desired device and directory.

The CNC shows the size (in bytes) of each user screen (page) and symbol.

COPY

To make copies within the "Memkey Card" (CARD A) or between the "CARD A" and the external devices.

Examples:

to copy screen (page) 5 from the "Memkey Card" to serial line 2

COPY PAGE 5 IN SERIAL LINE 2 (DNC)

to copy screen (page) 50 from serial line 2 into the "Memkey Card"

COPY SERIAL LINE 2 (DNC) IN PAGE 50 ENTER

to copy symbol 15 as symbol 16 within the "Memkey Card"

COPY SYMBOL 15 IN SYMBOL 16 ENTER

DELETE

To delete a screen or symbol from the "Memkey Card". To do that, proceed as follows:

- Press the [DELETE] softkey
- Press the [PAGE] or [SYMBOL] softkey
- Key in the page or screen number to be deleted and press [ENTER]

The CNC will request confirmation of the command.

RENAME

To assign a new name or comment to a page or symbol of the "Memkey Card".

If there is another one with the same number, the CNC will display a warning message and it will offer the chance to modify the command.

Examples:

to change the page number from 20 to 55

RENAME PAGE 20 TO NEW NUMBER 55 ENTER

to change the comment of symbol 10

RENAME SYMBOL 10 TO NEW COMMENT "Test" ENTER

EDIT

To edit a new user screen (page) or symbol proceed as follows:

- Press the [EDIT] softkey
- Press the [PAGE] or [SYMBOL] softkey
- Key in the page or symbol number
- Press [ENTER]

If the page or symbol does not exists, an empty page will appear in the editing area.

How to edit user screens and symbols is described later on in this chapter.

If the selected screen or symbol has been changed, the CNC will request whether it is to be saved or not in the following instances:

- When exiting the screen editor.
- When selecting another screen (page) or symbol.

SAVE

To save the page or symbol being edited into the "Memkey Card".

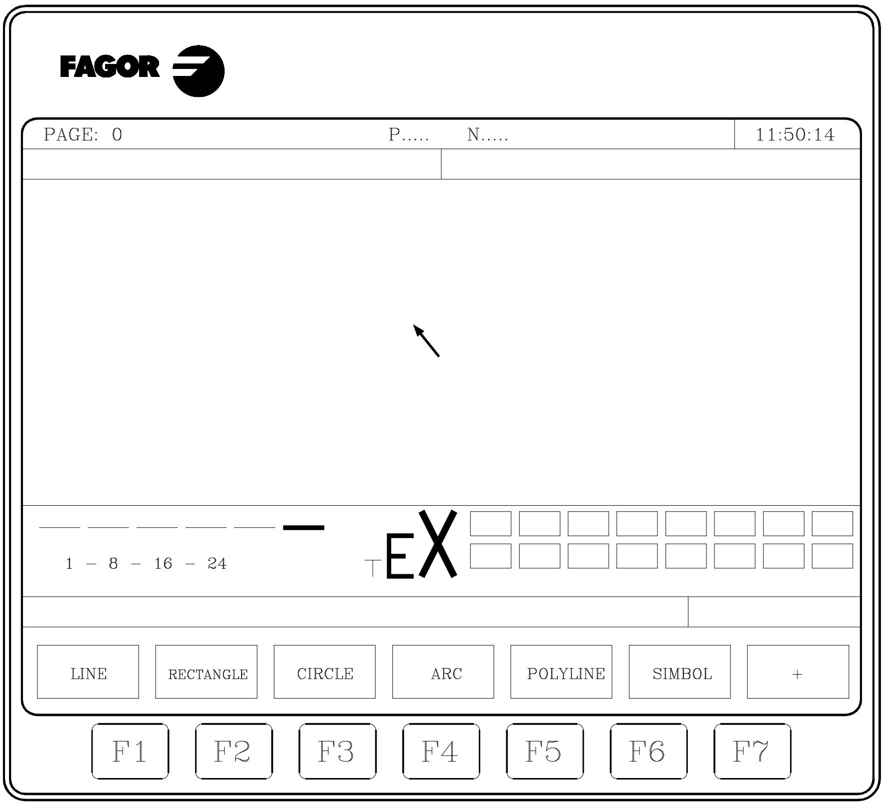
10.2 *EDITING CUSTOM SCREENS (PAGES) AND SYMBOLS*

In order to edit a page or symbol, it is necessary to selected first by means of the EDIT option of the UTILITIES mode of operation.

To edit or modify a page or symbol, use the options: **GRAPHIC ELEMENTS, TEXTS**, and **MODIFICATIONS**.

The information contained in a page or symbol must not occupy more than 4Kb; otherwise, the CNC will issue the corresponding error message.

Once the page or symbol has been selected, the CNC will display a screen similar to this one:



* The upper left-hand side of the screen will show the number of the page or symbol being edited.

* The main window will show the selected page or symbol. When it is a new page or symbol, the main window will be "blank" (blue background).

* There is also a window at the bottom of the screen which shows the different editing parameters and highlights their selected values.

The various parameters available are:

* The type of drawing line used when defining the graphic elements.

* The cursor moving steps (cursor advance) in pixels.

* The letter size to create the texts for the pages and symbols.

* The background and foreground (main) colors for the graphic elements and for the letters.

  One of the color rectangles shown has another rectangle in it. The inside rectangle indicates the selected main color and the outside rectangle indicates the selected background color.

This window also shows The cursor position coordinates in pixels. The horizontal position is indicated by the X value (1 through 638) and the vertical position by the Y value (0 through 334).

Once one of the options (**GRAPHIC ELEMENTS, TEXTS** or **MODIFICATIONS**) has been selected, it will be possible to modify the editing parameters any time.

This way, it will be possible to edit texts and shapes of different color and size.

Press **INS** to access this menu.

Once in this mode, the CNC will show the softkeys corresponding to the various options to modify these parameters. These options are described next.

Press **INS** again to quit this mode and return to the previous menu.

CURSOR ADVANCE

With this option it is possible to select the cursor moving step in pixels (1, 8, 16, 24).

Follow these steps after pressing this softkey:

1.- Use the right and left arrow keys to select the desired step.

The currently selected step will be highlighted.

2.- Press **ENTER** to validate the selected step or **ESC** to quit this mode leaving the previous selection intact.

When editing a new page or symbol, the CNC assumes the default value of 8.

TYPE OF LINE

With this option it is possible to select the type of line used to define the graphic elements.

Follow these steps after pressing this softkey:

1.- Use the right and left arrow keys to select the desired type of line.

The currently selected line type will be highlighted.

2.- Press **ENTER** to validate the selected step or **ESC** to quit this mode leaving the previous selection intact.

When editing a new page or symbol, the CNC assumes the “fine line” by default.

It is not possible to use the thick line to draw polylines or polygons. They are always drawn in fine line.

TEXT SIZE

With this option it is possible to select the size of the letters used to write the texts to be inserted in the pages or symbols.

Three sizes are available:

* Normal size.

  All the characters of the keyboard, numbers, signs, upper and lower case letters, can be written in this size.

* Double and triple sizes.

  Only capital letters **A through Z**, numbers **0 through 9** ; the “*****”, “**+**”, “**-**”, “**.**”, “**:**”, **"#", "%", "/", "<", ">", "?"** signs and the special characters: **"Ç", "Ä", "Ö", "Ü", "ß"** can be written in these sizes.

  When selecting lower case letters for these sizes, the CNC will convert them automatically into upper case.

Follow these steps to select the text size after pressing this softkey:

1.-Use the right and left arrow keys to select the desired size.

The currently selected size will be highlighted.

2.-Press **ENTER** to validate the selected step or **ESC** to quit this mode leaving the previous selection intact.

When editing a new page or symbol, the CNC assumes the normal size by default.

BACKGROUND COLOR

With this option it is possible to select the background color over which the different graphic elements and texts will be edited.

It is not possible to select the background color when editing a symbol since it is an attribute of the page and not of the symbol. Therefore, when inserting a symbol into a page, the symbol will take the background of that page.

If the desired background color is **WHITE**, it is recommended to use a different color while creating the page since the cursor the “drawing” cursor is always white and will become invisible with this background color. Once the complete page (screen) is created, the background color can be changed to the desired one.

One of the color rectangles shown has another rectangle in it. The inside rectangle indicates the selected main color and the outside rectangle indicates the selected background color.

To select the background color, follow these steps:

1.- Use the right and left arrow keys to select the desired color among the 16 shown.

The CNC will show the background color being selected by placing the main-color rectangle inside the rectangle corresponding to the background color being selected.

2.- Press **ENTER** to validate the selected color or **ESC** to quit this mode leaving the previous selection intact.

When editing a new page or symbol, the CNC assumes a blue background color by default.

MAIN COLOR

With this option it is possible to select the color used to draw and write texts on the page (screen) or symbol.

One of the color rectangles shown has another rectangle in it. The inside rectangle indicates the selected main color and the outside rectangle indicates the selected background color.

To select the main color, follow these steps:

1.- Use the right and left arrow keys to select the desired color among the 16 shown.

The CNC will show the main color being selected by placing a white inside rectangle. It will also display the rectangle containing both the selected background color and the main color being selected here.

2.- Press **ENTER** to validate the selected color or **ESC** to quit this mode leaving the previous selection intact.

When editing a new page or symbol, the CNC assumes white as the main color by default.

GRID

This softkey superimposes a grid over the screen in order to facilitate the lay out of the different components of the page or symbol being created or modified. This grid is formed by white or black points (depending on the background color) separated 16 pixels from each-other.

The grid points will be white when the selected background color corresponds to one of the 8 upper color rectangles and they will be black when the selected background color corresponds to one of the 8 lower color rectangles.

Press this softkey again to get rid of the grid.

Every time the grid is displayed, the CNC will reset the cursor advance (step) to 16 pixels.

Therefore, the cursor will move from grid point to grid point every time the arrow keys are pressed to position it on the screen. However, the cursor advance may be modified afterwards by selecting it with the **CURSOR ADVANCE** softkey.

10.3 GRAPHIC ELEMENTS

Before accessing this option, it is necessary to select the page or symbol to be edited or modified by means of the EDIT option of the UTILITIES mode of operation.

With this option it is possible to include graphic elements in the selected page or symbol. The CNC displays a screen 80 columns wide (640 pixels for X coordinate) by 21 rows high (336 pixels for Y coordinate).

When editing a new page, the CNC will position the cursor in the center of the screen and when editing a new symbol, it will position it at the upper left-hand corner.

The cursor is white and can be moved around with the up and down arrow keys and the left and right arrow keys.

The cursor can also be moved by using the following keystroke combinations:

SHIFT	→	Positions the cursor at the last column (X638)
SHIFT	←	Positions the cursor at the first column (X1)
SHIFT	↑	Positions the cursor at the first row (Y0).
SHIFT	↓	Positions the cursor at the last row (Y334).

It is also possible to key in the XY coordinates of the point where the cursor is to be positioned. To do this, follow these steps:

* Press “X” or “Y”.

  The CNC will highlight, in the editing parameter display window, the cursor position along the selected axis (column or row).

* Key in the position value corresponding to the point where the cursor is to be placed along this axis.

  The horizontal position is defined as the X value between 1 and 638 and the vertical position as the Y value between 0 and 334.

  Once these coordinates have been keyed in, press **ENTER** and the CNC will position the cursor at the indicated coordinates.

Once this option is selected, it is possible to modify the editing parameters at any time even while defining the graphic elements. This way, it is possible to edit shapes of different line and color.

Press **INS** to access this menu.

Once in this mode, press the corresponding softkey to modify those parameters.

Press **INS** again to quit this mode and return to the previous menu.

The possible graphic elements which can be used to create a page or symbol are selected with the softkeys and are the following:

LINE

Follow these steps after pressing this softkey:

1.- Place the cursor at the beginning of the line and press **ENTER** to validate it.

2.- Move the cursor to the end of the line (the CNC will continuously show the line being drawn).

3.- Press **ENTER** to validate the line or **ESC** to cancel it.

Repeat the preceding steps to draw more lines. If no more lines are desired, press **ESC** to return to the previous menu.

RECTANGLE

Follow these steps after pressing this softkey:

1.- Place the cursor on one of the corners of the rectangle and press **ENTER** to validate it.

2.- Move the cursor to the opposite corner. The CNC will continuously show the rectangle being drawn.

3.- Press **ENTER** to validate the rectangle or **ESC** to cancel it.

Repeat these steps to draw more rectangles. If no more rectangles are desired, press ESC to return to the previous menu.

CIRCLE

Follow these steps after pressing this softkey:

1.- Place the cursor at the center of the circle and press **ENTER** to validate it.

2.- Move the cursor in order to define the radius. As the cursor moves, the CNC will show the circle corresponding to that radius.

3.- Press **ENTER** to validate the circle or **ESC** to cancel it.

Once the circle is validated, the cursor is positioned at its center in order to facilitate the drawing of concentric circles.

Repeat these steps to draw more circles. If no more circles are desired, press **ESC** to return to the previous menu.

ARC

Follow these steps after pressing this softkey:

1.- Place the cursor at one of the arc's ends and press **ENTER** to validate it.

2.- Move the cursor to the other end of the arc (the CNC will show a line joining both ends) and press **ENTER** to validate it.

The cursor is now positioned automatically at the center of that line.

3.- Move the cursor to define the curvature. The line will become an arc passing through 3 points (the two ends and the cursor point).

4.- Press **ENTER** to validate it or **ESC** to cancel it.

Repeat these steps to draw more arcs. If no more arcs are desired, press ESC to return to the previous menu.

POLYLINE

A polyline consists of several lines where the last point of one of them is the beginning point for the next one.

Follow these steps after pressing this softkey:

1.- Place the cursor at one of the ends of the polyline and press **ENTER** to validate it.

2.- Move the cursor to the end of the first line (which will be the beginning of the next one). The CNC will continuously show the line being drawn.

Press **ENTER** to validate the line or **ESC** to quit this option (which will delete the complete polyline).

3.- Repeat steps 1 and 2 for the rest of the lines.

Note that the maximum number of lines in a polyline is 127.

Once the polyline is drawn, press **ENTER** again to validate it or **ESC** to quit this option deleting the complete polyline.

Repeat these steps to draw more polylines and if no more polylines are desired, press ESC to return to the previous menu.

SYMBOL

This option allows a symbol to be drawn in the page or symbol being edited.

After pressing this softkey, the following steps will be taken.

1.- Enter the number of the symbol to include in the page or symbol being edited and press the ENTER key to validate it.

The CNC will show the cursor situated at the reference point corresponding to the symbol (upper left hand corner of the symbol).

2.- Move the cursor to the position where it is required to place the symbol. In this move, only the cursor will move and not the symbol.

3.- Press the ENTER key to validate it or the ESC key if you wish to quit.

Once the symbol has been validated the CNC will show it in the place indicated.

4.- To include more symbols, repeat the above operations.

5.- Press the ESC key to quit and go back to the previous menu.

If a symbol is being edited this symbol cannot be included in itself. Therefore, if symbol 4 is being edited, any symbol can be included except symbol 4.

Warning:

If a symbol is deleted, the CNC will update all the pages or symbols that contain it because all the calls to it will remain active.

When displaying a page or symbol which has a call to a nonexistent symbol (deleted or not defined), that area of the page will appear blank.

If this symbol is edited again later, the new representation assigned to the symbol will appear in all the pages and symbols which contain a call to it.

POLYGON

A polygon is a closed polyline whose beginning and end points coincide.

After pressing the softkey, the following steps will be taken:

1.- Place the cursor on one of the vertices of the polygon and press the ENTER key to validate it.

2.- Move the cursor to the following vertex of the polygon (the CNC will show the line you are trying to draw).

Press the ENTER key to validate the line or the ESC key if you wish to abandon.

3.- Repeat step 2 for the remaining vertices.

Once all vertices are defined, press the ENTER key and the CNC will complete the polygon or the ESC key if you wish to quit.

The maximum number of sides on the polygon is limited to 127.

FILLED POLYGON

After pressing this softkey, follow the steps as in the POLYGON option, but in this case, after completing the definition of the polygon it will be filled with the color used for its definition.

FILLED CIRCLE

After pressing this softkey follow the steps as in the CIRCLE option, but in this case, after completing the definition of the circle it will be filled with the color used for its definition.

FILLED RECTANGLE

After pressing this softkey follow the steps as in the RECTANGLE option, but in this case, after completing the definition of the rectangle it will be filled with the color used for its definition.

10.4 TEXTS

Before accessing this option, it is necessary to select the page or symbol to be edited or modified by means of the EDIT option of the UTILITIES mode of operation.

With this option it is possible to include texts in the selected page or symbol. The CNC displays a screen 80 columns wide (640 pixels for X coordinate) by 21 rows high (336 pixels for Y coordinate).

When editing a new page, the CNC will position the cursor in the center of the screen and when editing a new symbol, it will position it at the upper left-hand corner.

The cursor is white and can be moved around with the up and down arrow keys and the left and right arrow keys.

The cursor can also be moved by using the following keystroke combinations:

SHIFT ➡ Positions the cursor at the last column (X638)

SHIFT ⬅ Positions the cursor at the first column (X1)

SHIFT ⬆ Positions the cursor at the first row (Y0).

SHIFT ⬇ Positions The cursor at the last row (Y334).

It is also possible to key in the XY coordinates of the point where the cursor is to be positioned. To do this, follow these steps:

* Press “X” or “Y”.

  The CNC will highlight, in the editing parameter display window, the cursor position along the selected axis (column or row).

* Key in the position value corresponding to the point where the cursor is to be placed along this axis.

  The horizontal position is defined as the X value between 1 and 638 and the vertical position as the Y value between 0 and 334.

  Once these coordinates have been keyed in, press **ENTER** and the CNC will position the cursor at the indicated coordinates.

Once this option is selected, it is possible to modify the editing parameters at any time even while defining the graphic elements. This way, it is possible to edit texts of different size and color.

Press **INS** to access this menu.

Once in this mode, press the corresponding softkey to modify those parameters.

Press **INS** again to quit this mode and return to the previous menu.

It is also possible to insert one of the texts available at the CNC or a text previously keyed in by the user. To do this, the following softkey options are available:

USER DEFINED TEXT

Follow these steps to insert the desired text:

1.- Press **ENTER**.

The CNC will display a text editing window. The cursor within this window can be moved with right and left arrow keys.

2.- "Type" the desired text.

A rectangle will be displayed which will enlarge as the text is "typed" in the editing window thus indicating the screen space that this text will occupy.

Press **ESC** to cancel this option and the previous menu will be displayed.

3.- Press **ENTER** once the text has been correctly "typed in".

The typed text will remain in the editing window and the cursor will be positioned in the main window.

4.- Position the rectangle by moving the cursor.

5.- Press **ENTER** to validate this command and the text will replace the rectangle on the screen.

Note that once the text has been "entered", neither its size nor its color can be modified. Therefore, these options must be selected before pressing **ENTER**.

TEXT NUMBER

With this option it is possible to select a text used by the CNC itself in its various operating modes and insert it into the current page or symbol.

To insert one of these predetermined texts, follow these steps:

1.- Press the corresponding softkey.

The CNC will show a screen area to indicate the text number. The cursor may be moved within this area with the right and left arrow keys.

2.- Indicate the desired number by keying it in from the keyboard and press **ENTER**.

The CNC will display the text corresponding to this number and the rectangle indicating the screen space it occupies.

If another text is desired, key in the other number and press **ENTER** again.

Press **ESC** to quit this option without inserting the text and the CNC will show the previous menu.

3.- Once the desired text has been selected, press **ENTER**.

The typed text will remain in the editing window and the cursor will be positioned in the main window.

4.- Position the rectangle by moving the cursor.

5.- Press **ENTER** to validate this command and the text will replace the rectangle on the screen.

Observe that once the text has been “entered”, neither its size nor its color can be modified. Therefore, these options must be selected before pressing **ENTER**.

Warning:

This application may be useful when the pages or symbols being edited are to be shown in other languages since the CNC will translate them into the chosen language.

Usually, when the texts are to be shown in one single language, it is more practical to simply write them up instead of searching them in a list of more than 1500 predetermined messages.

However, should anyone desire the printout of these predetermined texts, feel free to request it from Fagor Automation.

10.5 MODIFICATIONS

Before accessing this option, it is necessary to select the page or symbol to be edited or modified by means of the EDIT option of the UTILITIES mode of operation.

With this option it is possible to include texts in the selected page or symbol. The CNC displays a screen 80 columns wide (640 pixels for X coordinate) by 21 rows high (336 pixels for Y coordinate).

When editing a new page, the CNC will position the cursor in the center of the screen and when editing a new symbol, it will position it at the upper left-hand corner.

The cursor is white and can be moved around with the up and down arrow keys and the left and right arrow keys.

The cursor can also be moved by using the following keystroke combinations:

SHIFT	➡	Positions the cursor at the last column (X638)
SHIFT	⬅	Positions the cursor at the first column (X1)
SHIFT	⬆	Positions the cursor at the first row (Y0).
SHIFT	⬇	Positions the cursor at the last row (Y334).

It is also possible to key in the XY coordinates of the point where the cursor is to be positioned. To do this, follow these steps:

* Press "X" or "Y".

The CNC will highlight, in the editing parameter display window, the cursor position along the selected axis (column or row).

* Key in the position value corresponding to the point where the cursor is to be placed along this axis.

The horizontal position is defined as the X value between 1 and 638 and the vertical position as the Y value between 0 and 334.

Once these coordinates have been keyed in, press **ENTER** and the cursor will be positioned at the indicated coordinates.

The possible options to modify a page or symbol are:

CLEAR PAGE

Allows the selected page or symbol to be deleted.

Once this softkey has been pressed, the CNC will request an OK before executing the indicated operation.

If this option is executed, the CNC will delete the page or symbol being edited, but it will keep in the "Memkey Card" the contents of that page or symbol the last time the "SAVE" command was executed.

DELETE ELEMENTS

This option allows an element of the displayed page or symbol to be selected and then deleted.

To do this follow these steps:

1.- Place the cursor in the position to delete an element and press the **ENTER** key to validate it.

An area of between ± 8 pixels from the position indicated will be analyzed.

If the element to be deleted is a filled circle or a filled polygon, the cursor must be positioned on a point on the circumference or external polygon (periphery).

2.- If any graphic element or text exists in this area, this will be highlighted and you will be asked if you wish to delete it.

Press the ENTER key to delete this element, otherwise the ESC key.

Should there be several elements in this area, the CNC will highlight them in succession and it will ask for confirmation before deleting any of them.

MOVE SCREEN

With this option it is possible to reposition the whole page (not its individual elements separately) and it can only be used to move pages and not symbols.

It allows the entire page to be moved with the right, left, up and down arrow keys.

The center of the page is taken as a reference for this movement.

To do this follow these steps:

1.- The CNC will show the page with the cursor placed in the middle of the screen.

2.- Move the cursor to the position to place the page reference point.

Press ESC to quit this option without making any changes and the CNC will show the previous menu.

Repeat these steps to perform more moves, otherwise, press ESC and the CNC will show the previous menu.

11. *MACHINE PARAMETERS*

In order for the machine tool to execute the programmed instructions correctly, the CNC must know specific data on the machine such as feedrates, accelerations, feedbacks, automatic tool changes, etc.

This data is determined by the manufacturer of the machine and must be stored in the machine parameter tables.

These tables may be edited in this work mode or copied into the "Memkey Card" or a PC as described later on.

The CNC has the following groups of machine parameters:

* General machine parameters
* Axis parameters (one table per axis)
* Spindle parameters
* Drive parameters
* RS-422 and RS-232-C serial port configurations
* Ethernet configuration parameters
* PLC parameters
* M miscellaneous functions
* Leadscrew error compensation (one table per axis)
* Cross Compensations between two axes (for example: Beam sag).

First, the general machine parameters must be set as by means of these the machine axes are defined and therefore the Axis Parameter tables.

It must also be defined whether the machine has cross compensation and between which axes, and the CNC will generate the corresponding cross compensation parameters.

By means of the general machine parameters, the table lengths for the Tool Magazine, Tools, Tool Offsets and the miscellaneous M functions are defined.

By means of the Axis Parameters it is defined whether the axis has Leadscrew error Compensation or not and the length of the corresponding table.

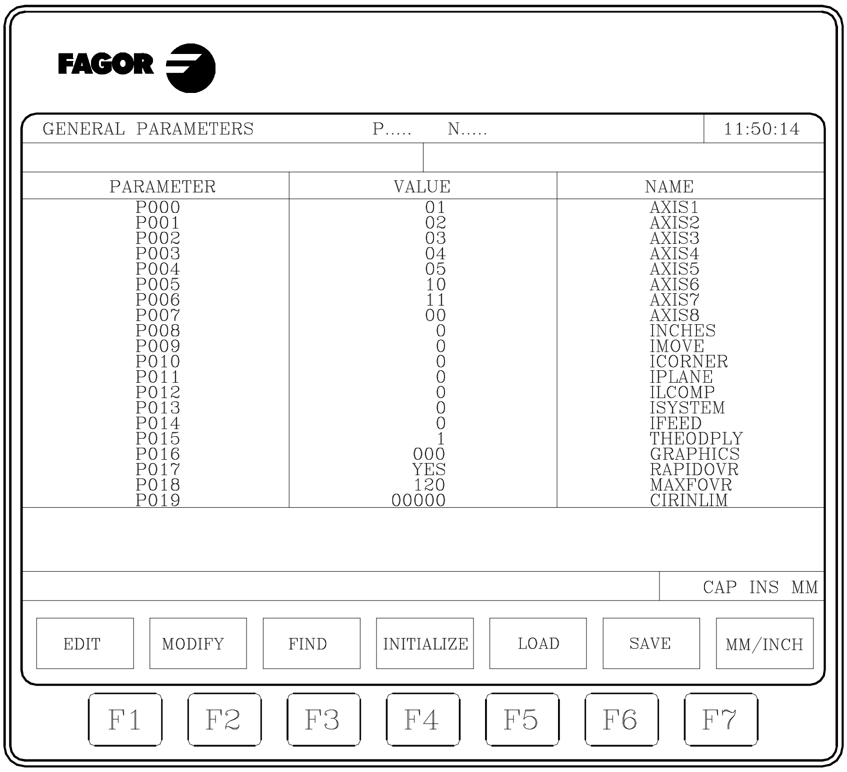
Once the general machine parameters are defined, press **SHIFT RESET** for the CNC to enable the required tables.

It is recommended to save the tables in the "Memkey Card" or out to a peripheral device or PC.

When accessing this operating mode, the CNC will show the tables that are saved in the "Memkey Card" (CARD A).

11.1 MACHINE PARAMETER TABLES

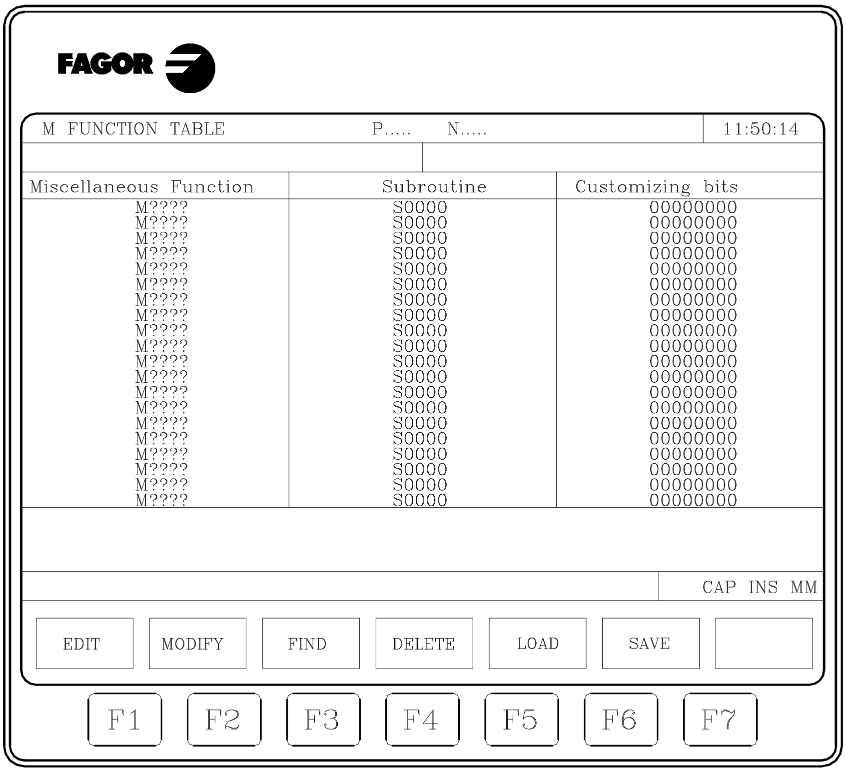
The General, Axis, Spindle, Serial ports and PLC tables have the following structure:



Where the parameter number is indicated, the value assigned to it and the name or mnemonic associated with this parameter.

11.2 MISCELLANEOUS FUNCTION TABLES

The table corresponding to the miscellaneous M functions has the following structure:



The number of M functions in the table is defined by means of the general machine parameter "NMISCFUN". The following is defined for each line:

* The number (0-9999) of the defined miscellaneous M functions:

If an M function is not defined, the CNC will show M????.

* The number of the subroutine to be associated with this miscellaneous function.

* 8 customizing bits

x	x	x	x	x	x	x	x
7	6	5	4	3	2	1	0

Bit 0 Indicates whether the CNC must (=0) or must not (=1) wait for the signal AUXEND (signal of the M executed) to resume program execution.

Bit 1 Indicates whether the M function is executed before (=0) or after (=1) the movement of the block in which it is programmed.

Bit 2 Indicates whether the execution of the M function interrupts (=1) or not (=0) the preparation of the blocks.

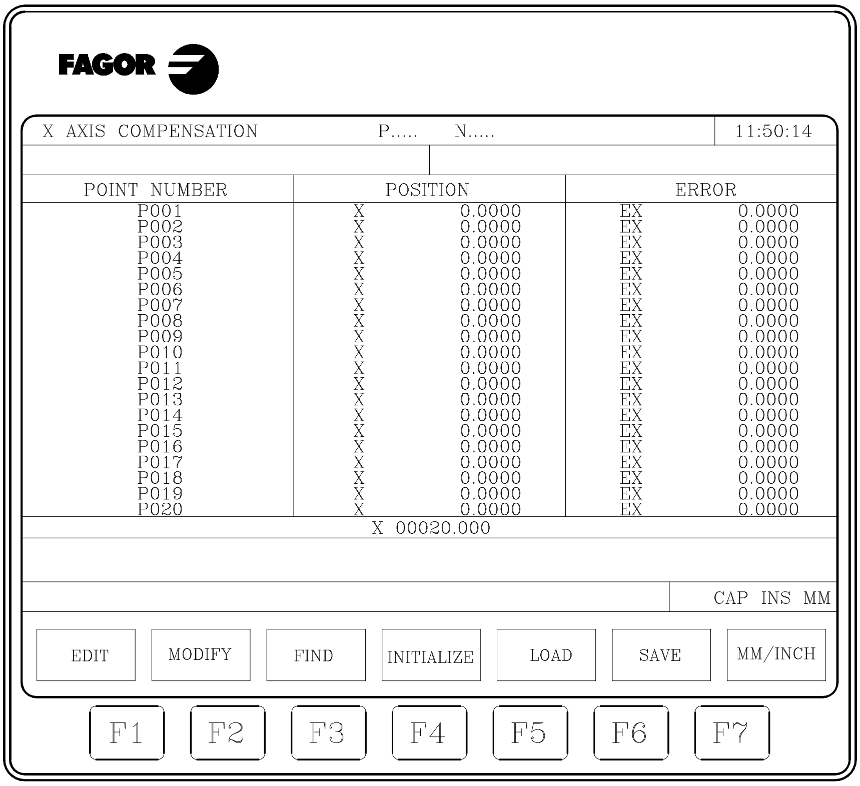
Bit 3 Indicates whether the M function is executed after calling the associated subroutine (=0) or only the associated subroutine is executed (=1).

Bit 4 When bit 2 is set to "1", it indicates whether block preparation is to be interrupted until the "M" function starts executing (=0) or until its execution is finished (=1).

The rest of the bits are not being used at this time.

11.3 LEADSCREW ERROR COMPENSATION TABLES

The tables for leadscrew error compensation have the following structure:



The number of points of each of these is defined by means of the axis machine parameter "NPOINTS". The following is defined for each of line:

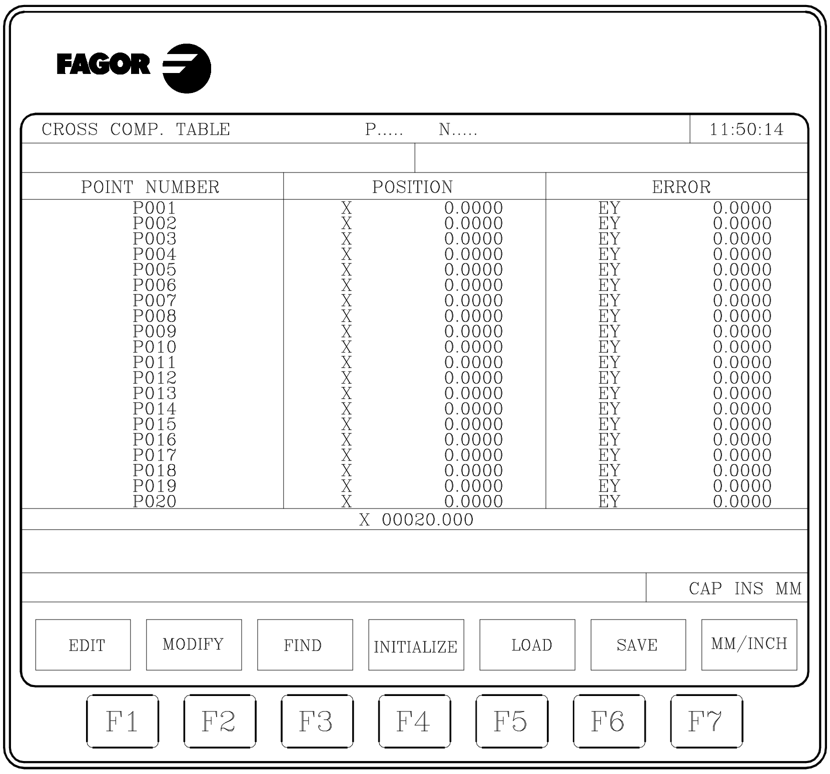
* Position of the axis to be compensated.

* Error of this axis in this position.

Also, the current position of the selected axis is displayed and updated as the machine axis moves.

11.4 CROSS COMPENSATION TABLES

The tables corresponding to cross compensation have the following structure.



The number of points of each table is defined by means of the general machine parameter “NPCROSS”, "NPCROSS2" and "NPCROSS3" respectively.

Each table defines:

* The position of the axis causing the error.

* The error suffered by the axis at that point.

Also, the current position of the selected axis is displayed. This position is updated as the axis moves.

11.5 *OPERATION WITH PARAMETER TABLES*

Once one of the tables has been selected, the cursor can be moved over the screen line by line by means of the “up and down arrow keys” or move from page to page by means of the “page up and page down keys”.

In addition, the user has an area of the screen for editing, it being possible to move the cursor over the screen by means of the “right arrow key and left arrow key”.

The CNC offers the following softkey options for each table:

EDIT

The desired parameter.

When selecting this option, the softkeys will change their color to a white background and they will show the various editing options.

In those tables corresponding to leadscrew and cross compensation, the position values of the axis must be edited as follows:

* Move the axis and when the error is found large enough to be considered, press the softkey corresponding to this axis.
* The CNC will include, in the editing area, the name of the axis followed by the position value corresponding to that point.

  This value can be modified if so desired.
* Press the softkey corresponding to the error and key in its value.

Once the parameter is edited, press **ENTER**. This new parameter will be included in the table and the cursor will be positioned over it. The editing area will be cleared, thus allowing other parameters to be edited.

Press **ESC** to quit this mode.

MODIFY

With this option it is possible to modify the selected parameter.

Before pressing this softkey, the desired parameter must be selected.

When selecting this option, the softkeys will change their color to a white background and they will show the various editing options.

By pressing **ESC**, the information displayed in the editing window (corresponding to the selected parameter) will be cleared. From this point on, a new value can be entered.

To quit this option, first clear the editing window using the **CL** key or the **ESC** key and then press **ESC** again. The selected parameter will not be modified.

Once this modification has concluded, press the **ENTER** key to validate it.

FIND

The beginning or end of the table, or the parameter whose number is indicated by positioning the cursor on the required parameter.

<u>BEGINNING</u> When pressing this softkey, the cursor positions over the first parameter of the table quitting this option.

<u>END</u> When pressing this softkey, the cursor positions over the last parameter of the table quitting this option.

<u>PARAMETER</u> When pressing this softkey, the CNC will request the number of the parameter to be found. Key in that number and press **ENTER**. The cursor will be positioned over the indicated parameter quitting this option.

INITIALIZE

With this option it is possible to reset all the parameters of the selected table to their default values.

These default values are indicated in the chapter corresponding to machine parameters in the installation manual.

LOAD

To load tables stored in the "Memkey Card" (CARD A) or in a peripheral device or PC through the two serial lines (RS232C or RS422).

The transmission begins after pressing the corresponding softkey. When using a serial line, the receptor must be ready before starting the transmission.

To interrupt the transmission, press the [ABORT] softkey.

If the length of the table received does not coincide with the length of the current table, the CNC will acts as follows:

If the table received is shorter than the current one, the received lines are modified and the rest keep their previous values.

If the table received is longer than the current one, the CNC updates all the lines of the current table and when detecting that there is no more room, the CNC issues the corresponding error message.

SAVE

The tables may be saved in the "Memkey Card" (CARD A) or in a peripheral device or PC through the two serial lines (RS232C or RS422).

The transmission begins after pressing the corresponding softkey. When using a serial line, the receptor must be ready before starting the transmission.

To interrupt the transmission, press the [ABORT] softkey.

MM/INCHES

Every time this softkey is pressed, the CNC will change the display format of those parameters affected by these units from millimeters to inches and vice versa.

The lower right-hand window will show the units currently selected.

Note that this change does not affect the general machine parameter "**INCHES**" which indicates the measuring units by default.

12. *DIAGNOSIS*

In this operating mode it is possible to know the configuration of the CNC as well as testing the system.

The CNC offers the following softkey options:

System Configuration
Hardware test
Tests
Adjustments
User
Hard disk

12.1 CONFIGURATION

This option shows the current system configuration.

Once this option has been chosen, two new softkeys will appear in order to select the hardware configuration or the software configuration of the system.

12.1.1 HARDWARE CONFIGURATION

This option shows the following information:

CENTRAL UNIT (CPU) CONFIGURATION
It indicates the current configuration of the CNC's CPU: Power supply, boards, video, CAN, etc..

CNC RESOURCES
It indicates, in Kb:
The RAM memory available for the User and for the system.
The memory capacity of the "Memkey Card".

PLC RESOURCES
It indicates:
whether the PLC is integrated into the CPU-CNC
the number of local and remote inputs and outputs.

LCD ADJUSTMENT (Softkey available only for monochrome LCD monitors)
Pressing this softkey shows the new set of softkeys for adjusting the brightness / contrast on the adjustment screen.
The [+] and [-] softkeys may be used to adjust the brightness / contrast.
Pressing the [RESTORE] softkey restores the previous values.
Pressing the [SAVE] softkey assumes the new values.

12.1.2 SOFTWARE CONFIGURATION

This option shows:

* All available software options.

* The software version installed.
  Both for the CNC and the hard disk (HD) module.

* The id codes of the unit. They are only to be used by the Service Department.

The [CODE VALIDATION] softkey must be used after consulting with the Service Department when wishing to implement more software features.

12.2 HARDWARE TEST

This option checks the power supply voltages corresponding to the system and to the boards as well as the internal temperature of the central unit. It displays the following information:

SUPPLY VOLTAGE

It indicates the voltage of the lithium battery and the voltages supplied by the Power Supply.

The voltages supplied by the Power Supply are internally used by the CNC.

Next to the voltages, it displays the value range (maximum and minimum values), the real value and whether it is OK or not.

BOARD VOLTAGE

It indicates which boards must be supplied with 24Vdc and whether each one of them are suplied with the correct voltage

INSIDE TEMPERATURE

It shows the value range (maximum and minimum values), the inside temperature of the Central Unit and whether that value is OK or not.

12.3 TESTS

12.3.1 MEMORY TEST

This option checks the status of the internal CNC memory, that of the memory available for the User and for the System.

To carry out this verification, the PLC program must be stopped, otherwise, the CNC will ask the operator whether this operation is to be carried out or not.

12.3.2 CODE TEST

This option checks the status of the internal CNC Flash memory. These memories contain the CNC software version currently installed.

12.4 ADJUSTMENTS

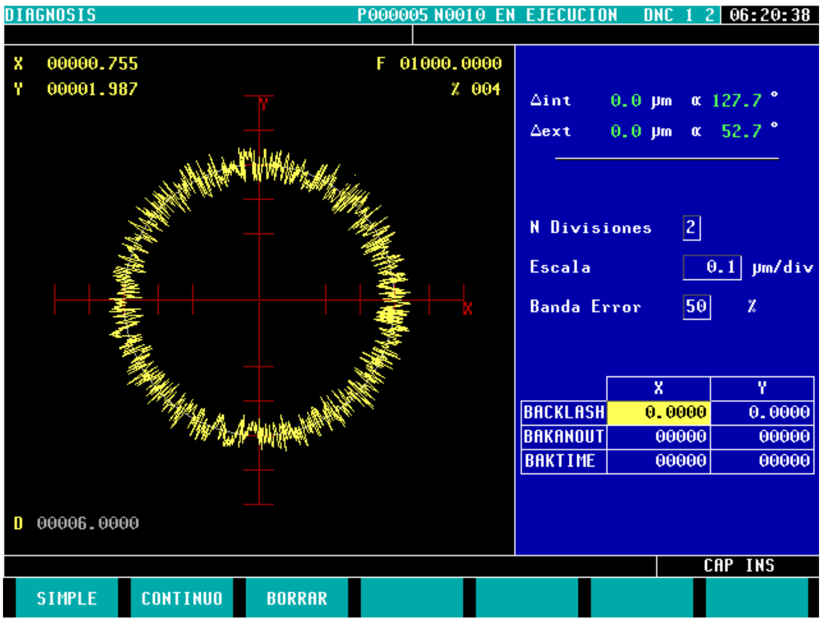
12.4.1 CIRCLE GEOMETRY (BALLBAR) TEST

With this adjustment, it is possible to improve the peak observed when reversing the axes. It consists in machining a circle (without compensation) and compare it with the graph shown by the CNC.

The following example shows a program for machining repetitive circles.

```
        X0 Y0
        G5 G1 F1000
N10     G2 X0 Y0 I20 J0
        (RPT N10, N10) N50
        M30
```

After selecting this program in the execution mode and running it, access the Diagnosis mode, Adjustment, Circle Geometry Test and the CNC will display this screen:



If the machine parameters are protected, it will request the access password because some of them are shown at the bottom right hand side.
If the password is not known, those values cannot be modified, however, the screen and the circle geometry test may be accessed.

The left side of the CNC screen shows the result of the test.
The data at the top right are refreshed by the CNC at the end of the test.
The data at the center right must be defined before running the test.
The bottom right side of the screen shows the parameters associated with the plane axes and their values.

Before running the test, the graphic representation on the left must be defined. To do this, set the data on the center right side of the screen:

Number of divisions, from left to right, of the theoretical circle.
Scale or value, in microns, of each division.
Margin of error or percentage of circle radius occupied by the error margin (divisions area).

When knowing the password for the machine parameters, the values shown at the bottom right may be modified. The CNC assigns the new values to the relevant machine parameters. Therefore, the initial values should be jotted down for future reference.

Once the graphic display area and the machine parametes have been defined, proceed to capture the data by pressing the following softkeys:

SIMPLE
: It deletes whatever was drawn and starts drawing, over the theoretical circle, the machining error magnified according to the scale previously defined until completing a whole circle or until the [STOP] softkey or the ESC key is pressed.

CONTINUOUS
: It deletes whatever was drawn and starts drawing, over the theoretical circle, a series of circles with the machining error magnified according to the scale previously defined until the [STOP] softkey or the ESC key is pressed.

DELETE
: It may be pressed at any time, even while drawing on the screen. It clears the screen and resets the statistics shown on its right.

During continuous drawing, it is possible to modify the machine parameters and observe the new drawing superimposed on the previous one press the [DELETE] softkey to display only the new one.

The data appearing at the top right are updated while capturing data.

? internal   Maximum negative value of the error over the theoretical radius, in microns or tenth-thousandths of an inch, and its angular position.

? external   Maximum positive of the error over the theoretical radius, in microns or tenth-thousandths of an inch, and its angular position.

Once data has been capture, it draws two lines to indicate the angular positions of both errors on the graph. They are show with dashes when the error exceeds the value assigned to the display area in its quadrant and goes over to the opposite quadrant..

Note: While capturing points for the geometry test, it does not draw the execution graphics.

12.5 USER

This option will execute the program which is selected with the general machine parameter "**USERDIAG**" in the user channel.

To quit its execution and return to the previous menu, press **ESC**

12.6 HARD DISK

Once this option has been selected, two softkeys will be displayed:

Test — It check the status of the hard disk (user memory available). It takes about 30 minutes.
In order to perform this test, the PLC program must be stopped. If it is running, the CNC will ask the operator whether it is to be stopped or not.

Compress — It compresses the hard disk by defragmenting it. It also includes a hard disk surface check. The duration of this test depends on the number of files it contains and on how defragmented the hard disk is.

12.7 INTERESTING NOTES

The **CNC** carries out a series of sequential tests.

If the result obtained is not correct, it may stop axes feed and spindle rotation (by cancelling their analog voltages and Enables), as well as stopping the execution of the **PLC** program or activating the external **EMERGENCY** output (**01**).

Test type	**When is it carried out?**	**Stops the axes and the spindle**	**Stops the PLC**	**Activates Emergency output**
Temperature	Always	YES	No	YES
Battery out	Always	No	No	No
Flash memory	From Diagnosis	YES	YES	V
(CARD A)	On power-up	No	No	No
RAM memory	FromDiagnosis	YES	No	YES
External emergency	EXEC/SIMUL	YES	No	YES
Board voltage	EXEC/SIMUL	YES	No	YES
PLC running	EXEC/SIMUL	YES	---	YES
PLC user error	EXEC/SIMUL	YES	No	No
PLC Watchdog	PLC running	YES	YES	YES

CNC 8040

Programming Manual
(M model)

Ref. 0204-ing

The information described in this manual may be subject to variations due to technical modifications.

FAGOR AUTOMATION, S.Coop. Ltda. reserves the right to modify the contents of the manual without prior notice.

Please note that the availability of some of the features described in this manual depends on the software options you just obtained.

	GP model	M model
Number of axes with standard software	4	4
Number of axes with optional software	7	7
Solid graphics	-----	Option
Irregular pockets with islands	-----	Option
Rigid tapping	Option	Option
Tool life monitoring	-----	Option
Probing canned cycles	-----	Option
DNC	Option	Option
COCOM version	Option	Option
Profile editor	Option	Option
Tool radius compensation	Option	Standard
Tangential control	Option	Option
Retracing	-----	Option
Electronic threading	-----	Standard
Tool magazine management	-----	Standard
Machining canned cycle	-----	Standard
Multiple machining	-----	Standard
Conversational software (MC and MCO)	-----	Option

INDEX

VERSION HISTORY

INTRODUCTION

1. OVERVIEW

2. CREATING A PROGRAM

3. AXES AND COORDINATE SYSTEMS

4. REFERENCE SYSTEMS

5. PROGRAMMING BY ISO CODE

6. PATH CONTROL

7. ADDITIONAL PREPARATORY FUNCTIONS

8. TOOL COMPENSATION

9. CANNED CYCLES

10. MULTIPLE MACHINING

11. IRREGULAR POCKET CANNED CYCLE (WITH ISLANDS)

12. WORKING WITH A PROBE

13. PROGRAMMING IN HIGH-LEVEL LANGUAGE

14. PROGRAM CONTROL STATEMENTS

APPENDIX

VERSION HISTORY (M).
(Mill Model)

April 2002
Software: 5.3x.

Feature	Manual
New expansion board models for the 8055i	Installation Programming
Bus CAN OPEN to control remote digital I/Os at the 8055i	Installation Operation Error solutions
New PLC instructions: IREMRD and OREMWR:	Installation
Leadscrew error compensation on rotary axes between 0-360 degrees	Installation
PLC statistic deletion with a single softkey	Operation
Show only the XY plane in top-view graphics	Operation
Absolute reference mark management via Sercos (see a.m.p. REFVALUE)	Installation

Introduction

SAFETY CONDITIONS

Read the following safety measures in order to prevent damage to personnel, to this product and to those products connected to it.

This unit must only be repaired by personnel authorized by Fagor Automation.

Fagor Automation shall not be held responsible for any physical or material damage derived from the violation of these basic safety regulations.

Precautions against personal damage

Before powering the unit up, make sure that it is connected to ground

In order to avoid electrical discharges, make sure that all the grounding connections are properly made.

Do not work in humid environments

In order to avoid electrical discharges, always work under 90% of relative humidity (non-condensing) and 45º C (113º F).

Do not work in explosive environments

In order to avoid risks, damage, do no work in explosive environments.

Precautions against product damage

Working environment

This unit is ready to be used in Industrial Environments complying with the directives and regulations effective in the European Community

Fagor Automation shall not be held responsible for any damage suffered or caused when installed in other environments (residential or homes).

Install the unit in the right place

It is recommended, whenever possible, to instal the CNC away from coolants, chemical product, blows, etc. that could damage it.

This unit complies with the European directives on electromagnetic compatibility. Nevertheless, it is recommended to keep it away from sources of electromagnetic disturbance such as.

- Powerful loads connected to the same AC power line as this equipment.
- Nearby portable transmitters (Radio-telephones, Ham radio transmitters).
- Nearby radio / TC transmitters.
- Nearby arc welding machines
- Nearby High Voltage power lines
- Etc.

Ambient conditions

The working temperature must be between +5° C and +45° C (41ºF and 113º F)
The storage temperature must be between -25° C and 70° C. (-13º F and 158º F)

Precautions during repair

Do not manipulate the inside of the unit

Only personnel authorized by Fagor Automation may manipulate the inside of this unit.

Do not manipulate the connectors with the unit connected to AC power.

Before manipulating the connectors (inputs/outputs, feedback, etc.) make sure that the unit is not connected to AC power.

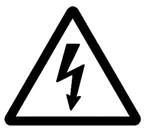
Safety symbols

Symbols which may appear on the manual

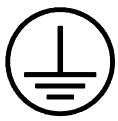
WARNING. symbol
It has an associated text indicating those actions or operations may hurt people or damage products.

Symbols that may be carried on the product

WARNING. symbol
It has an associated text indicating those actions or operations may hurt people or damage products.



"Electrical Shock" symbol
It indicates that point may be under electrical voltage



"Ground Protection" symbol
It indicates that point must be connected to the main ground point of the machine as protection for people and units.

MATERIAL RETURNING TERMS

When returning the Monitor or the Central Unit, pack it in its original package and with its original packaging material. If not available, pack it as follows:

1.- Get a cardboard box whose three inside dimensions are at least 15 cm (6 inches) larger than those of the unit. The cardboard being used to make the box must have a resistance of 170 Kg (375 lb.).

2.- When sending it to a Fagor Automation office for repair, attach a label indicating the owner of the unit, person to contact, type of unit, serial number, symptom and a brief description of the problem.

3.- Wrap the unit in a polyethylene roll or similar material to protect it.

    When sending the monitor, especially protect the CRT glass

4.- Pad the unit inside the cardboard box with poly-utherane foam on all sides.

5.- Seal the cardboard box with packing tape or industrial staples.

FAGOR DOCUMENTATION FOR THE CNC

OEM Manual	Is directed to the machine builder or person in charge of installing and starting-up the CNC.
USER-M Manual	Is directed to the end user or person who will operate this CNC in M mode. It contains 2 manuals: Operating Manual — describing how to operate the CNC. Programming Manual — describing how to program the CNC.
USER-T Manual	Is directed to the end user or person who will operate this CNC in T mode. It contains 2 manuals: Operating Manual — describing how to operate the CNC. Programming Manual — describing how to program the CNC.
MC Manual	Is directed to the end user or person who will operate this CNC in MC mode.
TC Manual	Is directed to the end user or person who will operate this CNC in TC mode.
MCO/TCO Manual	Is directed to the end user or person who will operate this CNC in MCO/ TCO mode.
DNC Software Manual	Is directed to people using the optional DNC communications software.
DNC Protocol Manual	Is directed to people wishing to design their own DNC communications software to communicate with the CNC.
FLOPPY DISK Manual	Is directed to people using the Fagor Floppy Disk Unit and it shows how to use it.

1. *OVERVIEW*

The CNC can be programmed both at the machine (from the front panel) or from external peripheral devices (tape reader/cassette recorder, computer, etc. Memory available to the user for carrying out the part programs is 1 Mbyte.

The part programs and the values in the tables which the CNC has can be entered as follows :

* **From the front panel**. Once the editing mode or table required has been selected, the CNC allows you to enter data from the keyboard.

* **From a Computer (DNC) or Peripheral Device**. The CNC allows data to be interchanged with a computer or peripheral device, using RS232C and RS422 cables.

  If this is controlled from the CNC, it is necessary to preset the corresponding table or part program directory (utilities) you want to communicate with.

  Depending on the type of communication required, the serial port machine parameter “PROTOCOL” should be selected.

  “PROTOCOL” = 0 if the communication is with a peripheral device.
  “PROTOCOL” = 1 if the communication is via DNC.

1.1 PART-PROGRAMS

Editing

To create a part-program, access the Edit mode. See chapter 5 in this manual.

The new part-program edited is stored in the CNC's RAM memory.

A copy of the part-programs may be stored in the "MemKey Card", at a PC connected through serial line 1 or 2 or in the hard disk (HD). See chapter 7 in this manual.

When using a PC through serial line 1 or 2, proceed as follows:

• Execute the "Fagor50.exe" applications program at the PC.
• Activate DNC communications at the CNC. See chapter 8 in this manual.
• Select the work directory as shown in chapter 7 of this manual. Option: Utilities\ Directory\ Serial L.\ Change directory.

With the Edit mode of operation, part-programs residing in the CNC's RAM memory may be modified. To modify a program stored in the "MemKey Card", in a PC or in the hard disk, it must be previously copied into RAM memory.

Execution

Part-programs stored anywhere may be executed or simulated. See chapter 3 in this manual.

The user customizing programs must be in RAM memory so the CNC can execute them.

The GOTO and RPT instructions cannot be used in programs executed from a PC connected through the serial lines. See chapter 14 of the programming manual.

The subroutines can only be executed if they reside in the CNC's RAM memory. Therefore, to execute a subroutine stored in the "MemKey Card", in a PC or in the hard disk, it must be first copied into the CNC's RAM memory.

From a program in execution, another program can be executed which is in RAM memory, in the "MemKey Card", in a PC or in the hard disk using the EXEC instruction. See chapter 14 of the programming manual.

Utilities

This operating mode, chapter 7 of this manual, lets display the part-program directory of all the devices, make copies, delete, rename and even set the protections for any of them.

Ethernet

When having the Ethernet option and if the CNC is configured as another node within the computer network, the following operations are possible from any PC of the network:

- Access the part-program directory of the Hard Disk(HD).
- Edit, modify, delete, rename, etc.the programs stored on the hard disk (HD).
- Copy programs from the hard disk to the PC and vice versa.

To configure the CNC as another node within the computer network, see section 3.3.4 of the installation manual.

Operations that may be carried out with part-programs:

	RAM Memory	CARD A	HD	DNC
Consult the program directory in ...	Yes	Yes	Yes	Yes
Consult the subroutine directory in ...	Yes	No	No	No
Create work directory in ..	No	No	No	No
Change work directory in ..	No	No	No	Yes
Edit a program in ..	Yes	No	No	No
Modify a program in ..	Yes	No	No	No
Delete a program from ..	Yes	Yes	Yes	Yes
Copy from/to RAM memory to/from ...	Yes	Yes	Yes	Yes
Copy from/to CARD A to/from ...	Yes	Yes	Yes	Yes
Copy from/to HD to/from ...	Yes	Yes	Yes	Yes
Copy from/to DNC to/from ...	Yes	Yes	Yes	Yes
Rename a program in ..	Yes	Yes	Yes	No
Change the comment of a program in ..	Yes	Yes	Yes	No
Change protections of a program in ..	Yes	Yes	Yes	No
Execute a part-program in ..	Yes	Yes	Yes	Yes
Execute a user program in ..	Yes	No	No	No
Execute the PLC program in ..	Yes	*	No	No
Execute programs using the GOTO or RPT instructions from ..	Yes	Yes	Yes	No
Execute subroutines stored in ..	Yes	No	No	No
Execute programs stored in RAM, CARD A or HD using the EXEC instruction from ..	Yes	Yes	Yes	Yes
Execute programs via DNC with the EXEC instruction from ..	Yes	Yes	Yes	No
Open programs stored in RAM, CARD A or HD using the OPEN instruction from ..	Yes	Yes	Yes	Yes
Open programs via DNC using the OPEN instruction from ..	Yes	Yes	Yes	No
Consult from a PC and through Ethernet, the program directory in ...	No	No	Yes	No
Consult from a PC and through Ethernet, the subroutine directory in ...	No	No	No	No
Create from a PC and through Ethernet, a directory in...	No	No	No	No

* If it is not in RAM memory, it generates an executable code in RAM and it executes it..

1.1.1 CONSIDERATIONS FOR THE ETHERNET CONNECTION

When configuring the CNC as another node in the computer network , the programs stored in the hard disk module (HD) may be edited and modified from any PC.

Instructions for seting up a user PC to access CNC directories

Recommended configuration:

- Open the «Windows Explorer»
- On the «Tools» menu, select the «Connect to Network Drives» option.
- Select the Drive. For example: «D»
- Indicate the path: CNC name followed by the name of the shared directory. For example: \\FAGORCNC\CNCHD
- When selecting the option: «Connect again when initiating the session», the selected CNC will appear on each power-up as another path of the «Windows Explorer» without having to define it again.

This connection is established through Ethernet and, therefore, the CNC does not control the syntax of the programs while they are received or modified.

Whenever accessing the program directory of the Hard Disk (HD), the following verification takes place:

File name.

The file number must always have 6 digits and the extension PIM (for milling) or PIT (for lathe). Examples: 001204.PIM 000100.PIT

If the file has been given the wrong name, for example: 1204.PIM or 100.PIT, the CNC will not change it, but it will display it with the comment: '********************'.

The file cannot be modified from the CNC. It must be edited from the PC to correct the error.

File size.

If the file is empty, size = 0, the CNC will display it with the comment: '********************'.

The file can be edited or deleted either from the CNC or from the PC.

First line of the program

The first line of the program must have the % character, the comment associated with the file (up to 20 characters) and between the 2 commas (,) the program attributes: O (OEM), H (hidden), M (modifiable), X (executable).

Examples %Comment ,MX,
% ,OMX,

If the first line does not exist. The CNC will display it with an empty comment and with the modifiable (M) and executable (X) attributes.

When the format of the first line is wrong, the CNC does not modify it, but it displays it with the comment: '********************'. The file can be modified or deleted from the CNC or from the PC.

It is the wrong format when:

- the comment has more than 20 characters
- a comma (,) is missing for separating the attributes
- the attributes have a strange character

1.2 DNC CONNECTION

The CNC offers as optional feature the possibility of working in DNC (Distributed Numerical Control), enabling communication between the CNC and a computer to carry out the following functions:

* Directory and delete commands.
* Transfer of programs and tables between the CNC and a computer.
* Remote control of the machine.
* The ability to supervise the status of advanced DNC systems.

1.3 COMMUNICATION PROTOCOL VIA DNC OR PERIPHERAL DEVICE

This type of communication enables program-and-table transfer commands, plus the organization of CNC directories such as the Computer Directory, for copying/deleting programs, etc. to be done either from the CNC or the computer.

When you want to transfer files, it is necessary to follow this protocol:

* The "%" symbol will be used to start the file, followed by the **program comment** (optional), of up to 20 characters.

  Then, and separated by a comma ",", comes the attribute (protection) each file has: reading, modifying, etc. This protection is optional and does not have to programmed.

  To end the file header, **RETURN (RT)** or **LINE FEED (LF)** characters should be sent separated by a comma (",").

  Example:

  %Fagor Automation, -MX, RT

* Following the header, the file blocks should be programmed. These will all be programmed according to the programming rules indicated in this manual. After each block, to separate it from the others, the **RETURN (RT)** or **LINE FEED (LF)** characters should be used.

  Example:

  N20 G90 G01 X100 Y200 F2000 LF
  (RPT N10, N20) N3 LF

If communication is made with a peripheral device, you will need to send the 'end of file' command. This command is selected via the machine parameter for the serial port: "EOFCHR", and can be one of the following characters:

ESC ESCAPE
EOT END OF TRANSMISSION
SUB SUBSTITUTE
EXT END OF TRANSMISSION.

2. *CREATING A PROGRAM*

A CNC (numerical control) program consists of a series of blocks or instructions.

These blocks or instructions are made of words composed of capital letters and numerical format.

The CNC's numerical format consists of :

- the symbols **. + -**
- the figures **0 1 2 3 4 5 6 7 8 9**

Programming allows spaces between letters, numbers and symbols, in addition to ignoring the numerical format if it has zero value, or a symbol if it is positive.

The numerical format of a word can be replaced by an arithmetic parameter in programming. Later and during basic execution, the control will replace the arithmetic parameter by its value, for example :

If XP3 has been programmed, during execution the CNC will replace P3 by its numerical value, obtaining results such as X20, X20.567, X-0.003, etc.

2.1 CREATING A PROGRAM IN THE CNC

All the blocks which make up the program have the following structure :

Block header + program block + end of block

2.1.1 BLOCK HEADER

The block header is optional, and may consist of one or more **block skip conditions** and by the **block number or label**. Both can be programmed in this order.

CONDITION FOR BLOCK SKIP, /, /1, /2, /3.

These three block skip conditions, given that “/” and “/1” is the same, are governed by the marks BLKSKIP1, BLKSKIP2 and BLKSKIP3 of the PLC.

If any of these marks is active, the CNC will not execute the block or blocks in which it has been programmed. The execution takes place in the following block.

Up to 3 skip conditions can be programmed in one block. These will be evaluated one by one, respecting the order in which they have been programmed.

The control reads 20 blocks ahead of the one being executed in order to calculate in advance the path to be run.

The condition for block skip will be analyzed at the time when the block is read i.e. 20 blocks before execution.

If the block skip needs to be analyzed at the time of execution, it is necessary to interrupt the block preparation, by programming G4 in the previous block.

BLOCK LABEL OR NUMBER N(0-9999)

This is used to identify the block, and is only used when block references or jumps are made.

They are represented by the letter N followed by up to 4 figures (0-9999). It is not necessary to follow any order, and randomly arranged numbers are allowed.

If two or more blocks with the same label number are present in the same program, the CNC will always give priority to the first number.

Although it is not necessary to program it, by using a SOFTKEY the CNC allows the automatic programming of labels. The programmer can select the initial number and the step between labels.

2.1.2 PROGRAM BLOCK

This is written with commands in ISO and High Level languages.

To prepare a program, blocks written in both languages will be used, although each one should be edited with commands in just one language.

2.1.2.1 ISO LANGUAGE

This language is specially designed to control axis movement, as it gives information and movement conditions, in addition to data on feedrate. It includes :

* Preparatory functions for movement, used to determine geometry and working conditions, such as linear and circular interpolations, threading, etc.

* Control functions for axis feedrate and spindle speeds.

* Tool control functions.

* Complementary functions, with technological instructions.

2.1.2.2 HIGH LEVEL LANGUAGE

This enables access to general purpose variables and to system tables and variables.

It gives the user a number of control sentences which are similar to the terminology used in other languages, such as : IF, GOTO, CALL, etc.

It also allows the use of any type of expression (arithmetic, referential, or logical).

It also has instructions for the construction of loops, plus subroutines with local variables. "Local variable" is understood to mean one which is only recognized by the subroutine in which it has been defined.

It is also possible to create libraries, grouping subroutines with useful and tested functions, which can be accessed from any program.

2.1.3 END OF BLOCK

The end of block is optional and may consist of the indication of **number of repetitions of the block** and of the **block comment**. Both must be programmed in this order.

NUMBER OF REPETITIONS OF THE BLOCK, N(0-9999)

This indicates the number of times the block will be executed.

Movement blocks can only be repeated which, at the time of their execution, are under the influence of a modal subroutine.

In these cases, the CNC executes the programmed move and the active machining operation (canned cycle or modal subroutine) the indicated number of times.

The number of repetitions is represented by the letter N followed by up to 4 digits (0-9999).

The active machining operation does not take place if N0 is programmed. Only the movement programmed within the block takes place.

BLOCK COMMENT

The CNC allows you to incorporate any kind of information into all blocks in the form of a comment.

The comment is programmed at the end of the block, and should begin with the character ";".

If a block begins with ";", all its contents will be considered as a comment, and it will not be executed.

Empty blocks are not permitted. They should contain at least one comment.

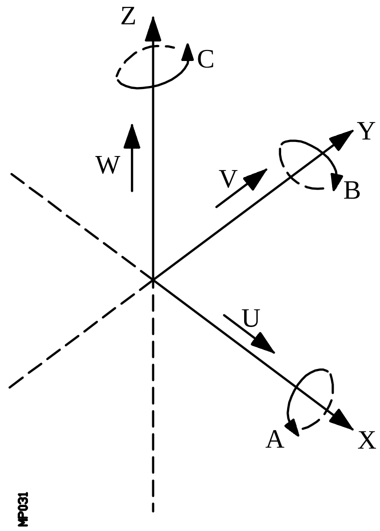
3. *AXES AND COORDINATE SYSTEMS*

Given that the objective of the CNC is to control the movement and positioning of axes, it is necessary to determine, by means of coordinates, the position of the point to be reached.

The CNC allows you to use absolute, relative or incremental coordinates throughout the same program.

3.1 NOMENCLATURE OF THE AXES

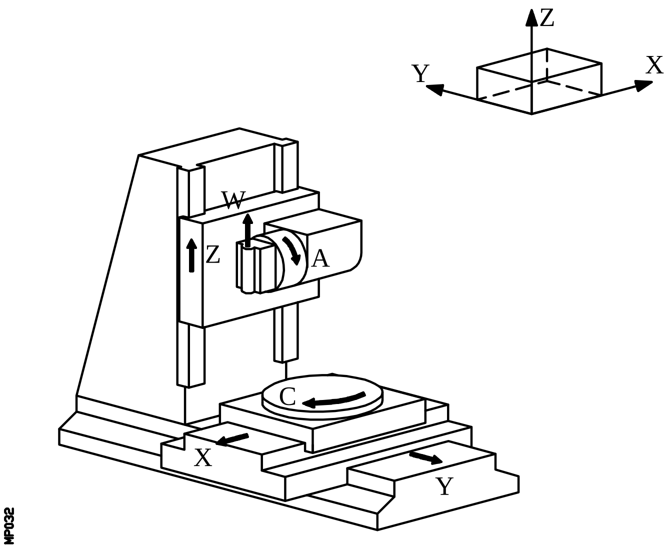
The axes are named according to DIN 66217.



Characteristics of the system of axes :

* **X & Y**: main movements on the main work plane of the machine.
* **Z**: parallel to the main axis of the machine, perpendicular to the main XY plane.
* **U,V,W**: auxiliary axes parallel to X,Y, Z respectively
* **A,B,C**: rotary axes on each of the X,Y, Z axes.

The drawing below shows an example of the nomenclature of the axes on a milling-profiling machine with a tilted table.



3.1.1 *SELECTION OF THE AXES*

Of the 9 possible axes which can exist, the CNC allows the manufacturer to select up to 7 of them.

Moreover, all the axes should be suitably defined as linear/rotary, etc. through the axis machine parameters which appear in the Installation and Start-up Manual.

There is no limitation to the programming of the axes, and interpolations can be made simultaneously with up to 7 axes.

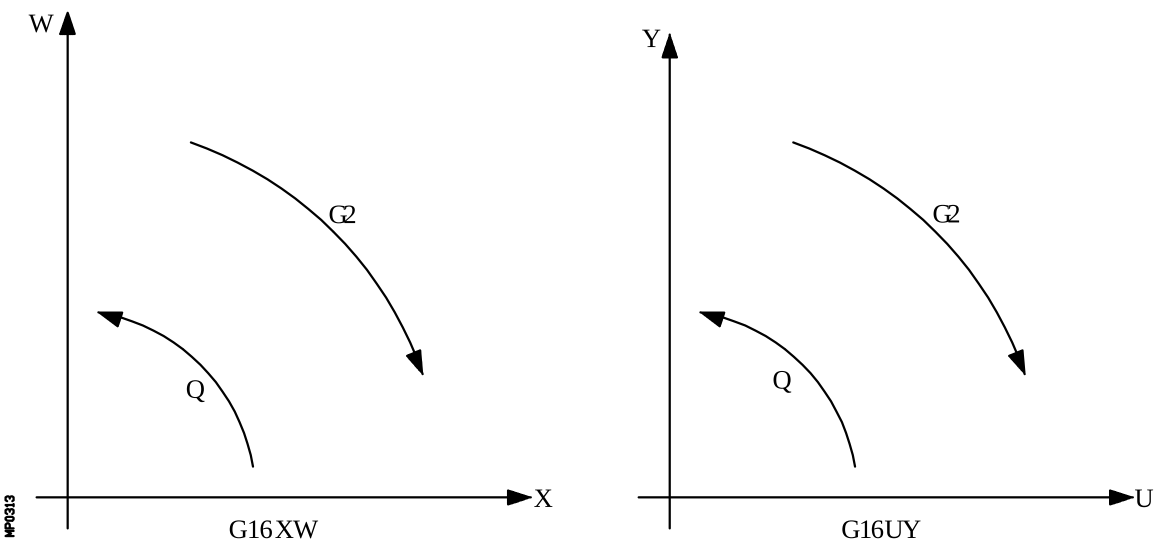
3.2 *PLANE SELECTION (G16, G17, G18, G19)*

Plane selection should be made when the following are carried out :

- Circular interpolations.
- Controlled corner rounding.
- Tangential entry and exit.
- Chamfer blend.
- Machining canned cycles.
- Pattern rotation.
- Tool radius Compensation.
- Tool length compensation.

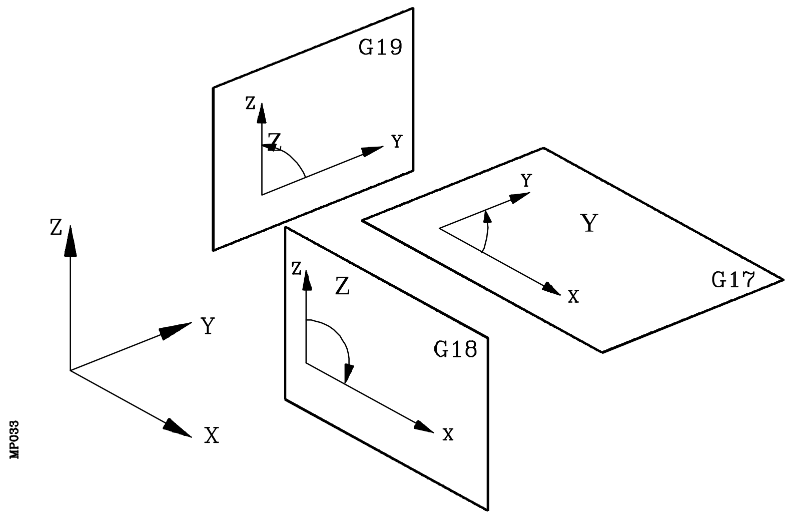
The “G” functions which enable selection of work planes are as follows :

* **G16 axis1 axis2**. Enables selection of the desired work plane, plus the direction of G02 G03 (circular interpolation), **axis1** being programmed as the abscissa axis and **axis2** as the ordinate axis.



* **G17**. Selects the XY plane
* **G18**. Selects the ZX plane
* **G19**. Selects the YZ plane

The G16, G17, G18 and G19 functions are modal and incompatible among themselves. The G16 function should be programmed on its own within a block.



The G17, G18, and G19 functions define two of the three main axes (X, Y, Z) as belonging to the work plane, and the other as the perpendicular axis to the same.

When radius compensation is done on the work plane, and length compensation on the perpendicular axis, the CNC does not allow functions G17, G18, and G19 if any one of the X, Y, or Z axes is not selected as being controlled by the CNC.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC will assume that the plane defined by the general machine parameter as "IPLANE" is the work plane.

3.3 PART DIMENSIONING. MILLIMETERS (G71) OR INCHES (G70)

The CNC allows you to enter units of measurement with the programming, either in millimeters or inches.

It has a general machine parameter “INCHES” to define the unit of measurement of the CNC.

However, these units of measurement can be changed at any time in the program. Two functions are supplied for this purpose :

* G70 : Programming in inches.
* G71 : Programming in millimeters.

Depending on whether G70 or G71 has been programmed, the CNC assumes the corresponding set of units for all the blocks programmed from that moment on.

The G70 and G71 functions are modal and are incompatible.

The CNC allows the programming of figures from 0.0001 to 99999.9999 (with or without sign) when it works in millimeters (G71). This is called format +/- 5.4, or from 0.00001 to 3937.00787 (with or without sign) if it is programmed in inches (G70). This is called format +/- 4.5.

However, and to simplify the instructions, we can say that the CNC admits +/- 5.5 format, thereby admitting +/- 5.4 in millimeters and +/- 4.5 in inches.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC will assume that the system of units of measurement is the one defined by the general machine parameter “INCHES”.

3.4 *ABSOLUTE/INCREMENTAL PROGRAMMING (G90, G91)*

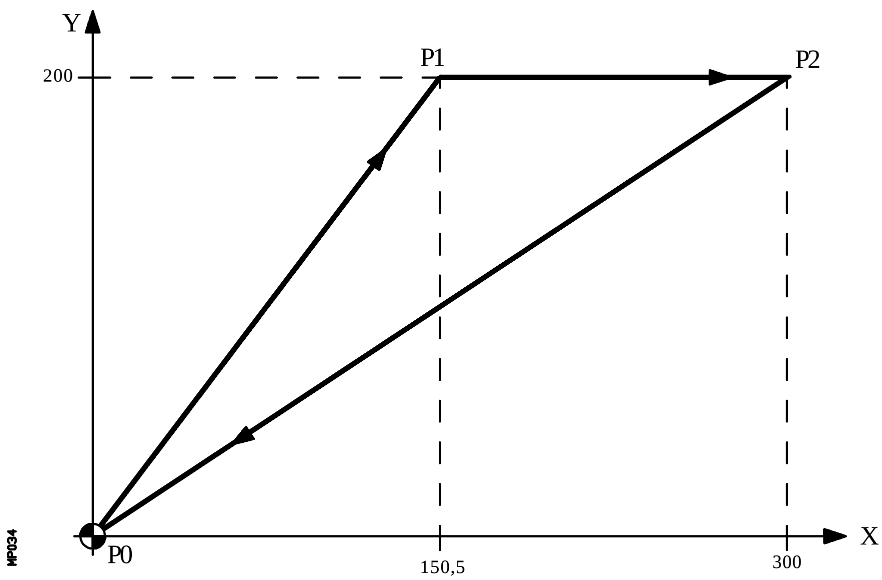
The CNC allows the programming of the coordinates of one point either with absolute **G90** or incremental **G91** values.

When working with absolute coordinates (G90), the point coordinates refer to a point of origin of established coordinates, often the part zero (datum).

When working in incremental coordinates (G91), the numerical value programmed corresponds to the movement information for the distance to be travelled from the point where the tool is situated at that time. The sign in front shows the direction of movement.

Functions G90/G91 are modal and incompatible.

Example :



Absolute coordinates

G90	XO	YO	;	Point P0
	X150.5	Y200	;	Point P1
	X300		;	Point P2
	X0	Y0	;	Point P0

Incremental coordinates

G90	X0	Y0	;	Point P0
G91	X150.5	Y200	;	Point P1
	X149.5		;	Point P2
	X-300	Y-200	;	Point P0

On power-up, after executing M02, M30 or after an EMERGENCY or RESET, the CNC will assume G90 or G91 according to the definition by the general machine parameter "ISYSTEM".

3.5 PROGRAMMING OF COORDINATES

The CNC allows the selection of up to 7 of the 9 possible axes X,Y,Z,U,V,W,A,B,C.

Each of these may be linear, linear to position only, normal rotary, rotary to position only or rotary with hirth toothing (positioning in complete degrees), according to the specification in the machine parameter of each “AXISTYPE” axis.

With the aim of always selecting the most suitable coordinate programming system, the CNC has the following types :

* Cartesian coordinates
* Polar coordinates
* Cylindrical coordinates
* Angle and one Cartesian coordinate.

3.5.1 CARTESIAN COORDINATES

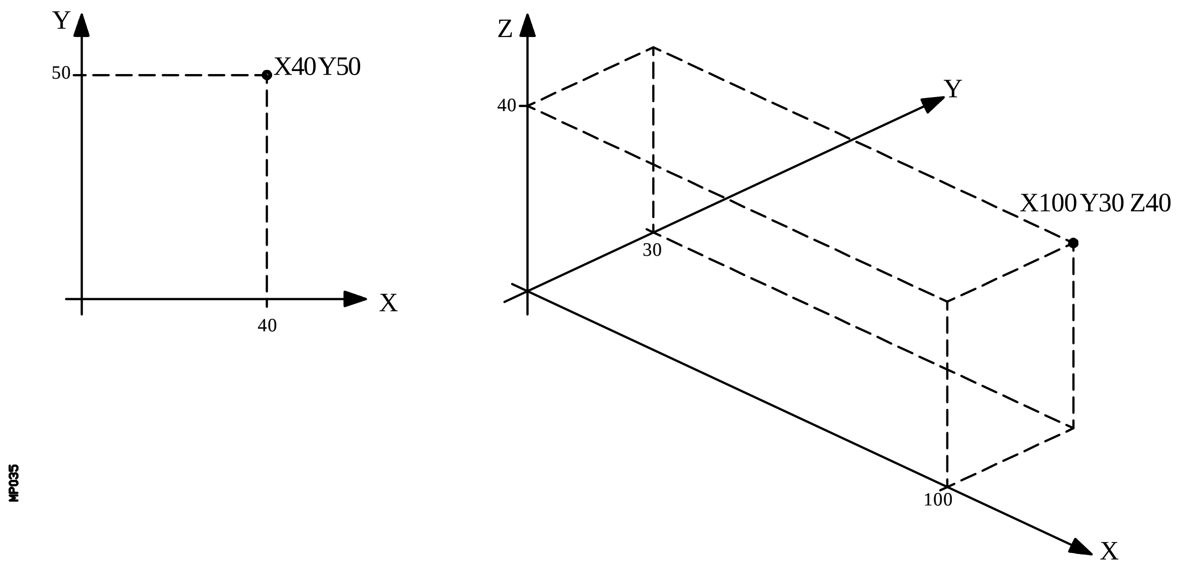
The Cartesian Coordinate System is defined by two axes on the plane, and by three or more axes in space.

The origin of all these, which in the case of the axes X Y Z coincides with the point of intersection, is called **Cartesian Origin** or **Zero Point of the Coordinate System**.

The position of the different points of the machine is expressed in terms of the coordinates of the axes, with two, three, four, or five coordinates.

The coordinates of the axes are programmed via the letter of the axis (X,Y,Z,U,V,W,A,B,C, always in this order) followed by the coordinate value.

The values of the coordinates are absolute or incremental, depending on whether it is working in G90 or G91, and its programming format is +/- 5.5.

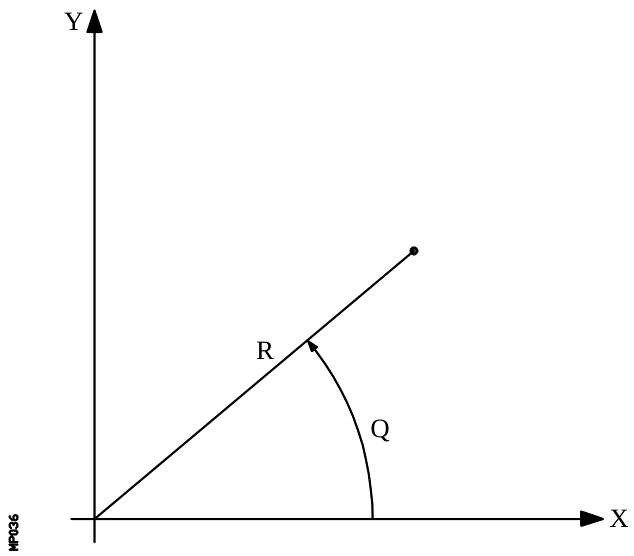


3.5.2 POLAR COORDINATES

In the event of the presence of circular elements or angular dimensions, the coordinates of the different points on the plane (2 axes at the same time), it may be easier to express them in polar coordinates.

The reference point is called **Polar Origin**, and this will be the origin of the **Polar Coordinate System**.

A point on this system would be defined by :



- The **RADIUS (R)**, the distance between the polar origin and the point.

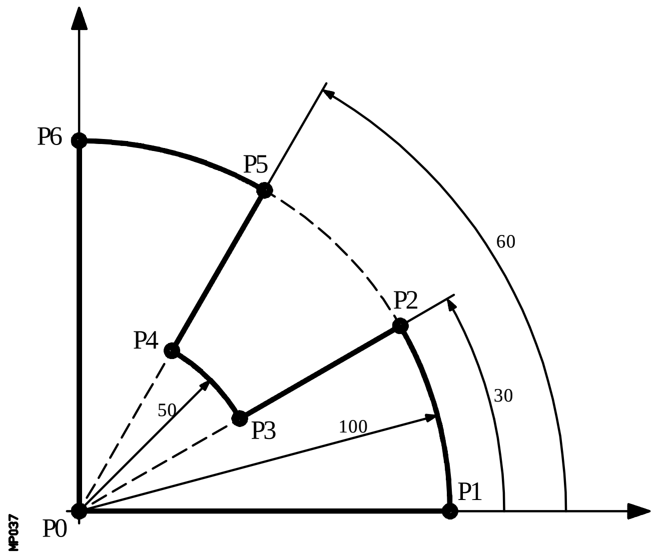
- The **ANGLE (Q)**, formed by the abscissa axis and the line which joins the polar origin with the point (in degrees).

The values **R** and **Q** are absolute or incremental depending on whether you are working with G90 or G91, and their programming format will be **R +/- 5.5 Q+/- 5.5**.

The **R** values **may be negative** when programming in incremental coordinates; but the **resulting** value assigned to the **radius must always be positive**.

If a **Q** value over 3600 is programmed, the module will be taken after dividing it by 360. Thus, Q420 is the same a Q60, and Q-240 is the same as Q-60.

Programming example assuming that the Polar Origin is located at the Coordinate Origin.



Absolute coordinates

```
G90       XO    YO  ; Point P0
     G01  R100  Q0  ; Point P1, in a straight line (G01)
     G03        Q30 ; Point P2, in an arc (G03)
     G01  R50   Q30 ; Point P3, in a straight line (G01)
     G03        Q60 ; Point P4, in an arc (G03)
     G01  R100  Q60 ; Point P5, in a straight line (G01)
     G03        Q90 ; Point P6, in an arc (G03)
     G01  R0    Q90 ; Point P0, in a straight line (G01)
```

Incremental coordinates

```
G90       X0    Y0  ; Point P0
G91  G01  R100  Q0  ; Point P1, in a straight line (G01)
     G03        Q30 ; Point P2, in an arc (G03)
     G01  R-50  Q0  ; Point P3, in a straight line (G01)
     G03        Q30 ; Point P4, in an arc (G03)
     G01  R50   Q0  ; Point P5, in a straight line (G01)
     G03        Q30 ; Point P6, in an arc (G03)
     G01  R-100 Q0  ; Point P0, in a straight line (G01)
```

The polar origin, apart from being able to be preset using function G93 (described later) can be modified in the following cases :

* On power-up, after executing M02, M30 EMERGENCY or RESET, the CNC will assume, as the polar origin, the coordinate origin of the work plane defined by the general machine parameter"IPLANE".

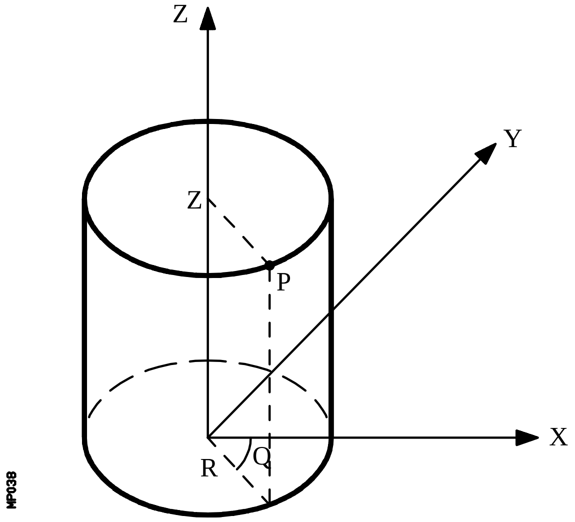
* Every time the work plane is changed (G16,G17,G18 or G19), the CNC assumes the coordinate origin of the new work plane selected as the polar origin.

* When executing a circular interpolation (G02 or G03), and if the general machine parameter "PORGMOVE" has a value of 1, the center of the arc will become the new polar origin.

3.5.3 CYLINDRICAL COORDINATES

To define a point in space, the system of cylindrical coordinates can be used as well as the Cartesian coordinate system.

A point in this system would be defined by :



* The projection of this point on the main plane, which should be defined in polar coordinates (R Q).

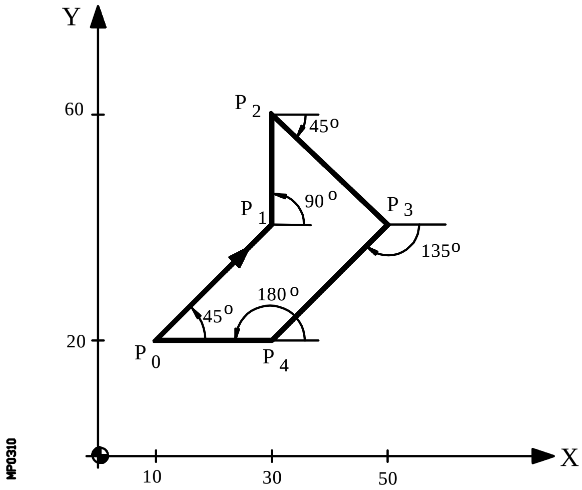
* Rest of axes in cartesian coordinates.

Examples : R30 Q10 Z100 R20 Q45 Z10 V30 A20

3.5.4 ANGLE AND ONE CARTESIAN COORDINATE

A point on the main plane can be defined via one of its cartesian coordinates, and the exit angle of the previous path.

Example of programming assuming that the main plane is XY:



X10	Y20	;	Point P0, starting point
Q45	X30	;	Point P1
Q90	Y60	;	Point P2
Q-45	X50	;	Point P3
Q-135	Y20	;	Point P4
Q180	X10	;	Point P0

If you wish to represent a point in space, the remaining coordinates can be programmed in cartesian coordinates.

3.6 ROTARY AXES

The types of rotary axes available are:

Normal rotary axis.
Positioning-only rotary axis.
Hirth rotary axis.

Each one of them can be divided into:

Rollover	When their position reading goes from 0º to 360º.
No rollover	When their position reading goes from -99999º to 99999º

They are all programmed in degrees. Therefore, their readings are not affected by the inch/mm conversion.

Normal rotary axes

They can be interpolated with linear axes.
Movement: in G00 and G01
Rollover axis programming:

G90 The sign indicates the turning direction and the target position (between 0 and 359.9999).

G91 The sign indicates the turning direction. If the programmed movement exceeds 360º, the axis will rotate more than one turn before positioning at the desired point.

Non-rollover axis programming: In G90 and G91 like a linear axis.

Positioning-only Axes

They cannot be interpolated with linear axes.
Movement: Always in G00 and they do not admit tool radius compensation (G41, G42).
Rollover axis programming:

G90 Always positive and via the shortest path. End coordinate between 0 & 359.9999

G91 The sign indicates the turning direction. If the programmed movement exceeds 360º, the axis will rotate more than one turn before positioning at the desired point.

Non-rollover axis programming: In G90 and G91 like a linear axis.

HIRTH axes

They work like the positioning-only axis except that they do not admit decimal position values (coordinates).

More than one hirth axis can be used, but they can only be moved one at a time.

3.7 WORK ZONES

The CNC provides four work zones or areas, and also limits the tool movement in each of these.

3.7.1 DEFINITION OF THE WORK ZONES

Within each work zone, the CNC allows you to limit the movement of the tool on each axis, with upper and lower limits being defined in each axis.

G20: Defines the lower limits in the desired zone.
G21: Defines the upper limits in the desired zone.

The format to program these functions is:

G20 K X...C +/- 5.5
G20 K X...C +/- 5.5

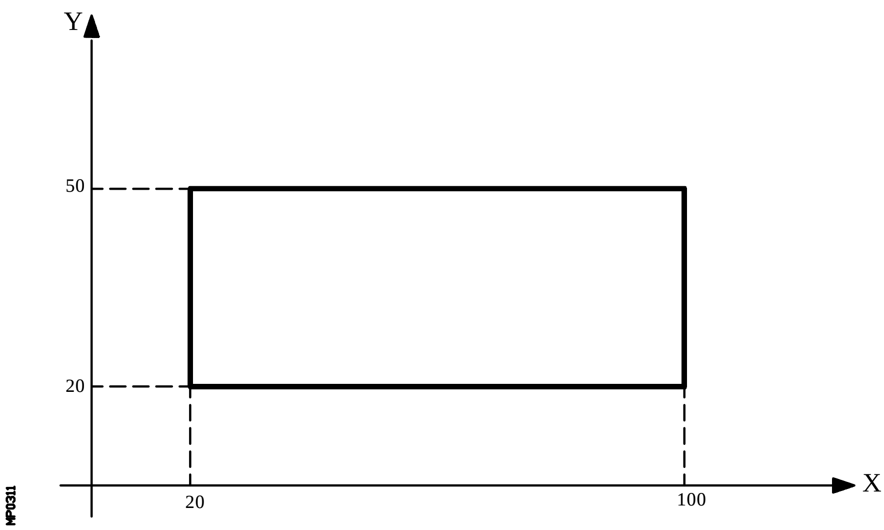
In which :

* **K** Indicates the work zone you wish to define (1, 2, 3 or 4)

* **X...C** Indicates the coordinates (upper or lower) with which you wish to limit the axes. These coordinates will be programmed with reference to machine zero (home).

It is not necessary to program all the axes, so only defined axes will be limited.

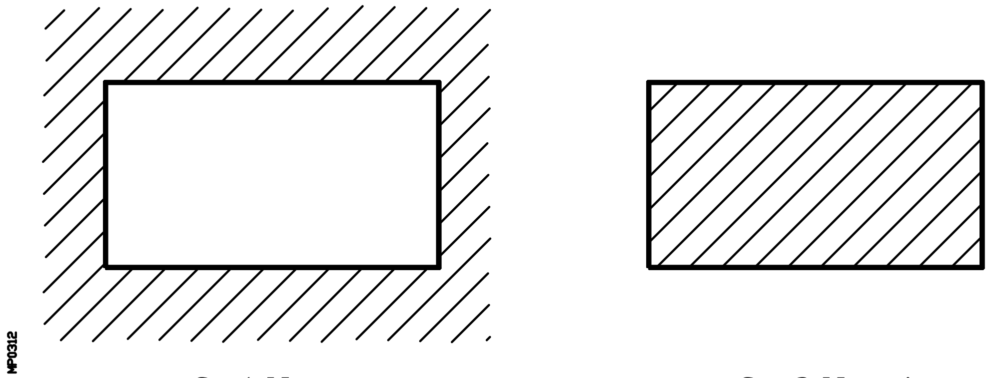
Example:



G20 K1 X20 Y20
G21 K1 X100 Y50

3.7.2 *USING WORK ZONES*

Within each work zone, the CNC allows you to restrict the movement of the tool, either prohibiting its exit from the programmed zone (no exit zone) or its entry into the programmed zone (no entry zone).



S= 1 No entry zone

S = 2 No exit zone

The CNC will take the dimensions of the tool into account at all times (tool offset table) to avoid it exceeding the programmed limits.

The presetting of work zones is done via Function **G22**, the programming format being:

G22 K S

In which :

* **K** Indicates the work zone you wish to define (1, 2, 3 or 4)
* **S** Indicates the enabling/disabling of the work zone:
  - S=0 disabled.
  - S=1 enabled as a no-entry zone.
  - S=2 enabled as a no-exit zone.

On power-up, the CNC will disable all work zones. However, upper and lower limits for these zones will not undergo any variation, and they can be re-enabled through the G22 function.

4. REFERENCE SYSTEMS

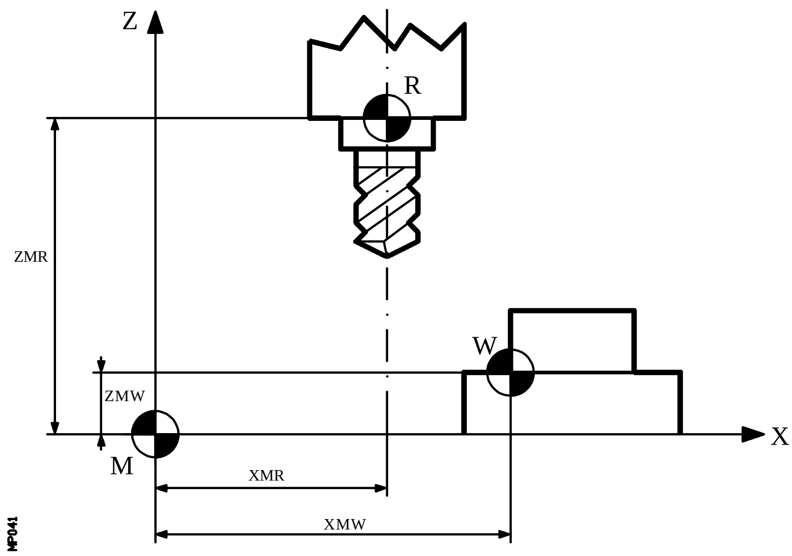
4.1 REFERENCE POINTS

A CNC machine needs the following origins and reference points defined :

* **Machine Reference Zero** or home. This is set by the manufacturer as the origin of the machine's coordinate system.

* **Part zero** or point of origin of the part. This is the point of origin which is set for programming the measurements of the part. It can be freely selected by the programmer, and its value with respect to machine zero can be set by the zero offset.

* **Machine Reference point.** This is a point on the machine established by the manufacturer around which the synchronization of the system is done. The control positions the axis on this point, instead of moving it as far as the Machine Reference Zero, taking, at this point, the reference coordinates which are defined via the axis machine parameter "REFVALUE".



M	Machine reference zero
W	Part zero
R	Machine reference point
XMW,YMW,ZMW, etc.	Coordinates of part zero
ZMR,YMR,ZMR, etc.	Coordinates of machine reference point ("REFVALUE")

4.2 MACHINE REFERENCE SEARCH (G74)

The CNC allows you to program the machine reference search in two ways :

* **MACHINE REFERENCE SEARCH OF ONE OR MORE AXES IN A PARTICULAR ORDER**

  G74 is programmed followed by the axes in which you want to carry out the reference search. For example : G74 X Z C Y

  The CNC begins the movement of all the selected axes which have a machine reference switch (machine axis parameter "DECINPUT") and in the direction indicated by the axis machine parameter "REFDIREC".

  This movement is carried out at the feedrate indicated by the axis machine parameter "REFEED1" for each axis until the home switch is hit.

  Next, the home search (marker pulse or home) will be carried out in the programmed order.

  This second movement will be carried out one axis at a time, at the feedrate indicated in the axis machine parameter "REFEED2" until the machine reference point is reached (i.e. the marker pulse is found).

* **MACHINE REFERENCE SEARCH USING THE ASSOCIATED SUBROUTINE.**

  The G74 function will be programmed alone in the block, and the CNC will automatically execute the subroutine whose number appears in the general machine parameter "REFPSUB". In this subroutine it is possible to program the machine reference searches required, and also in the required order.

In a block in which G74 has been programmed, no other preparatory function may appear.

If the machine reference search is done in JOG mode, the part zero selected is lost. The coordinates of the reference point indicated in the machine axis parameter "REFVALUE" is displayed. In all other cases, the part zero selected is maintained, so the displayed coordinates refer to this part zero.

If the G74 command is executed in MDI, the display of coordinates depends on the mode in which it is executed : Jog, Execution, or Simulation.

4.3 PROGRAMMING WITH RESPECT TO MACHINE ZERO (G53)

Function G53 can be added to any block which has path control functions.

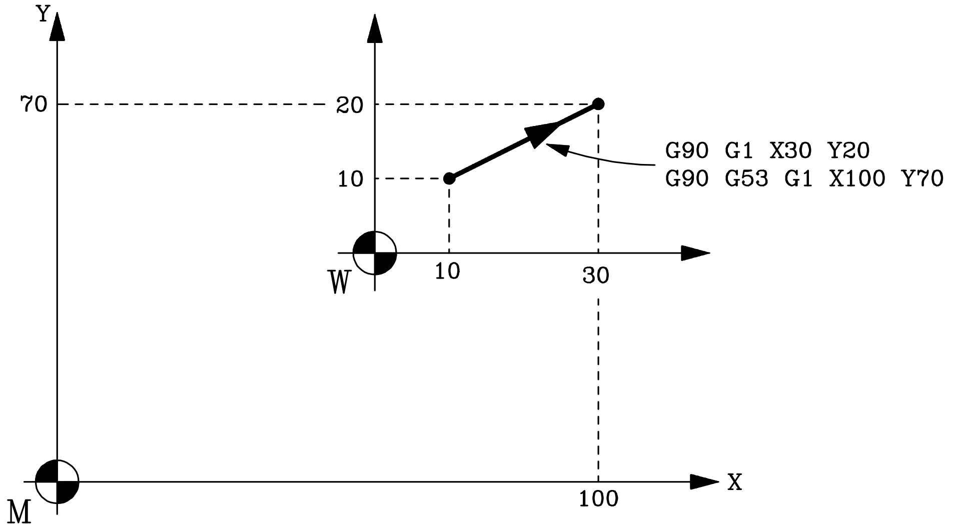
It is only used when the programming of block coordinates relating to machine zero is required. These coordinates should be expressed in millimeters or inches, depending on how the general machine parameter "INCHES" is defined.

By programming G53 alone (without motion information) the current active zero offset is canceled regardless of whether it was originated by a G54-G59 or a G92 preset. This G92 origin preset is described next. Once a Zero Offset has been selected, it will remain active until another one is selected or until a home search is carried out (G74). This Zero Offset will remain active even after powering the CNC off.

Function G53 is not modal, so it should be programmed every time you wish to indicate the coordinates referred to machine zero.

This function temporarily cancels radius and tool length compensation.

Example:

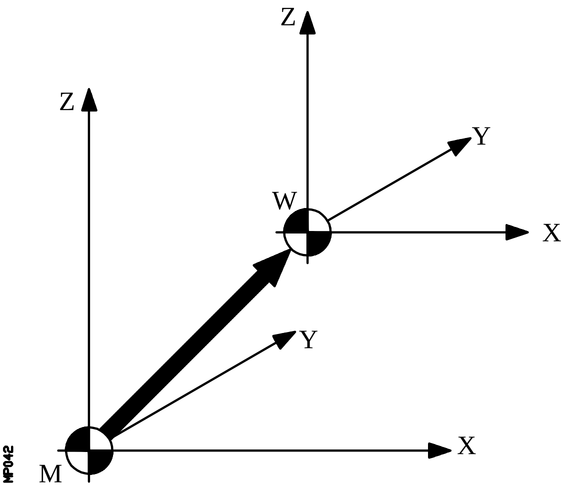


M Machine Reference Zero (home)
W Part Zero.

4.4 PRESETTING OF COORDINATES AND ZERO OFFSETS

The CNC allows you to carry out zero offsets with the aim of using coordinates related to the plane of the part, without having to modify the coordinates of the different points of the part at the time of programming.

The zero offset is defined as the distance between the part zero (point of origin of the part) and the machine zero (point of origin of the machine).



M Machine zero
W Part zero

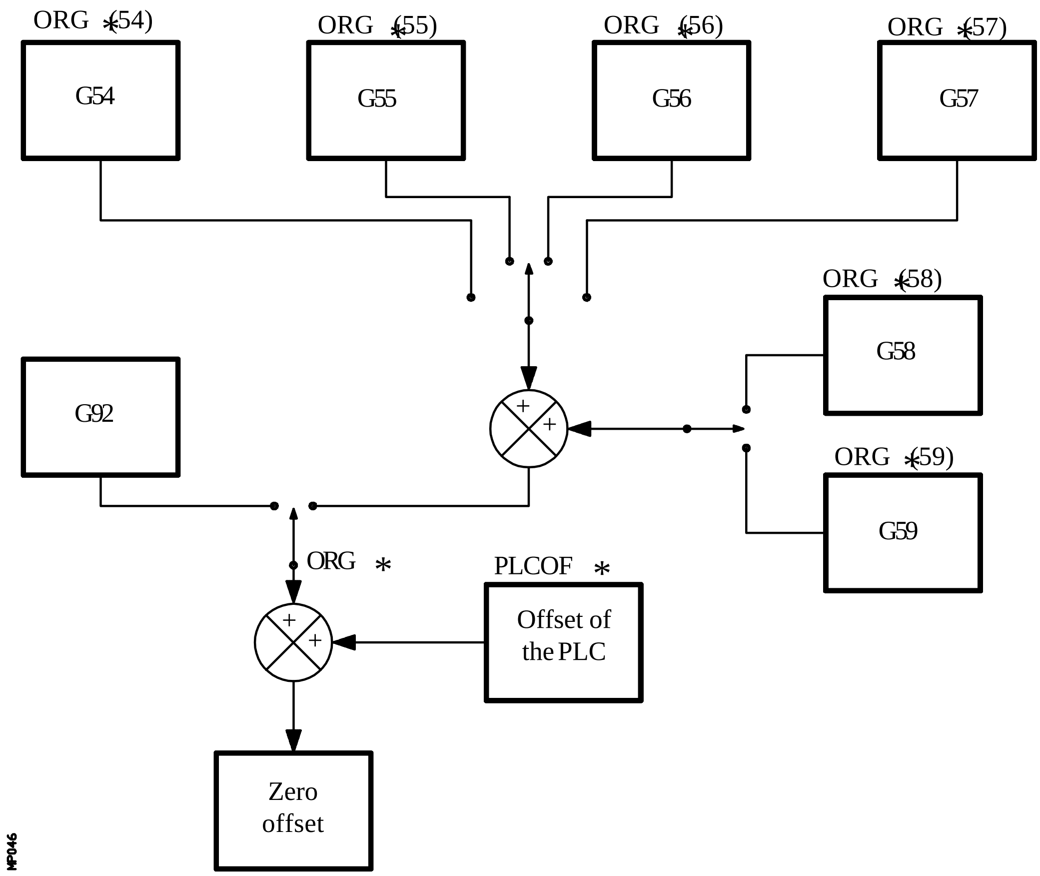
This zero offset can be carried out in one of two ways :

* Via Function G92 (coordinate preset). The CNC accepts the coordinates of the programmed axes after G92 as new axis values.

* Via the use of zero offsets (G54,G55,G56,G57,G58, G59). The CNC accepts as a new part zero the point located relative to machine zero at the distance indicated by the selected table(s).

Both functions are modal and incompatible, so if one is selected the other is disabled.

There is, moreover, another zero offset which is governed by the PLC. This offset is always added to the zero offset selected and is used (among other things) to correct deviations produced as a result of expansion, etc.


ORG *(54)
G54
ORG *(55)
G55
ORG *(56)
G56
ORG *(57)
G57
ORG *(58)
G58
G92
ORG *(59)
G59
ORG *
PLCOF *
Offset of the PLC
Zero offset
MP046

4.4.1 COORDINATE PRESET AND LIMITATION OF THE S VALUE (G92)

Via Function G92 one can select any value in the axes of the CNC, in addition to limiting the spindle speed.

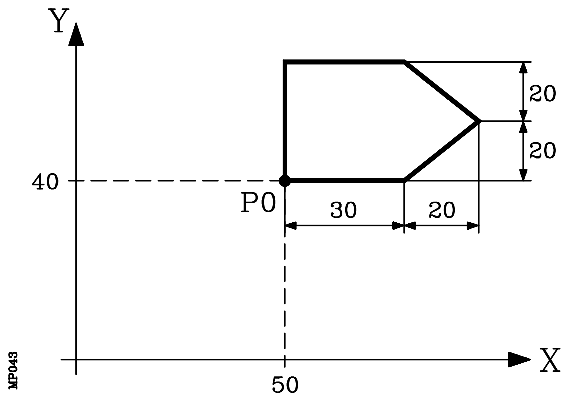
* COORDINATE PRESET

When carrying out a zero offset via Function G92, the CNC assumes the coordinates of the axes programmed after G92 as new axis values.

No other function can be programmed in the block where G92 is defined, the programming format being :

G92X...C +/- 5.5

Example :




```
G90  X50   Y40   ; Positioning in P0
G92  X0    Y0    ; Preset P0 as part zero
G91  X30         ; Programming according to part coordinates
     X20   Y20
     X-20  Y20
     X-30
           Y-40
```

* LIMITATION OF SPINDLE SPEED

When executing a "G92 S5.4" type block, the CNC limits the spindle speed from that instant on to the value set by S5.4.

If later on, a block is to be executed at a greater "S", the CNC will execute that block at the maximum "S" set with function G92S

Neither is it possible to exceed this maximum value from the keyboard on the front panel.

4.4.2 ZERO OFFSETS (G54..G59)

The CNC has a table of zero offsets, in which several zero offsets can be selected. The aim is to generate certain part zeros independently of the part zero active at the time.

Access to the table can be obtained from the front panel of the CNC (as explained in the Operating Manual), or via the program using high-level language commands.

There are two kinds of zero offsets :

Absolute zero offsets (G54,G55,G56 & G57), which must be referred to machine zero.

Additive zero offsets (G58,G59).

Functions G54, G55, G56, G57, G58 & G59 must be programmed alone in the block, and work in the following way:

When one of the G54, G55, G56, G57 functions is executed, the CNC applies the zero offset programmed with respect to machine zero, cancelling the possible active zero offsets.

If one of the additive offsets G58 or G59 is executed, the CNC adds its values to the absolute zero offset active at the time. Previously cancelling the additive offset which might be active.

You can see (in the following example) the zero offsets which are applied when the program is executed.

G54 Applies zero offsets G54 -------------------- > G54
G58 Adds zero offsets G58 ----------------------- > G54+G58
G59 Cancels G58 and adds G59 ----------------- > G54+G59
G55 Cancels whatever and applies G55--------- > G55

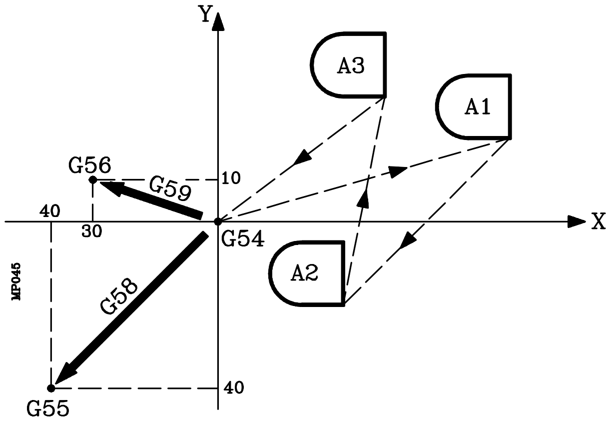
Once a Zero Offset has been selected, it will remain active until another one is selected or until a home search is carried out (G74) in JOG mode. This Zero Offset will remain active even after powering the CNC off.

This kind of zero offsets established by program is very useful for repeated machining operations at different machine positions.

Example :

The zero offset table is initialized with the following values:

G54:	X200	Y100			
G55:	X160	Y 60	G58:	X-40	Y-40
G56:	X170	Y110	G59:	X-30	Y 10



Using absolute zero offsets:

G54	; Applies G54 offset
Profile execution	; Executes profile A1
G55	; Applies G55 offset
Profile execution	; Executes profile A2
G56	; Applies G56 offset
Profile execution	; Executes profile A3

Using incremental zero offsets:

G54	; Applies G54 offset
Profile execution	; Executes profile A1
G58	; Applies offsets G54+G58
Profile execution	; Executes profile A2
G59	; Applies offsets G54+G59
Profile execution	; Executes profile A3

4.5 POLAR ORIGIN PRESET (G93)

Function G93 allows you to preset any point from the work plane as a new origin of polar coordinates.

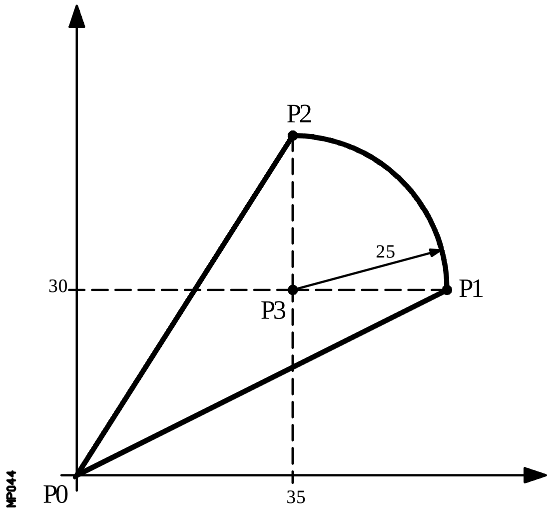
This function must be programmed alone in the block, its format being :

G93 I+/-5.5 J+/-5.5

Parameters I & J respectively define the abscissa and ordinate axes, of the new origin of polar coordinates.

Example :

Assuming that the tool is at X0 Y0




```
G93         I35  J30   ;  Preset P3 as polar origin
G90 G01  R25 Q0    ;  Point P1, in a straight line (G01)
    G03          Q90 ;  Point P2, in an arc (G03)
    G01  X0   Y0    ;  Point P0, in a straight line (G01)
```

If G93 is only programmed in a block, the point where the machine is at that moment becomes the polar origin.

Warning:

The CNC does not modify the polar origin when defining a new part zero; but it modifies the values of the variables: "PORGF" y "PORGS".

If, while selecting the general machine parameter “PORGMOVE” a circular interpolation is programmed (G02 or G03), the CNC assumes the center of the arc as the new polar origin.

On power-up; or after executing M02, M30; or after an EMERGENCY or RESET; the CNC assumes the currently active part zero as polar origin.

When selecting a new work plane (G16, G17, G18, G19), the CNC assumes as polar origin the part zero of that plane.

5. *PROGRAMMING BY ISO CODE*

A programmed block in ISO language can consist of :

Preparatory functions **(G)**
Axis coordinates **(X...C)**
Feedrate **(F)**
Spindle speed **(S)**
Tool number **(T)**
Tool offset number **(D)**
Auxiliary functions **(M)**

This order should be maintained within each block, although it is not necessary for every block to contain the information.

The CNC allows you to program figures from 0.00001 to 99999.9999 with or without sign, working in millimeters (G71), called format +/-5.4, or either from 0.00001 to 3937.00787 with or without sign if the programming is done in inches (G70), called format +/-4.5.

Nevertheless, and in order to simplify explanations, we can say that the CNC admits Format +/-5.5, meaning that it admits +/-5.4 in millimeters and +/-4.5 in inches.

Any function with parameters can also be programmed in a block, apart from the number of the label or block. Thus, when the block is executed the CNC substitutes the arithmetic parameter for its value at that time.

5.1 PREPARATORY FUNCTIONS

Preparatory functions are programmed using the letter G followed by 2 digits.

They are always programmed at the beginning of the body of the block and are useful in determining the geometry and working condition of the CNC.

Table of G functions used in the CNC :

Function	M	D	V	Meaning	Section
G00	*	?	*	Rapid travel	6.1
G01	*	?	*	Linear interpolation	6.2
G02	*		*	Clockwise (helical) circular interpolation	6.3
G03	*		*	Counter-clockwise (helical) circular interpolation	6.3
G04				Dwell/block preparation stop	7.1, 7.2
G05	*	?	*	Round corner	7.3.1
G06			*	Absolute arc center coordinates	6.4
G07	*	?		Square corner	7.3.2
G08			*	Arc tangent to previous path	6.5
G09			*	Arc defined by three points	6.6
G10	*	*		Mirror image cancellation	7.5
G11	*		*	Mirror image on X axis	7.5
G12	*		*	Mirror image on Y axis	7.5
G13	*		*	Mirror image on Z axis	7.5
G14	*		*	Mirror image in the programmed directions	7.5
G15	*		*	Longitudinal axis selection	8.2
G16	*		*	Selection of main plane in two directions	3.2
G17	*	?	*	Main plane X-Y and longitudinal Z	3.2
G18	*	?	*	Main plane Z-X and longitudinal Y	3.2
G19	*		*	Main plane Y-Z and longitudinal X	3.2
G20				Definition of lower work zone limits	3.7.1
G21				Definition of upper work zone limits	3.7.1
G22			*	Activate/cancel work zones	3.7.2
G28	*		*	Second spindle selection	5..4
G29	*	*		Main spindle selection	5..4
G28-G29			*	Axis toggle	7.9
G30	*		*	Spindle synchronization (offset)	5.5
G32	*		*	Feedrate as an inverted function of time.	6.15
G33	*		*	Threadcutting	6.12
G34				Variable pitch threading	6.13
G36			*	Automatic radius blend	6.10
G37			*	Tangential entry	6.8
G38			*	Tangential exit	6.9
G39			*	Automatic chamfer blend	6.11
G40	*	*		Cancellation of tool radius compensation	8.1
G41	*		*	Right-hand tool radius compensation	8.1
G41N	*		*	Collision detection	8.3
G42	*		*	Left-hand tool radius compensation	8.1
G42N	*		*	Collision detection	8.3
G43	*	?	*	Tool length compensation	8.2
G44	*	?	*	Cancellation of tool length compensation	8.2
G45	*		*	Tangential control	6.16
G50	*		*	Controlled corner rounding	7.3.3
G51	*		*	Look-Ahead	7.4
G52			*	Movement until making contact	6.14
G53			*	Program coordinates with respect to home	4.3

Function	M	D	V	Meaning	Section
G54	*		*	Absolute zero offset 1	4.4.2
G55	*		*	Absolute zero offset 2	4.4.2
G56	*		*	Absolute zero offset 3	4.4.2
G57	*		*	Absolute zero offset 4	4.4.2
G58	*		*	Additive zero offset 1	4.4.2
G59	*		*	Additive zero offset 2	4.4.2
G60			*	Straight line canned cycle	10.1
G61			*	Rectangular pattern canned cycle	10.2
G62			*	Grid pattern canned cycle	10.3
G63			*	Circular pattern canned cycle	10.4
G64			*	Arc pattern canned cycle	10.5
G65			*	Arc-chord pattern canned cycle	10.6
G66			*	Irregular pocket canned cycle	11.1
G67			*	Irregular pocket roughing	11.1.2
G68			*	Irregular pocket finishing	11.1.3
G69	*		*	Complex deep hole drilling	9.5.1
G70	*	?	*	Programming in inches	3.3
G71	*	?		programming in millimeters	3.3
G72	*		*	General and specific scaling factor	7.6
G73	*		*	Pattern rotation	7.7
G74			*	Machine reference search	4.2
G75			*	Probing until touching	12.1
G76			*	Probing while touching	12.1
G77	*		*	Slaved axis	7.8.1
G77S	*		*	Spindle synchronization	5.5
G78	*	*		Slaved axis cancellation	7.8.2
G78S	*	*		Cancellation of spindle synchronization	5.5
G79				Canned cycle parameter modification	9.2.1
G80	*	*		Canned cycle cancellation	9.3
G81	*		*	Drilling cycle	9.5.2
G82	*		*	Drilling cycle with dwell	9.5.3
G83	*		*	Simple deep hole drilling	9.5.4
G84	*		*	Tapping cycle	9.5.5
G85	*		*	Reaming cycle	9.5.6
G86	*		*	Boring cycle with withdrawal in G00	9.5.7
G87	*		*	Rectangular pocket milling cycle	9.5.8
G88	*		*	Circular pocket milling cycle	9.5.9
G89	*		*	Boring cycle with withdrawal in G01	9.5.10
G90	*	?		Programming in absolute	3.4
G91	*	?	*	Programming in incremental	3.4
G92				Coordinate preset/spindle speed limit	4.4.1
G93				Polar origin preset	4.5
G94	*	?		Feedrate in millimeters(inches) per minute	5.2.1
G95	*	?	*	Feedrate in millimeters(inches) per revolution	5.2.2
G96	*		*	Constant cutting point speed	5.2.3
G97	*	*		Constant tool center speed	5.2.4
G98	*	*		Withdrawal to the starting plane	9.5
G99	*		*	Withdrawal to the reference plane	9.5

M means modal, i.e. the G function, once programmed, remains active while another incompatible G function is not programmed.

D means BY DEFAULT, i.e. they will be assumed by the CNC when it is powered on, after executing M02, M30 or after EMERGENCY or RESET.

In those cases indicated by **?** , it should be understood that the DEFAULT of these G functions depends on the setting of the general machine parameters of the CNC.

V means that the G code is displayed next to the current machining conditions in the execution and simulation modes.

5.2 FEEDRATE F

The machining feedrate can be selected from the program. It remains active until another feedrate is programmed.

It is represented by the letter **F**. Depending on whether it is working in G94 or G95, it is programmed in mm/minute (inches/minute) or in mm/revolution (inches/revolution).

It's programming format is 5.5 in mm. and 4.5 in inches.

The maximum operating feedrate of the machine, limited on each axis by the axis machine parameter "MAXFEED", may be programmed via code F0, or by giving F the corresponding value.

The programmed feedrate F is effective working in linear (G01) or circular (G02, G03) interpolation. If function F is not programmed, the CNC assumes the feedrate to be F0. When working in rapid travel (G00), the machine will move at the rapid feedrate indicated by the axis machine parameter "G00FEED", apart from the F programmed.

The programmed feedrate F may be varied between 0% and 255% via the PLC, or by DNC, or between 0% and 120% via the switch located on the Operator Panel of the CNC.

The CNC, however, is equipped with the general machine parameter "MAXFOVR" to limit maximum feedrate variation.

If you are working in rapid travel (G00), rapid feedrate will be fixed at 100%, alternatively it can be varied between 0% and 100%, depending on how the machine parameter "RAPIDOVR" is set.

When functions G33 (electronic threading), G34 (variable pitch threading) or G84 (tapping canned cycle) are executed the feedrate cannot be modified. It functions at 100% of programmed F.

5.2.1 FEEDRATE IN MM/MIN OR INCHES/MIN (G94)

From the moment the code G94 is programmed, the control takes that the feedrates programmed through **F5.5** are in mm/min or inches/mm.

If the movement corresponds to a rotary axis, the CNC interprets the feedrate as being programmed in degrees/min.

If an interpolation is made between a rotary and a linear axis, the programmed feedrate is taken in mm/min or inches/min, and the movement of the rotary axis (programmed in degrees) will be considered programmed in millimeters or inches.

The relationship between the feedrate of the axis component and the programmed feedrate "F" is the same as that between the movement of the axis and the resulting programmed movement.

$$\text{Feedrate component} = \frac{\text{Feedrate F x Movement of axis}}{\text{Resulting programmed movement}}$$

Example :

On a machine which has linear X and Y axes and rotary C axis, all located at point X0 Y0 C0, the following movement is programmed :

G1 G90 X100 Y20 C270 F10000

You get:

$$\frac{F \triangle x}{\sqrt{(\triangle x)^2 + (\triangle y)^2 + (\triangle c)^2}} \qquad \frac{10000 \times 100}{\sqrt{100^2 + 20^2 + 270^2}} = 3464.7946$$

$$\frac{F \triangle y}{\sqrt{(\triangle x)^2 + (\triangle y)^2 + (\triangle c)^2}} \qquad \frac{10000 \times 20}{\sqrt{100^2 + 20^2 + 270^2}} = 692.9589$$

$$\frac{F \triangle c}{\sqrt{(\triangle x)^2 + (\triangle y)^2 + (\triangle c)^2}} \qquad \frac{10000 \times 270}{\sqrt{100^2 + 20^2 + 270^2}} = 9354.9455$$

Function G94 is modal i.e. once programmed it stays active until G95 is programmed.

On power-up, after executing M02, M30 or following EMERGENCY or RESET, the CNC assumes function G94 or G95 according to how the general machine parameter “IFEED” is set.

5.2.2 FEEDRATE IN MM/REV.OR INCHES/REV (G95)

From the moment when the code G95 is programmed, the control assumes that the feedrates programmed through **F5.5** are in mm/rev or inches/mm.

This function does not affect the rapid moves (G00) which will be made in mm/min or inch/min. By the same token, it will not be applied to moves made in the JOG mode, during tool inspection, etc.

Function G95 is modal i.e. once programmed it stays active until G94 is programmed.

On power-up, after executing M02, M30 or following EMERGENCY or RESET, the CNC assumes function G94 or G95 according to the general machine parameter “IFEED”.

5.2.3 CONSTANT SURFACE SPEED (G96)

When G96 is programmed the CNC takes the **F5.5** feedrate as corresponding to the cutting point of the tool on the part.

By using this function, the finished surface is uniform in curved sections.

In this manner (working in function G96) the speed of the center of the tool in the inside or outside curved sections will change in order to keep the cutting point constant.

Function G96 is modal i.e. once programmed, it is active until G97 is programmed.

On power-up, after executing M02, M30 or following EMERGENCY or RESET, the CNC assumes function G97.

5.2.4 CONSTANT TOOL-CENTER SPEED (G97)

When G97 is programmed the CNC takes the programmed **F5.5** feedrate as corresponding to the feedrate of the center of the tool.

In this manner (working in function G97) the speed of the cutting point on the inside or outside curved sections is reduced, keeping the speed of the center of the tool constant.

Function G97 is modal i.e. once programmed it is active until G96 is programmed.

On power-up, after executing M02, M30 or following EMERGENCY or RESET, the CNC assumes function G97.

5.3 SPINDLE SPEED (S)

The turning speed of the spindle is programmed directly in rpm via code S5.4.

The maximum value is limited by spindle machine parameters “MAXGEAR1”, MAXGEAR2, MAXGEAR 3 and MAXGEAR4", in each case depending on the spindle range selected.

It is also possible to limit this maximum value from the program by using function G92 S5.4.

The programmed turning speed S may be varied from the PLC, DNC, or by the SPINDLE keys “+” and “-” on the Operator Panel of the CNC.

This speed variation is made between the maximum and minimum values established by spindle machine parameters “MINSOVR and MAXSOVR”.

The incremental pitch associated with the SPINDLE keys “+” and “-” on the CNC Operator Panel in order to vary the programmed S value is fixed by the spindle machine parameter “SOVRSTEP”.

When functions G33 (threading), G34 (variable pitch threading) or G84 (tapping cycle) are executed the speed cannot be modified. It functions at 100% of programmed S.

5.4 SPINDLE SELECTION (G28, G29)

This CNC can govern two spindles: the main one and the second one. Both can be running at the same time but it can only control one at the time.

This selection is done by functions: G28 and G29.

G28: Selects the Second Spindle
G29: Selects the Main Spindle.

Once the desired spindle has been selected, it can be acted upon from the keyboard or by means of the following functions:

M3, M4, M5, M19
S****
G33, G34, G94, G95, G96, G97

Both spindles can work in open and closed loop.

Functions G28 and G29 are modal and incompatible with each other.

Function G28 and G29 must be programmed alone in the block. No more information can be programmed in that block.

On power-up, after executing and M02, M30 or after an EMERGENCY or RESET, the CNC assumes function G29 (selects the main spindle).

Operating example for when 2 spindles are used:

On power-up, the CNC assumes function G29 selecting the main spindle.
All the actions upon the keys or functions associated with the spindle will be applied on to the main spindle.
Example: S1000 M3 Main spindle clockwise at 1000 rpm

To select the second spindle, execute function G28.
From this moment on, all the actions upon the keys or functions associated with the spindle will be applied on to the second spindle.
The main spindle keeps turning (in its previous status).
Example: S1500 M4 Second spindle counter-clockwise at 1500 rpm.
The main spindle keeps turning clockwise at 1000 rpm

To select the main spindle again, execute function G29.

From this moment on, all the actions upon the keys or functions associated with the spindle will be applied on to the main spindle.
The second spindle keeps turning (in its previous status).
Example: S2000 The main spindle keeps turning clockwise but now at 2000 rpm.
The second spindle keeps turning counterclockwise at 1500 rpm.

5.5 SYNCHRONIZED SPINDLES (G30, G77S, G78S)

With function G77S, two spindles (main and secondary) may be synchronized in speed. This synchronism may be cancelled with function G78S.

Always program G77S and G78S because functions G77, G78 to slave and unslave the axes.

When the spindles are synchronized in speed, the second one turns at the same speed as the main spindle.

Function G77S may be executed at any time, open loop (M3, M4) or closed loop (M19), the spindles may even have different ranges (gears).

General output "SYNSPEED (M5560)" will be high while the spindle are in synch (same speed).

When this synchronism is cancelled (G78S), the second spindle recovers its previous speed and status (M3, M4, M5, M19) and the main spindle stays in the current status.

If while in synchronism, an S is programmed greater than the maximum allowed, the CNC applies the maximum value while they are synchronized. When cancelling this synchronism, the limit is no longer applied and the main spindle assumes the programmed speed.

While the spindles are synchronized in speed, function G77S active, with G30 they may also be synchronized in position and set an angular offset between them so the second spindle follows the main spindle at this set offset distance.

Programming format: G30 D ±359.9999 (offset in degrees)

For example, with G30 D90 the second spindle will turn 90º behind the main spindle.

<u>Considerations:</u>

Before activating the synchronism, both spindles must be homed (referenced).
To synchronized the spindles in position (G30) they must be synchronized in speed already (G77S).

While the spindles are synchronized, only the signals of the main spindle will be attended to PLCCNTL, SPDLINH, SPDLREV, etc. On the other hand, when making a thread, only the feedback and reference pulses of the main spindle will be taken into account.

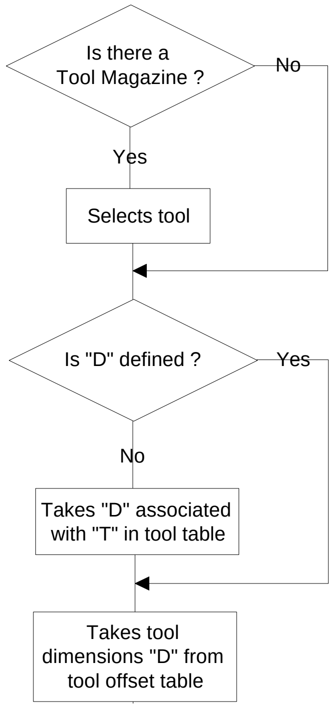
While the spindle synchronism is active, it is possible to:
Execute functions G94, G95, G96, G97, M3, M4, M5, M19 S***
Change the spindle speed via DNC, PLC or CNC (S)
Change the spindle speed override via DNC, PLC, CNC or keyboard
Change the spindle speed limit via DNC, PLC or CNC (G92 S)

But, it is **NOT** possible to:
Toggle the spindles: G28, G29
Change gears: M41, M42, M43, M44.

5.6 *TOOL NUMBER (T) AND TOOL OFFSET (D)*

With the "T" function, it is possible to select the tool and with the "D" function it is possible to select the offset associated with it. When defining both parameters, the programming order is T D. For example: T6 D17

If the machine has a tool magazine, the CNC looks up the "Tool magazine table" to know the position occupied by the selected tool and the desired one.



If the "D" function has not be defined, it looks up the "Tool table" to know the "D" offset associated with it.

It looks up the "tool offsets table" and assumes the tool dimensions corresponding to the "D" offset.

It looks up the "Tool geometry table" to know the cutter geometry (width, angle, cutting angle).

The "Tool geometry table is associated with the T or the D according to the machine manufacturer's criteria, general machine parameter "GEOMTYPE (P123)"

To access, check and define these tables, refer to chapter 6 of the Operating Manual.

<u>How to use the T and D functions</u>

- The "T" and "D" functions may be programmed alone or together as shown in the following example:

T5 D18	selects tool 5 and assumes the dimensions of tool offset 18
D22	Tool 5 stays selected and it assumes the dimensions of tool offset 22
T3	selects tool 3 and assumes the dimensions of the offset associated with that tool.

- When having a tool turret, it is rather common to use more tools than the number of tool positions of the turret. Thus, the same turret position must be used by more than one tool.

  In those cases, both "T" and "D" must be programmed.

  The "T" function refers to the turret position and the "D" function to the dimensions of the tool located in that position.

  Thus, for example, programming T5 D23 means selecting the turret position 5 and assuming the geometry and dimensions of tool offset 23.

- When having a tool holding arm with 2 cutters, both "T" and "D" must be programmed as well.

  The "T" function refers to the arm and the "D" function to the cutter dimensions.

  Thus, one may program T1 D1 or T1 D2 depending on which of the 2 cutters is to be used.

Tool length and radius compensation.

The CNC looks up the "tool offset table" and assumes the tool dimensions corresponding to the active "D" offset.

Length compensation is applied at all times, whereas radius compensation must be selected by the operator by means of functions G40, G41, G42.

If there is no tool selected or D0 is defined, neither tool length nor radius compensation is applied.

For further information, refer to chapter 8 "tool compensation" in this manual..

5.7 *MISCELLANEOUS FUNCTION (M)*

The miscellaneous functions are programmed by means of the **M4** Code, it being possible to program up to 7 functions in the same block.

When more than one function has been programmed in one block, the CNC executes these correlatively to the order in which they have been programmed.

The CNC is provided with an M functions table with “NMISCFUN” (general machine parameter) components, specifying for each element:

* The number (0-9999) of the defined miscellaneous M function.

* The number of the subroutine which is required to associate to this miscellaneous function.

* An indicator which determines if the M function is executed before or after the movement block in which it is programmed.

* An indicator which determines if the execution of the M function interrupts block preparation or not.

* An indicator which determines if the M function is executed or not, after the execution of the associated subroutine.

* An indicator which determines if the CNC must wait for the signal AUX END or not (Executed M signal, coming from the PLC), to continue the execution of the program.

If, when executing the M miscellaneous function, this is not defined in the M functions table, the programmed function will be executed at the beginning of the block and the CNC will wait for the AUX END to continue the execution of the program.

Some of the miscellaneous functions are assigned an internal meaning in the CNC.

If, while executing the associated subroutine of an “M” miscellaneous function, there is a block containing the same “M”, this will be executed but not the associated subroutine.

Warning:

All the miscellaneous “M” functions which have an associated subroutine must be programmed alone in a block.

5.7.1 M00. PROGRAM STOP

When the CNC reads code M00 in a block, it interrupts the program. To start up again, press CYCLE START.

We recommend that you set this function in the table of M functions, in such a way that it is executed at the end of the block in which it is programmed.

5.7.2 M01. CONDITIONAL PROGRAM STOP

This is identical to M00, except that the CNC only takes notice of it if the signal M01 STOP from the PLC is active (high logic level).

5.7.3 M02. END OF PROGRAM

This code indicates the end of program and carries out a "General Reset" function of the CNC (returning it to original state). It also carries out the M05 function.

It is recommended to set this function in the table of M functions, in such a way that it is executed at the end of the block in which it is programmed.

5.7.4 M30. END OF PROGRAM WITH RETURN TO FIRST BLOCK

Identical to M02 except that the CNC returns to the first block of the program.

5.7.5 M03. CLOCKWISE SPINDLE ROTATION

This code represents clockwise spindle start. As explained in the corresponding section, the CNC automatically executes this code in the machining canned cycles.

It is recommended to set this function in the table of M functions, so that it is executed at the beginning of the block in which it is programmed.

5.7.6 M04. COUNTER-CLOCKWISE SPINDLE ROTATION

This code represents counter-clockwise spindle start. We recommend that you set this function in the table of M functions, so that it is executed at the beginning of the block in which it is programmed.

5.7.7 M05. SPINDLE STOP

It is recommended to set this function in the table of M functions, so that it is executed at the end of the block in which it is programmed.

5.7.8 M06. TOOL CHANGE

If the general machine parameter “TOFFM06” (indicating that it is a machining center) is active, the CNC sends instructions to the tool changer and updates the table corresponding to the tool magazine.

It is recommended to set this function in the table of M functions, so that the subroutine corresponding to the tool changer installed in the machine is executed.

5.7.9 M19. SPINDLE ORIENTATION

With this CNC it is possible to work with the spindle in open loop (M3, M4) and with the spindle in closed loop (M19).

In order to work in closed loop, it is necessary to have a rotary encoder installed on the spindle of the machine.

To switch from open loop to closed loop, execute function M19 or M19 S±5.5. The CNC will act as follows:

* If the spindle does not have a home switch, the CNC changes the spindle speed until it reaches the one set by spindle machine parameter "REFEED2; finds the marker pulse (home) and, then, orients the spindle to the position defined by S±5.5.

* If the spindle has a home switch, the CNC modifies the spindle speed until it reaches the one set by spindle machine parameter "REFEED1". Then, it carries out the search for the home switch at this speed. Next, it looks for the marker pulse (home) at the speed set by spindle machine parameter "REFEED2" and, finally, it orients the spindle to the position defined by S±5.5.

If only M19 is executed, the spindle is oriented to position "S0" after having "found" the home switch.

To, now, orient the spindle to another position, program M19 S±5.5, the CNC will not perform the home search since it is already in closed loop and it will orient the spindle to the indicated position. (S±5.5).

The S±5.5 code indicates the spindle orient position, in degrees, from the encoder's marker pulse position (S0).

The sign indicates the counting direction and the 5.5 value is always considered to be absolute coordinates regardless of the type of units currently selected.

Example:

S1000 M3	Spindle in open loop
M19 S100	The spindle switches to closed loop. Home search and positioning (orientation) at 100º
M19 S-30	The spindle orients to -30º, passing through 0º.
M19 S400	The spindle turns a whole revolution and positions at 40º.

5.7.10 M41, M42, M43, M44. SPINDLE SPEED RANGE CHANGE

The CNC offers 4 spindle speed ranges M41, M42, M43 and M44 with maximum speed limits set by the spindle machine parameters “MAXGEAR1”, MAXGEAR2", “MAXGEAR3” and “MAXGEAR4”.

If machine parameter “AUTOGEAR” is set so the CNC executes the range change automatically, M41 thru M44 will be sent out automatically by the CNC without having to be programmed.

If this machine parameter is set for non-automatic gear change, M41 thru M44 will have to be programmed every time a gear change is required. Bear in mind that the maximum voltage value assigned to machine parameter “MAXVOLT” corresponds to the maximum speed indicated for each one of the speed ranges (machine parameters “MAXGEAR1” thru “MAXGEAR4”).

5.7.11 M45 AUXILIARY SPINDLE / LIVE TOOL

In order to use this miscellaneous function, it is necessary to set one of the axes of the machine as auxiliary spindle or live tool (general machine parameter P0 thru P7).

To use the auxiliary spindle or live tool, execute the command: **M45 S±5.5** where **S** indicates the turning speed in rpm and the sign indicates the turning direction.

The CNC will output the analog voltage corresponding to the selected speed according to the value assigned to the machine parameter "**MAXSPEED**" for the auxiliary spindle.

To stop the auxiliary spindle, program **M45** or **M45 S0**.

Whenever the auxiliary spindle or live tool is active, the CNC will let the PLC know by activating the general logic output "**DM45**" (M5548).

Also, it is possible to set the machine parameter for the auxiliary spindle "**SPDLOVR**" so the Override keys of the front panel can modify the currently active turning speed of the auxiliary spindle.

6. PATH CONTROL

The CNC allows you to program movements on one axis only or several at the same time.

Only those axes which intervene in the required movement are programmed. The programming order of the axes is as follows :

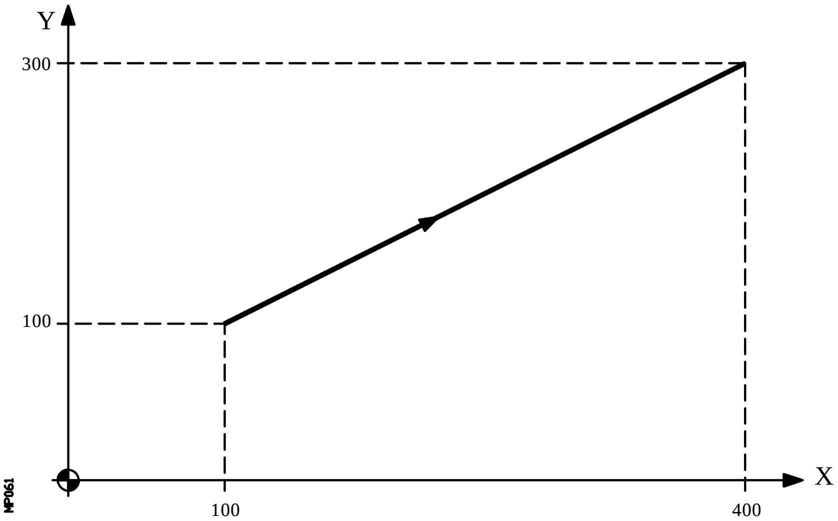
X, Y, Z, U, V, W, A, B, C

6.1 RAPID TRAVEL (G00)

The movements programmed after G00 are executed at the rapid feedrate indicated in the axis machine parameter "G00FEED".

Independently of the number of axis which move, the resulting path is always a straight line between the starting point and the final point.

Example:



X100 Y100 ; Starting point
G00 G90 X400 Y300 ; Programmed path

It is possible, via the general machine parameter "RAPIDOVR", to establish if the feedrate override % switch (when working in G00) operates from 0% to 100%, or whether it stays constant at 100%.

When G00 is programmed, the last "F" programmed is not cancelled i.e. when G01, G02 or G03 are programmed again "F" is recovered.

G00 is modal and incompatible with G01, G02, G03, G33, G34 and G75. Function G00 can be programmed as G or G0.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G00 or G01, depending on how general machine parameter "IMOVE" has been set.

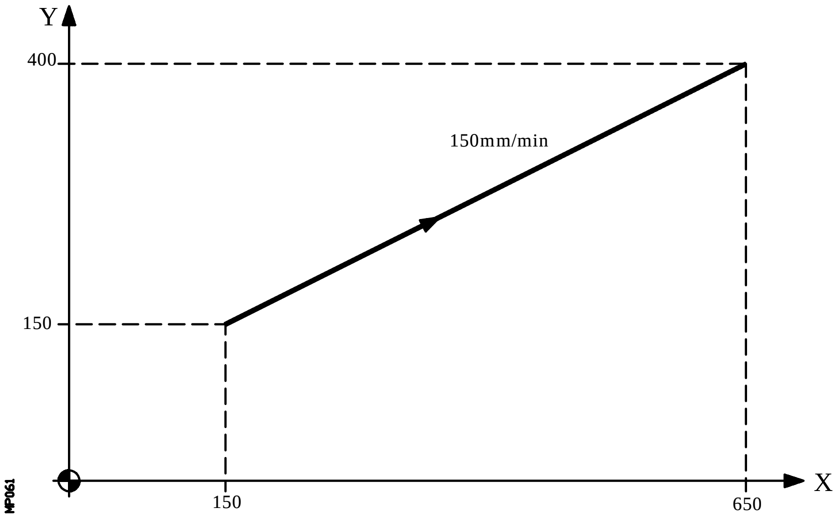
6.2 LINEAR INTERPOLATION (G01)

The movements programmed after G01 are executed according to a straight line and at the programmed feedrate "F".

When two or three axes move simultaneously the resulting path is a straight line between the starting point and the final point.

The machine moves according to this path to the programmed feedrate "F". The CNC calculates the feedrates of each axis so that the resulting path is the "F" value programmed.

Example:



G01 G90 X650 Y400 F150

The programmed feedrate "F" may vary between 0% and 120% via the switch located on the Control Panel of the CNC, or by selecting between 0% and 255% from the PLC, or via the DNC or the program.

Nevertheless, the CNC has general machine parameter "MAXFOVR" to limit maximum variation of the feedrate.

With this CNC, it is possible to program a positioning-only axis in a linear interpolation block. The CNC will calculate the feedrate for this positioning-only axis so it reaches the target coordinate at the same time as the interpolating axes.

Function G01 is modal and incompatible with G00, G02, G03, G33, G34 and G75. Function G01 can be programmed as G1.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G00 or G01, depending on how general machine parameter "IMOVE" has been set.

6.3 CIRCULAR INTERPOLATION (G02. G03)

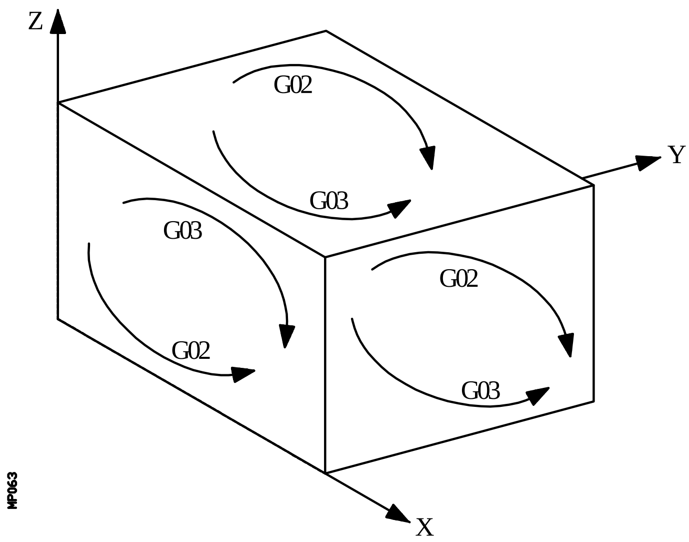
There are two ways of carrying out circular interpolation:

G02: Clockwise circular interpolation

G03: Counter-clockwise circular interpolation

Movements programmed after G02 and G03 are executed in the form of a circular path and at the programmed feedrate "F".

Clockwise (G02) and counterclockwise (G03) definitions are established according to the system of coordinates shown below:



This system of coordinates refers to the movement of the tool on the part.

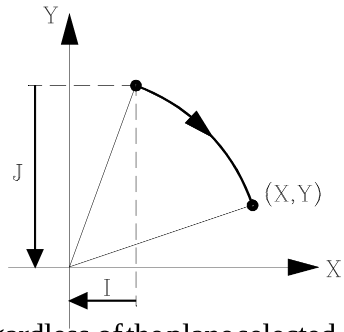
Circular interpolation can only be executed on a plane. The form of definition of circular interpolation is as follows :

a) CARTESIAN COORDINATES

The coordinates of the endpoint of the arc and the position of the center with respect to the starting point are defined according to the axes of the work plane.

The center coordinates (which should always be programmed even if they have "0" value) are defined by the letters I, J, or K, each one of these being associated to the axes as follows:

Axes X,U,A --> I
Axes Y,V,B —>J
Axes Z,W,C —>K



Programming format:

Plane XY: G02(G03) X±5.5 Y±5.5 I±5.5 J±5.5
Plane ZX: G02(G03) X±5.5 Z±5.5 I±5.5 K±5.5
Plane YZ: G02(G03) Y±5.5 Z±5.5 J±5.5 K±5.5

The programming order of the axes is always maintained regardless of the plane selected,, as are the respective center coordinates.

Plane AY: G02(G03) Y±5.5 A±5.5 J±5.5 I±5.5
Plane XU: G02(G03) X±5.5 U±5.5 I±5.5 I±5.5

b) POLAR COORDINATES

It is necessary to define the angle to be travelled **Q** and the distance from the starting point to the center (optional), according to the axes of the work plane.

The center coordinates are defined by the letters I, J, or K, each one of these being associated to the axes as follows:

Axes X,U,A —>I
Axes Y,V,B —>J
Axes Z,W,C —> K

If the center of the arc is not defined, the CNC will assume it that this coincides with the current polar origin.

Programming format:

Plane XY: G02(G03) Q±5.5 I±5.5 J±5.5
Plane ZX: G02(G03) Q±5.5 I±5.5 K±5.5
Plane YZ: G02(G03) Q±5.5 J±5.5 K±5.5

c) CARTESIAN COORDINATES WITH RADIUS PROGRAMMING

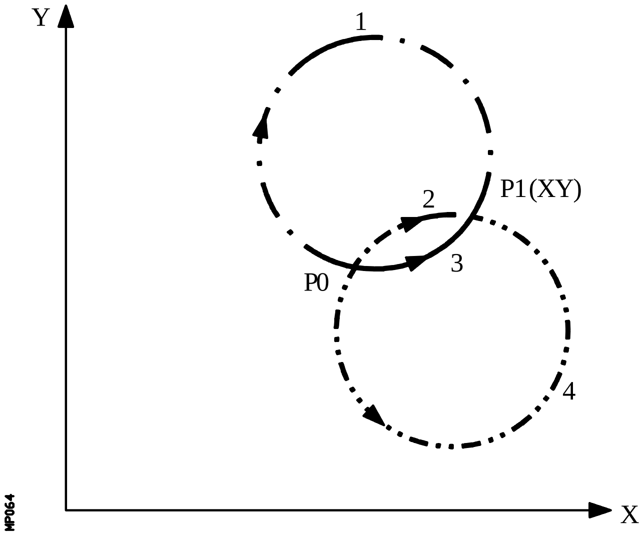
The coordinates of the endpoint of the arc and radius **R** are defined.

Programming format:

Plane XY: G02(G03) X±5.5 Y±5.5 R±5.5
Plane ZX: G02(G03) X±5.5 Z±5.5 R±5.5
Plane YZ: G02(G03) Y±5.5 Y±5.5 R±5.5

If a complete circle is programmed, with radius programming, the CNC will show the corresponding error, as infinite solutions exist.

If an arc is less than 180º, the radius is programmed with a plus sign, and a minus sign if it is more than 180º.

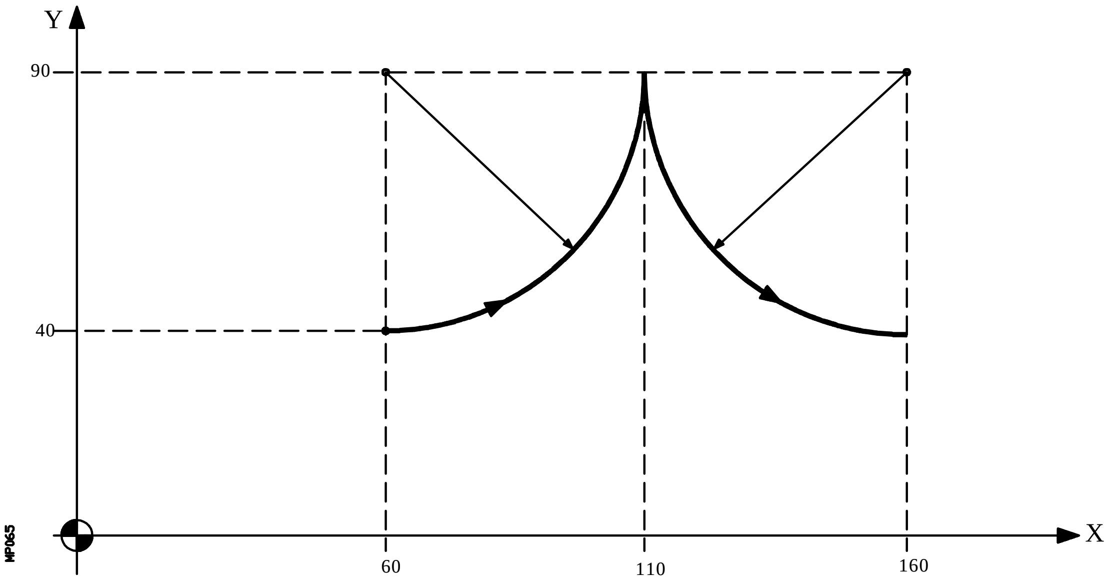


If P0 is the starting point and P1 the endpoint, there are 4 arcs which have the same value passing through both points.

Depending on the circular interpolation G02 or G03, and on the radius sign, the relevant arc is defined. Thus the programming format of the sample arcs is as follows:

Arc 1 G02 X.. Y.. R -..
Arc 2 G02 X.. Y.. R +..
Arc 3 G03 X.. Y.. R +..
Arc 4 G03 X.. Y.. R -..

Programming example:



Various programming modes are analyzed below, point X60 Y40 being the starting point.

Cartesian coordinates:

G90 G17 **G03 X110 Y90 I0 J50**
X160 Y40 I50 J0

Polar coordinates:

G90 G17 **G03 Q0 I0 J50**
Q-90 I50 J0

or:

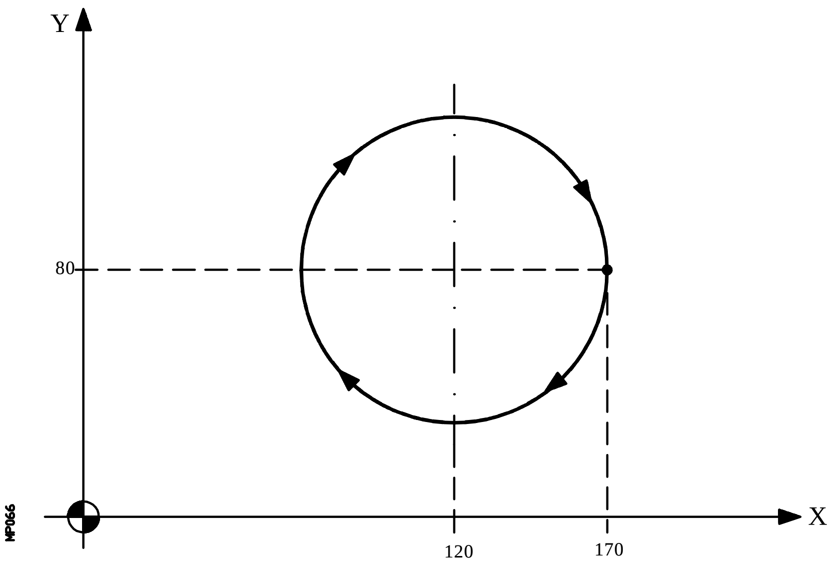
G93 I60 J90 ; defines polar center
G03 Q0
G93 I160 J90 ; defines new polar center
Q-90

Cartesian coordinates with radius programming:

G90 G17 **G03 X110 Y90 R50**
X160 Y40 R50

Example:

Programming of a (complete) circle in just one block:



Various programming modes analyzed below, point X170 Y80 being the starting Point.

Cartesian coordinates:

G90 G17 **G02 X170 Y80 I-50 J0**

or:

G90 G17 **G02 I-50 J0**

Polar coordinates:

G90 G17 **G02 Q360 I-50 J0**

or:

G93 I120 J80 ; defines polar center
G02 Q360

Cartesian coordinates with radius programming:

A complete circle cannot be programmed as there is an infinite range of solutions.

The CNC calculates, depending on the programmed arc, the radii of the starting point and endpoint. Although in theory both points should be exactly the same, the CNC enables you to select with the general machine parameter "CIRINERR", the maximum difference permissible between both radii, If this value is exceeded, the CNC displays the corresponding error.

The programmed feedrate "F" can be varied between 0% and 120% by using the switch located on the Operator Panel of the CNC, or by selecting it between 0% and 255% from the PLC, via the DNC or from the program.

The CNC, however, has general machine parameter "MAXFOVR" to limit the maximum variation of the feedrate.

If the general machine parameter "PORGMOVE" has been selected and a circular interpolation (G02 or G03) is programmed, the CNC assumes the center of the arc to be a new polar origin.

Functions G02 and G03 are modal and incompatible both among themselves and with G00, G01, G33 and G34. Functions G02 and G03 can be programmed as G2 and G3.

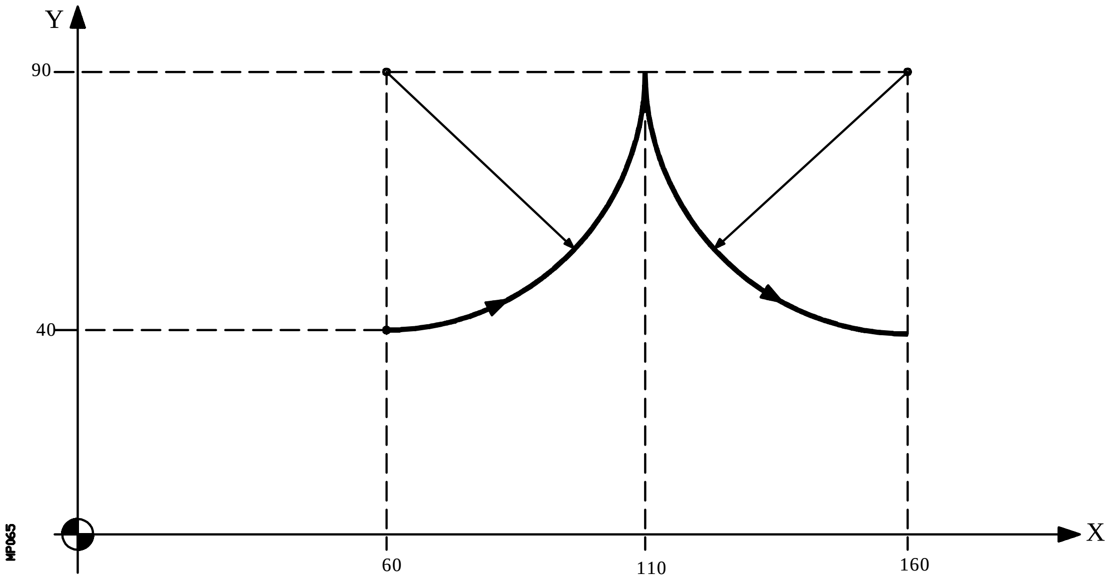
On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G00 or G01, depending on how general machine parameter "IMOVE" has been set.

6.4 CIRCULAR INTERPOLATION BY PROGRAMMING THE CENTER OF THE ARC IN ABSOLUTE COORDINATES (G06)

By adding function **G06** to a circular interpolation block you can program the coordinates of the center of the arc (I,J, or K) in absolute coordinates i.e. with respect to the zero origin and not to the beginning of the arc.

Function G06 is not modal, so it should be programmed any time the coordinates of the center of the arc are required in absolute coordinates. G06 can be programmed as G6.

Example:



Various programming modes are analyzed below, point X60 Y40 being the starting point.

Cartesian coordinates:

G90	G17	**G06**	G03	X110	Y90	**I60**	**J90**
		G06		X160	Y40	**I160**	**J90**

Polar coordinates:

G90	G17	**G06**	G03	Q0	**I60**	**J90**
		G06		Q-90	**I160**	**J90**

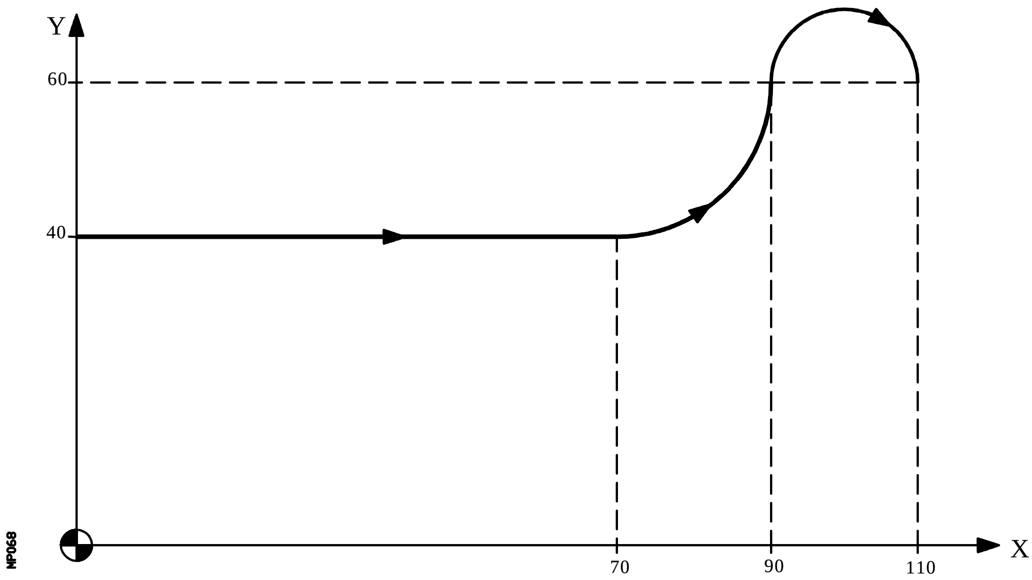
6.5 ARC TANGENT TO THE PREVIOUS PATH (G08)

Via function **G08** you can program an arc tangential to the previous path without having to program the coordinates (I.J &K) of the center.

Only the coordinates of the endpoint of the arc are defined, either in polar coordinates or in Cartesian coordinates according to the axes of the work plane.

Example:

Supposing that the starting point is X0 Y40, you wish to program a straight line, then an arc tangential to the line and finally an arc tangential to the previous one.




```
G90   G01   X70
      G08   X90   Y60   ; arc tangential to previous path
      G08   X110 Y60    ; arc tangential to previous path
```

Function G08 is not modal, so it should always be programmed if you wish to execute an arc tangential to the previous path. Function G08 can be programmed as G8.

Function G08 enables the previous path to be a straight line or an arc and does not alter its history. The same function G01, G02 or G03 stays active after the block is finished.

Warning:

When using function G08 it is not possible to execute a complete circle, as an infinite range of solutions exists. The CNC displays the corresponding error code.

6.6 *ARC DEFINED BY THREE POINTS (G09)*

Through function **G09** you can define an arc by programming the endpoint and an intermediate point (the starting point of the arc is the starting point of the movement). In other words, instead of programming the coordinates of the center, you program any intermediate point.

The endpoint of the arc is defined in Cartesian or polar coordinates, and the intermediate point is always defined in Cartesian coordinates by the letters I,J, or K, each one being associated to the axes as follows:

Axes X,U,A —> I
Axes Y,V,B —> J
Axes Z,W,C —> K

In Cartesian coordinates:

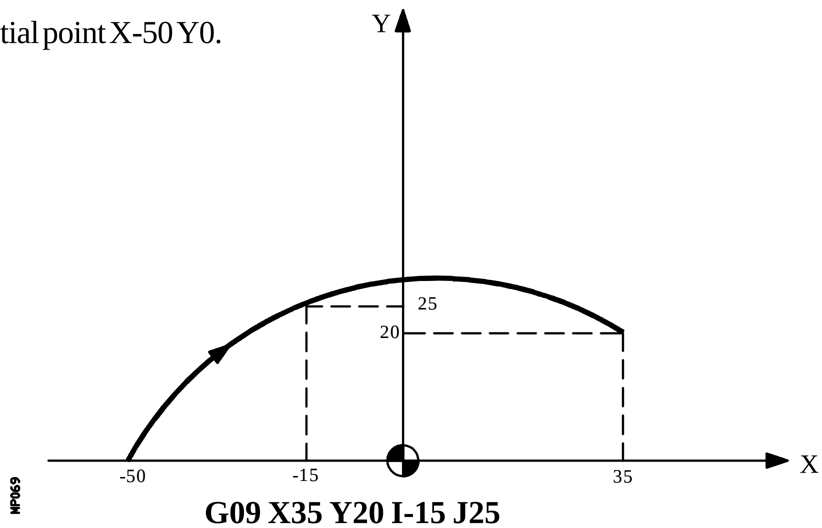
G17 **G09 X±5.5 Y±5.5 I±5.5 J±5.5**

Polar coordinates:

G17 **G09 R±5.5 Q±5.5 I±5.5 J±5.5**

Example:

Being initial point X-50 Y0.



G09 X35 Y20 I-15 J25

Function G09 is not modal, so it should always be programmed if you wish to execute an arc defined by three points. Function G09 can be programmed as G9.

When G09 is programmed it is not necessary to program the direction of movement (G02 or G03).

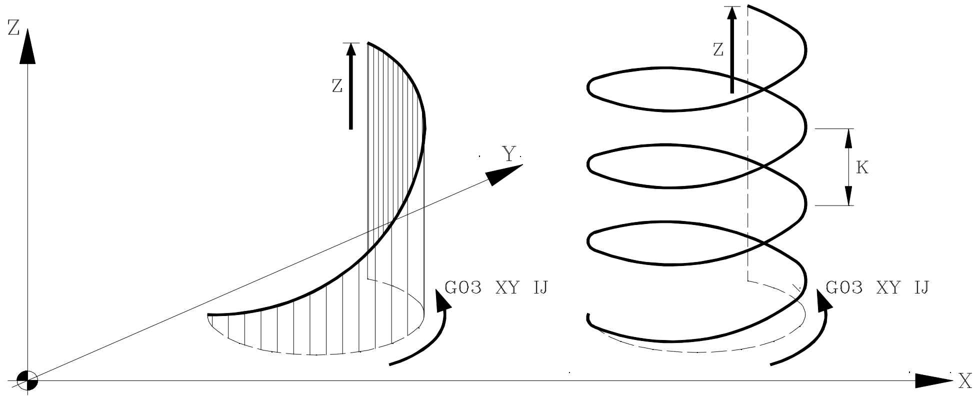
Function G09 does not alter the history of the program. The same G01, G02 or G03 function stays active after finishing the block.

Warning:

When using function G09 it is not possible to execute a complete circle, as you have to program three different points. The CNC displays the corresponding error code.

6.7 *HELICAL INTERPOLATION*

A helical interpolation consists in a circular interpolation in the work plane while moving the rest of the programmed axes.



The helical interpolation is programmed in a block where the circular interpolation must be programmed by means of functions: G02, G03, G08 or G09.

G02	X	Y	I	J	Z	G02	X	Y	R	Z	A
G03	Q	I	J	A	B	G08	X	Y	Z		
G09	X	Y	I	J	Z						

If the helical interpolation is supposed to make more than one turn, the linear movement of another axis must also be programmed (one axis only).

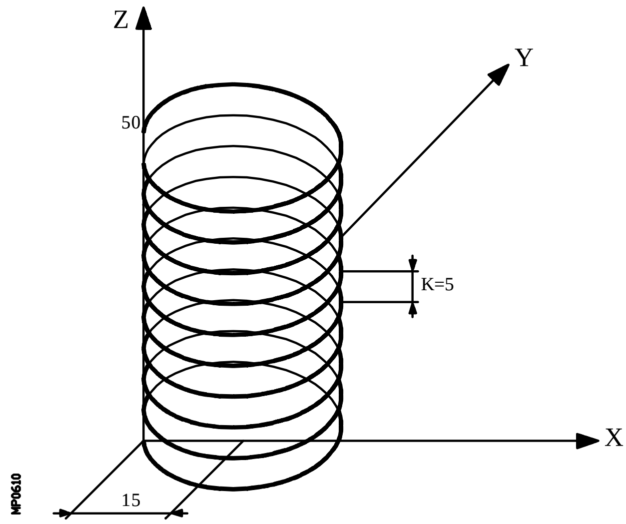
On the other hand, the pitch along the linear axis must also be set (format 5.5) by means of the I, J and K letters. Each one of these letters is associated with the axes as follows:

(I) for the X, U, A axes   (J) for the Y, V, B axes   (K) for the Z, W, C axes

G02	X	Y	I	J	Z	K	G02	X	Y	R	Z	K
G03	Q	I	J	A	I		G08	X	Y	B	J	
G09	X	Y	I	J	Z	K						

Example:

Programming in Cartesian and polar coordinates, the starting point being X0 Y0 Z0.



Cartesian coordinates:

G03 X0 Y0 I15 Z50 K5

Polar coordinates:

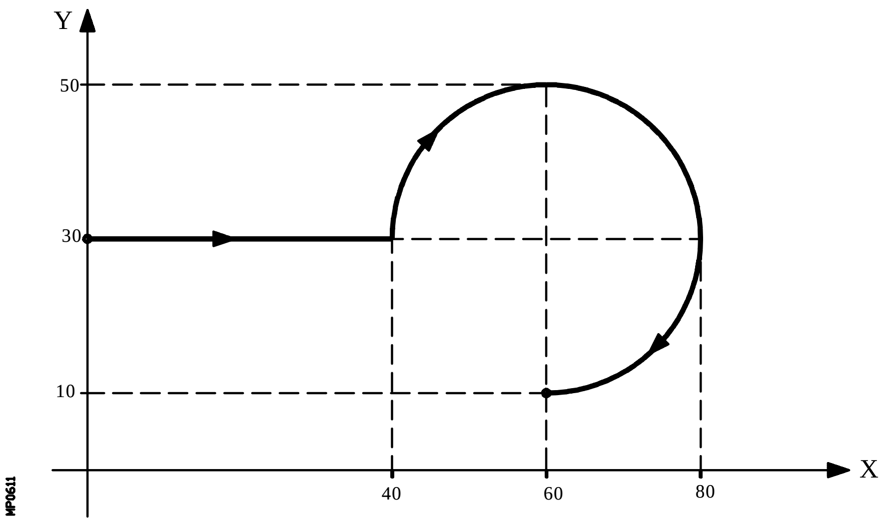
G03 Q180 I15 J0 Z50 K5

6.8 TANGENTIAL ENTRY AT BEGINNING OF A MACHINING OPERATION (G37)

Via function **G37** you can tangentially link two paths without having to calculate the intersection points.

Function G37 is not modal, so it should always be programmed if you wish to start a machining operation with tangential entry:

Example:

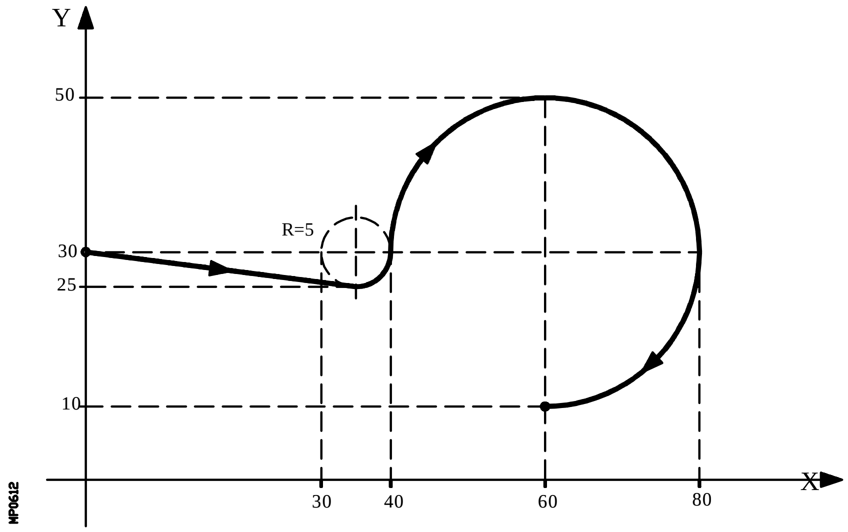


If the starting point is X0 Y30 and you wish to machine an arc (the path of approach being straight) you should program:

```
G90  G01 X40
     G02 X60 Y10 I20 J0
```

If, however, in the same example you require the entrance of the tool to the part to be machined tangential to the path and describing a radius of 5 mm, you should program:

```
G90  G01  G37  R5   X40
     G02  X60  Y10 I20  J0
```



As can be seen in the figure, the CNC modifies the path so that the tool starts to machine with a tangential entry to the part.

You have to program Function **G37** plus value **R** in the block which includes the path you want to modify.

R5.5 should appear in all cases following G37, indicating the radius of the arc which the CNC enters to obtain tangential entry to the part. Its value must always be positive.

Function G37 should only be programmed in the block which includes a straight-line movement (G00 or G01). If you program in a block which includes circular movement (G02 or G03), the CNC displays the corresponding error.

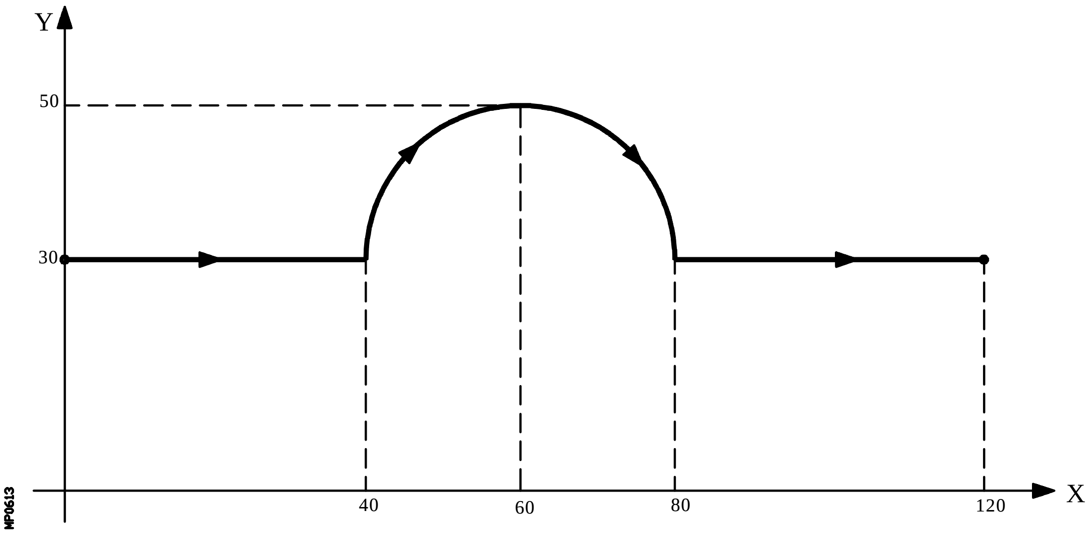
6.9 *TANGENTIAL EXIT AT THE END OF A MACHINING OPERATION (G38)*

Function **G38** enables the ending of a machining operation with a tangential exit of the tool. The path should be in a straight line (G00 or G01). Otherwise, the CNC will display the corresponding error.

Function G38 is not modal, so it should be programmed whenever a tangential exit of the tool is required.

Value **R 5.5** should always appear after G38. It also indicates the radius of the arc which the CNC applies to get a tangential exit from the part. This R value must always be positive.

Example:

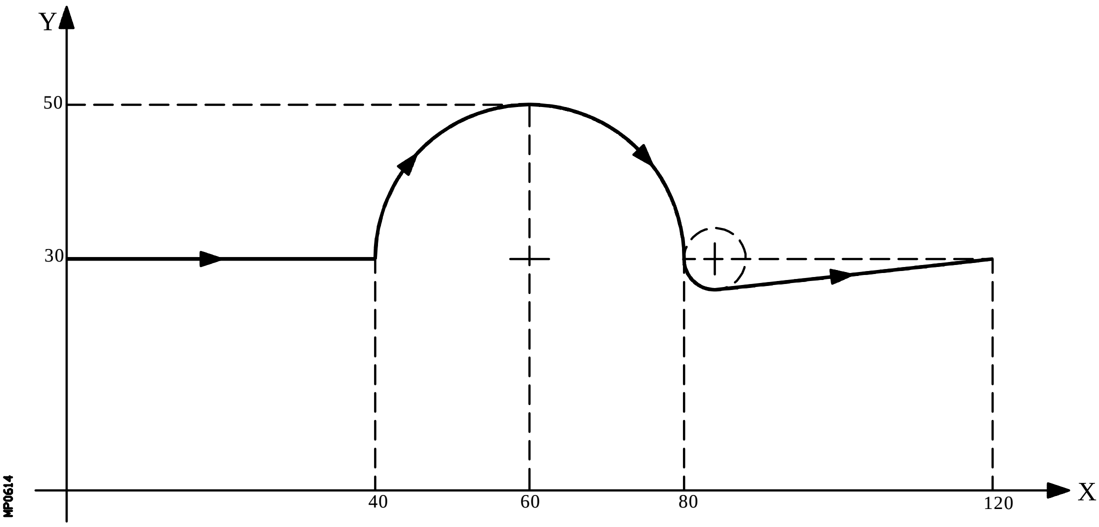


If the starting point is X0 Y30 and you wish to machine an arc (with the approach and exit paths in a straight line), you should program :

```
G90  G01  X40
     G02  X80   I20 J0
     G00  X120
```

If, however, in the same example you wish the exit from machining to be done tangentially and describing a radius of 5 mm, you should program :

```
G90 G01 X40
    G02 G38 R5 X80 I20 J0
    G00 X120
```



6.10 *AUTOMATIC RADIUS BLEND (G36)*

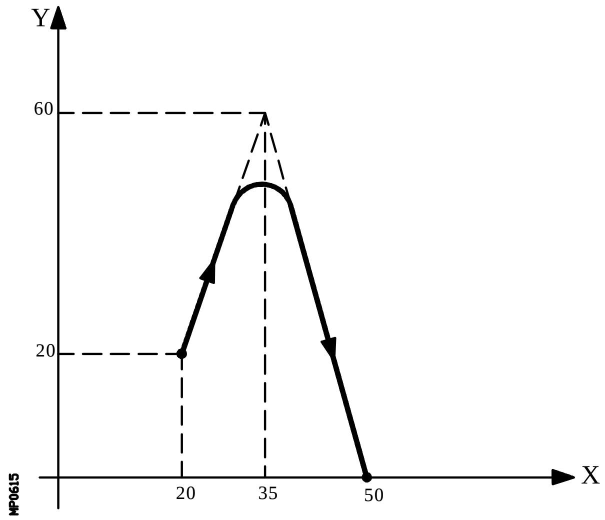
In milling operations, it is possible to round a corner via Function **G36** with a determined radius, without having to calculate the center nor the start and end points of the arc.

Function G36 is not modal, so it should be programmed whenever controlled corner rounding is required.

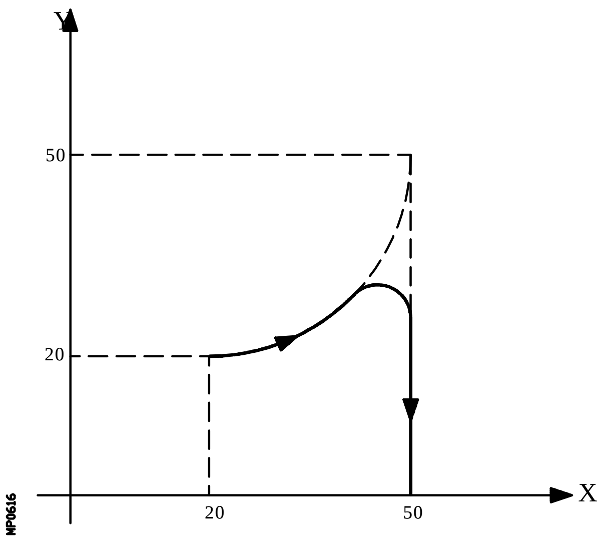
This function should be programmed in the block in which the movement the end you want to round is defined.

The **R5.5** value should always follow G36. It also indicates the rounding radius which the CNC applies to get the required corner rounding. This R value must always be positive.

Examples:



G90 G01 **G36** **R5** X35 Y60
X50 Y0



G90 G03 **G36** **R5** X50 I0 J30
G01 X50 Y0

6.11 *AUTOMATIC CHAMFER BLEND (G39)*

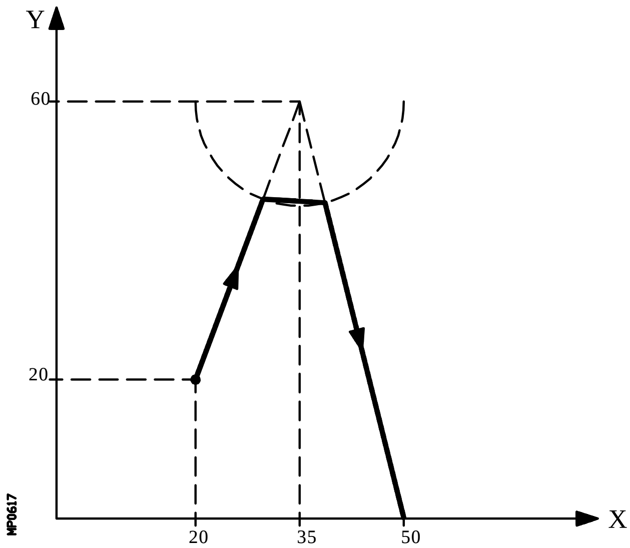
In machining operations it is possible (using G39) to chamfer corners between two straight lines, without having to calculate intersection points.

Function G39 is not modal, so it should be programmed whenever the chamfering of a corner is required.

This function should be programmed in the block in which the movement whose end you want to chamfer is defined.

The **R5.5** value should always follow G39. It also indicates the distance from the end of the programmed movement as far as the point where you wish to carry out the chamfering. This R value must always be positive.

Example :



G90 G01 **G39** **R5** X35 Y60
X50 Y0

6.12 *THREADING (G33)*

If the machine spindle is equipped with a rotary encoder, you can thread with a tool tip via function **G33**.

Although this threading is often done along the entire length of an axis, the CNC enables threading to be done interpolating more than one axis at a time.

Programming format: **G33 X.....C L Q**

X...C ±5.5	End point of the thread
L5.5	Thread pitch
Q ±3.5	Optional. It indicates the spindle angular position (±359.9999) of the thread's starting point. If not programmed, a value of "0" is assumed.

Considerations:

Whenever G33 is executed and before making the thread, the CNC referenced the spindle (home search) and positions the spindle at the angular position indicated by parameter Q.

Parameter "Q" is available when spindle machine parameter "M19TYPE" has been set to "1".

If the threads are blended together in round corner, only the first one can have an entry angle (Q).

While function G33 is active, neither the programmed feedrate "F" nor the programmed Spindle speed "S" can be varied. They will both be set to 100%.

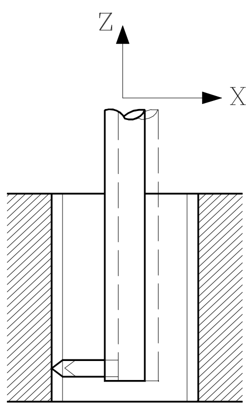
Function G33 is modal and incompatible with G00, G01, G02, G03, G34 and G75.

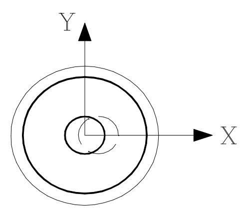
On power-up, after executing M02, M30 or after an EMERGENCY or RESET, the CNC assumes G00 or G01 depending on the setting of general machine parameter “IMOVE”

Example:

To make a 100mm deep and 5 mm pitch thread in a single pass at X0 Y0 Z0 with a threading tool located at Z10:

G90 G0 X Y Z	; Positioning
G33 Z-100 L5	; Threading
M19	; Spindle orientation
G00 X3	; Cutter withdrawal
Z30	; Withdrawal (exit the hole)





6.13 VARIABLE PITCH THREADS (G34)

In order to make variable pitch threads, the spindle must have a rotary encoder installed on it. Although this type of threads are often made along an axis, with this CNC they may be made by interpolating several axes simultaneously.

Programming format: G34 X.....C L Q K

X...C	±5.5	End-of-thread point
L	5.5	Starting pitch of the thread
Q	±3.5	Optional. It indicates the angular position of the spindle (±359.9999) for the starting point of the thread. This allows making threads with multiple entries. If not programmed, it assumes a value of "0"
K	±5.5	Increment or decrement of the thread pitch per spindle turn.

Considerations:

When executing function G34 and before making the thread, the CNC homes the spindle and positions it at the angular position indicate by parameter "Q".

Parameter "Q" is available when spindle machine parameter "M19TYPE=1".

When operating in round corner mode (G05), different threads may be joined together on a single part.

While G34 is active, neither the programmed feedrate F nor the programmed spindle speed S may be change. They are both set at 100%

G34 is modal and incompatible with G00, G01, G02, G03, G33 and G75.

On power-up, after executing an M02, M30 or after an EMERGENCY or RESET, the CNC assumes G00 or G01 depending on the setting of general machine parameter "IMOVE"

Joining a fixed-pitch thread (G33) with a variable-pitch thread (G34).

The starting thread pitch (L) of the G34 must be the same as that of the G33 thread.
The pitch increment in the first spindle turn in variable-pitch will be half the increment (K/2) and in the next turns will be the full increment (K).

Joining a variable-pitch thread (G34) with a fixed-pitch thread.

It is used to end a variable-pitch thread (G34) with a portion of the thread that still has the final pitch of the previous thread.
Since calculating the final thread pitch is rather complicated, the fixed-pitch thread is not programmed with G33, but with G34 … L0 K0. The CNC calculates the pitch.

Two variable-pitch threads (G34) CANNOT be programmed.

6.14 MOVE TO HARDSTOP (G52)

By means of function G52 it is possible to program the movement of an axis until running into an object. This feature may be interesting for forming machines, live tailstocks, bar feeders, etc.

Its programming format is: **G52 X..C ±5.5**

After G52, program the desired axis as well as the target coordinate of the move.

The axis will move towards the programmed target coordinate until running into something. If the axis reaches the programmed target coordinate without running into the hardstop it will stop.

Function G52 is not modal; therefore, it must be programmed every time this operation is to be carried out.

Also, it assumes functions G01 and G40 modifying the program history.
It is incompatible with functions G00, G02, G03, G34, G41, G42, G75 and G76.

6.15 FEEDRATE "F" AS AN INVERTED FUNCTION OF TIME (G32)

There are instances when it is easier to define the time required by the various axes of the machine to reach the target point instead of defining a common feedrate for all of them.

A typical case may be when a linear axis (X, Y, Z) has to move together (interpolated) with a rotary axis programmed in degrees.

Function G32 indicates that the "F" functions programmed next set the time it takes to reach the target point.

In order for a greater value of "F" to indicate a greater feedrate, the value assigned to "F" is defined as "Inverted function of time" and it is assumed as the activation of this feature.

"F" units: 1/min

Example: G32 X22 F4 indicates that the movement must be executed in ¼ minute. That is, in 0.25 minutes.

Function G32 is modal and incompatible with G94 and G95.

On power-up, after executing M02, M30 or after an Emergency or Reset, the CNC assumes G94 or G95 depending on the setting of general machine parameter "IFFED".

Considerations:

The CNC variable PROGFIN will show the feedrate programmed as an inverted function of time and variable FEED will show the resulting feedrate in mm/min or inches/min.

If the resulting feedrate of any axis exceeds the maximum value set by machine parameter "MAXFEED", the CNC will apply this maximum value.

The programmed "F" is ignored on G00 movements. All the movements will be carried out at the feedrate set by axis machine parameter "G00FEED".

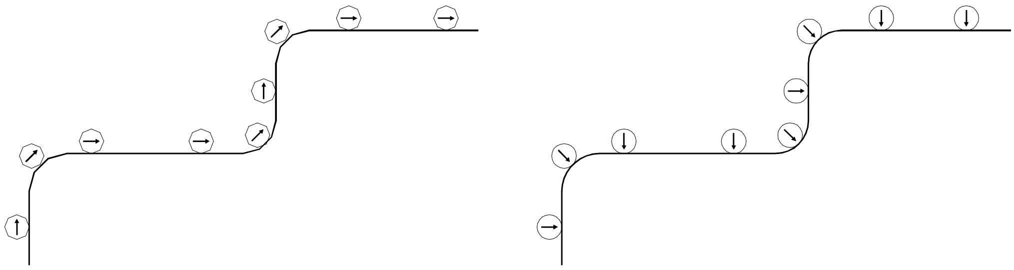
When programming "F0" the movement will be carried out at the feedrate set by axis machine parameter "MAXFEED".

Function G32 may be programmed and executed in the PLC channel.

Function G32 is canceled in JOG mode.

6.16 TANGENTIAL CONTROL (G45)

With the "Tangential control" feature, the axis may maintain the same orientation with respect to the programmed path.

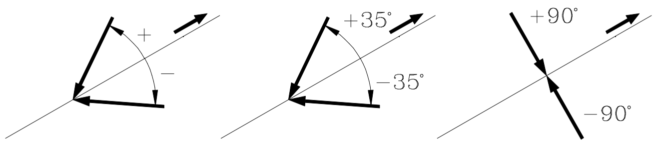


Orientation parallel to the path   Orientation perpendicular to the path

The path is defined by the axes of the active plane. The axis maintaining the orientation must be a rotary rollover axis (A, B or C).

Programming format: **G45 Axis Angle**

Axis	axis maintaining the orientation (A, B or C)
Angle	Indicates the angular position in degrees with respect to the path (±359.9999). If not programmed, "0" will be assumed.



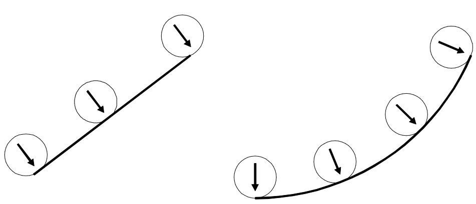
To cancel this function, program G45 alone (without defining the axis).

Every time G45 (tangential control) is activated, the CNC acts as follows:

1.- Positions the tangential axis, with respect to the first section in the programmed position.

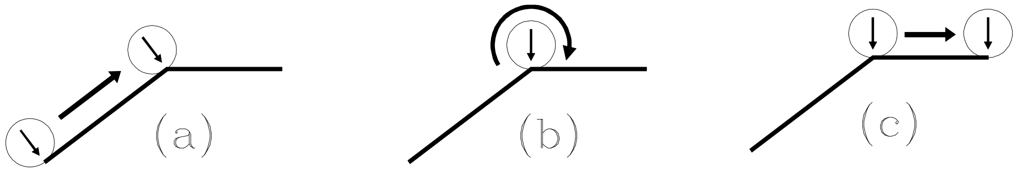
2.- The interpolation of the axes in the plane starts once the tangential axis has been positioned.

3.- On linear sections, the orientation of the tangential axis is maintained and in circular interpolations, the programmed orientation is maintained for the whole path.



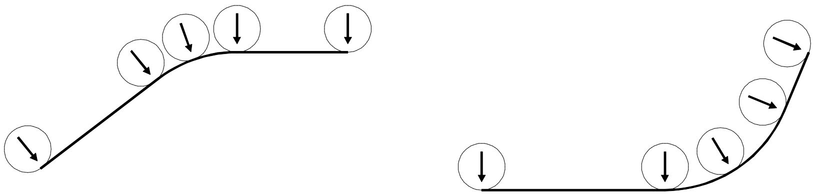
4.- If the joint of sections requires a new orientation of the tangential axis, the following takes place:

a) ends the current section.
b) orients the tangential axis with respect to the next section.
c) resumes execution.



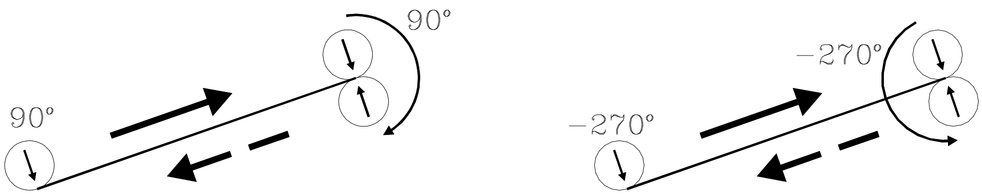
When working in round corner (G05), the tool orientation is not maintained at the corners since it begins before ending the current section.

It is recommended to work in square corner (G07). However, to work in round corner (G05), function G36 (automatic radius blend) should be used in order to also maintain tool orientation at the corners.



5.- To cancel the tangential control function, program G45 alone (without defining the axis).

Even when the tangential axis takes the same orientation by programming 90° or -270°, the turning direction in a direction change depends on the programmed value.



6.16.1 CONSIDERATIONS ABOUT FUNCTION G45

Tangential control, G45, is optional. It can only be executed in the main channel and is compatible with:

- Tool radius and length compensation (G40, 41, 42, 43, 44)
- Mirror image (G10, 11, 12, 13 14)
- Gantry axes , including the gantry axis associated with the tangential rotary axis.

The maximum feedrate while orienting the tangential axis is defined by machine parameter MAXFEED for that axis.

While tangential control is active, tool inspection is also possible. When accessing tool inspection, the tangential control is deactivated, the axes are free and when quitting tool inspection, tangential control may be activated again.

While in JOG mode, tangential control may be activated in MDI mode and the axes may be moved by programming blocks in MDI.

Tangential control is canceled when jogging the axes with the jog keys (not in MDI). Once the movement is over, tangential control is recovered.

On the other hand, the following is NOT possible:

- To define as tangential axis, one of the plane axes, the longitudinal axis or any other axis which is not rotary.
- To jog the tangential axis in JOG mode or by program using another G code while tangential control is active.
- Incline planes.

The TANGAN variable is read-only, from the CNC, PLC and DNC, associated with function G45. It indicates the angular position, in degrees, referred to the programmed path.

Also, general logic output TANGACT (M5558) indicates to the PLC that function G45 is active.

Function G45 is modal and is canceled when executing G45 alone (without defining the axis), on power-up, after executing an M02 or M30 or after an EMERGENCY or RESET.

7. ADDITIONAL PREPARATORY FUNCTIONS

7.1 INTERRUPTION OF BLOCK PREPARATION (G04)

The CNC reads up to 20 blocks ahead of the one it is executing, with the aim of calculating beforehand the path to be followed.

Each block is evaluated (in its absence) at the time it is read, but if you wish to evaluate it at the time of execution of the block you use function **G04**.

This function holds up the preparation of blocks and waits for the block in question to be executed in order to start the preparation of blocks once more.

A case in point is the evaluation of the "status of block-skip inputs" which is defined in the block header.

Example:

```
      .
      .
      .
     G04             ;  interrupts block preparation
  /1 G01  X10  Y20   ;  block-skip condition "/1"
      .
      .
```

Function G04 is not modal, so it should be programmed whenever you wish to interrupt block preparation.

It should be programmed on its own and in the block previous to the one in which the evaluation in execution is required. Function G04 can be programmed as G4.

Every time G04 is programmed, active radius and length compensation are cancelled.

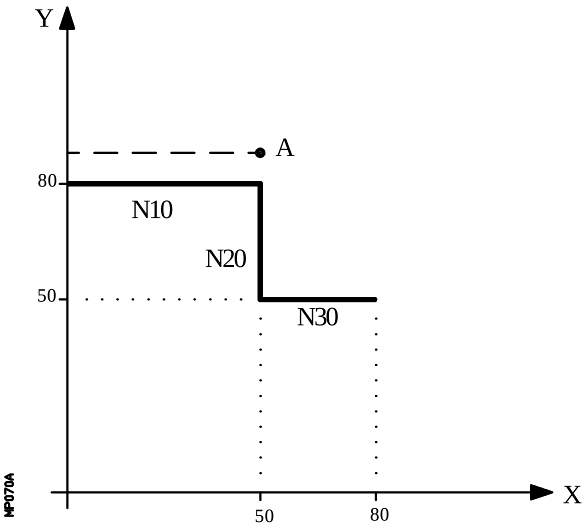
For this reason, care needs to be taken when using this function, because if it is introduced between machining blocks which work with compensation, unwanted profiles may be produced.

Example :

The following program blocks are executed in a section with G41 compensation :

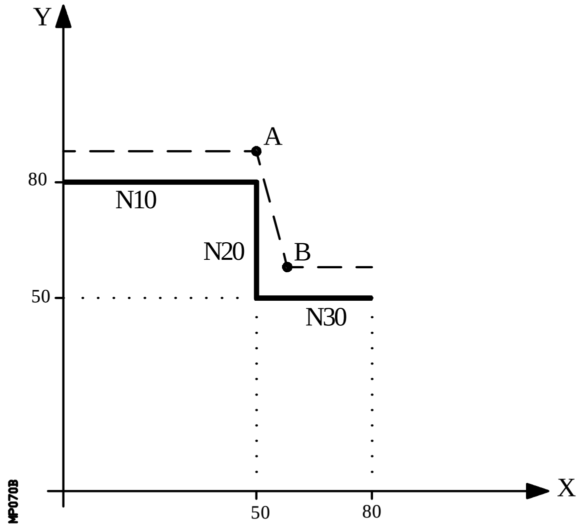
```
   ..........
   ..........
   N10 X50 Y80
   N15 G04
/1 N17 M10
    N20 X50 Y50
    N30 X80 Y50
   ..........
   ..........
```

Block N15 holds back the preparation of blocks so that the execution of block N10 ends up at point A.



Once the execution of block N15 has been carried out, the CNC continues preparing blocks starting from block N17.

Given that the next point corresponding to the compensated path is point “B”, the CNC moves the tool to this point, executing path “A-B”.



As you can see, the resulting path is not the required one, so we recommend avoiding the use of function G04 in sections which work with compensation.

7.2 DWELL (G04 K)

Timing can be programmed via function **G04 K**.

The timing value is programmed in hundredths of a second via format **K5 (0..99999).**

Example :

G04 K50 ; Timing of 50 hundredths of a second (0.5 seconds)
G04 K200 ; Timing of 200 hundredths of a second (2 seconds)

Function G04 K is not modal, so it should be programmed whenever timing is required. Function G04 K can be programmed as G4 K.

Timing is executed at the beginning of the block in which it is programmed.

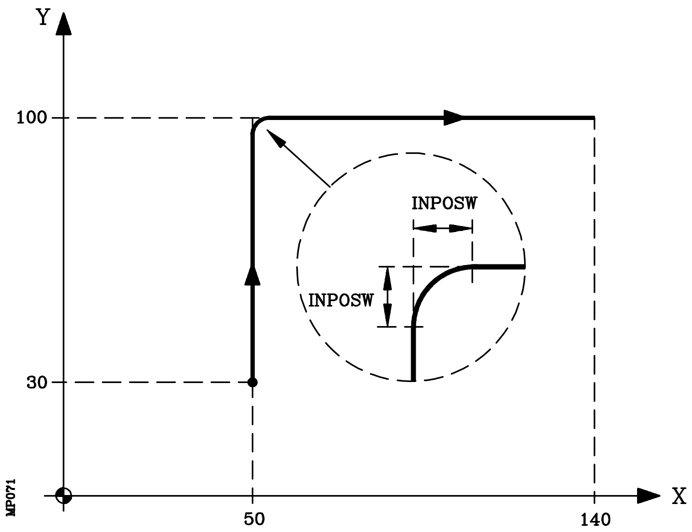
7.3 WORKING WITH SQUARE (G07) AND ROUND (G05,G50) CORNERS

7.3.1 SQUARE CORNER (G07)

When working in **G07** (square corner) the CNC does not start executing the following program block until the position programmed in the current block has been reached.

The CNC considers that the programmed position has been reached when the axis is within the "INPOSW" (in-position zone or dead band) from the programmed position.

Example:



G91 G01 **G07** Y70 F100
X90

The theoretical and real profile coincide, obtaining square corners, as seen in the figure.

Function G07 is modal and incompatible with G05 and G50. Function G07 can be programmed as G7.

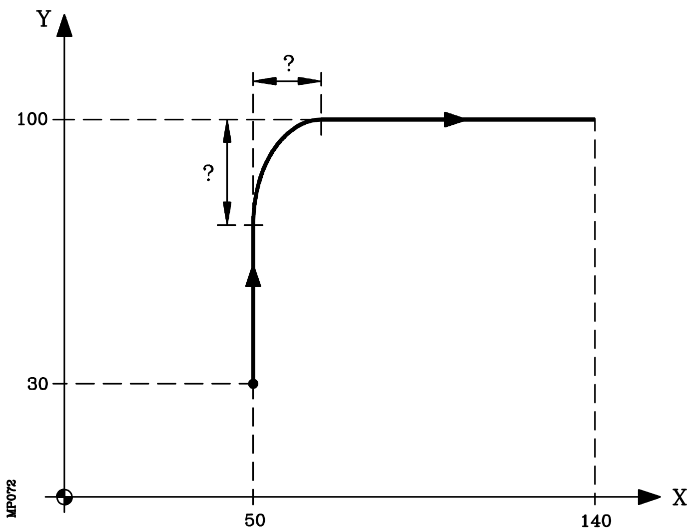
On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G05 or G07 depending on how the general machine parameter "ICORNER" is set.

7.3.2 *ROUND CORNER (G05)*

When working in **G05** (round corner), the CNC starts executing the following block of the program as soon as the theoretical interpolation of the current block has concluded. It does not wait for the axes to physically reach the programmed position.

The distance prior to the programmed position where the CNC starts executing the next block depends on the actual axis feedrate.

Example :



G91 G01 **G05** Y50 F100
X90

Via this function round corners can be obtained, as shown in the figure.

The difference between the theoretical and real profiles depends on the programmed feedrate value “F”. The higher the feedrate, the greater the difference between both profiles.

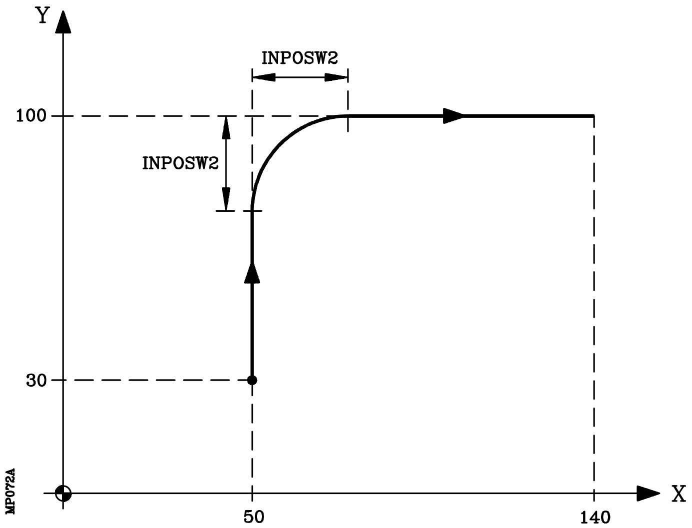
Function G05 is modal and incompatible with G07 and G50. Function G05 can be programmed as G5.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G05 or G07 depending on how the general machine parameter “ICORNER” is set.

7.3.3 *CONTROLLED ROUND CORNER (G50)*

When working in **G50** (controlled round corner); once the theoretical interpolation of the current block has concluded, the CNC waits for the axis to enter the area defined by machine parameter "INPOSW2" and it then starts executing the following block of the program.

Example :



G91 G01 **G50** Y50 F100
X90

Function G50 assures that the difference between the theoretical and actual paths stays smaller than what was set by machine parameter "INPOSW2".

On the other hand, when working in G05, the difference between the theoretical and real profiles depends on the programmed feedrate value “F”. The higher the feedrate, the greater the difference between both paths.

Function G50 is modal and incompatible with G07, G05 and G51.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G05 or G07 depending on how the general machine parameter “ICORNER” is set.

7.4 LOOK-AHEAD (G51)

Usually, a program consisting of very small movement blocks (CAM, etc.) run very slowly.

With this feature, high speed machining is possible for this type of programs.

It is recommended to have the CPU-TURBO feature when using LOOK-AHEAD because the CNC has to analyze the machining path ahead of time (up to 50 blocks) in order to calculate the maximum feedrate for each section of the path.

The programming format is: G51 [A] E

A (0-255)	Is optional and it defines the percentage of acceleration to be applied. When not programmed or programmed with a "0" value, the CNC assumes the acceleration value set by machine parameter for each axis.
E (5.5)	Maximum contouring error allowed.

Parameter "A" permits using a standard working acceleration and another one to be used when executing with Look-Ahead.

The smaller the "E" parameter value, the lower the machining feedrate.

When operating with "Look-Ahead", it is a good idea to adjust the axes so their following error (lag) is as small as possible because the contouring error will be at least equal to the minimum following error.

When calculating the axis feedrate, the CNC takes into consideration the following aspects:

* The programmed feedrate.
* The curvature and the corners.
* The maximum feedrates of the axes.
* The maximum accelerations.

If any of the circumstances listed below occurs while executing with Look-Ahead, the CNC slows down to "0" at the previous block and it recovers the machining conditions for Look-Ahead in the next motion block.

* Motionless block.
* Execution of auxiliary functions (M, S, T).
* Single block execution mode.
* MDI mode.
* TOOL INSPECTION mode.

If a Cycle Stop, Feed-Hold, etc. occurs while executing in Look-Ahead mode, the machine may not stop at the current block, several additional blocks will be necessary to stop with the permitted deceleration.

Function G51 is modal and incompatible with G05, G07 and G50. Should any of them be programmed, function G51 will be canceled and the new one will be selected.

On the other hand, the CNC will issue Error 7 (Incompatible G functions) when programming any of the following functions while G51 is active:

* G33 Electronic threading
* G34 Variable pitch thread
* G52 Movement against hardstop
* G74 Home search
* G75, G76 Probing
* G95 Feedrate per revolution

Function G51 must be programmed alone in a block and there must be no more information in that block.

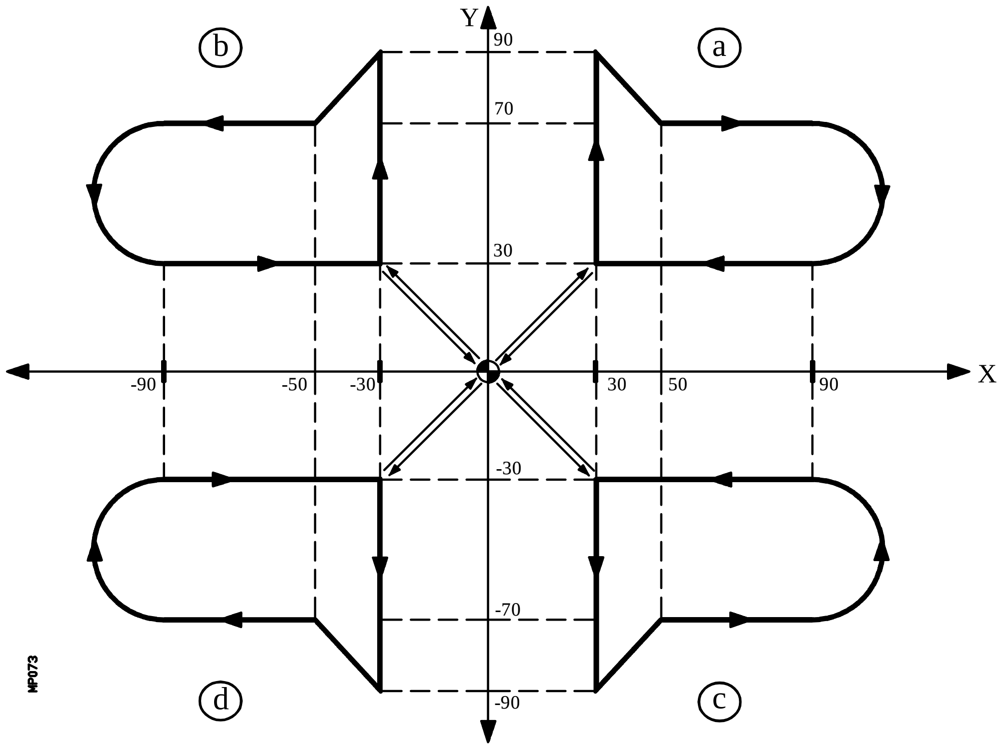
On power-up, after executing an M02, M30, of after an EMERGENCY or RESET, the CNC will cancel G51, if it was active, and it will assume G05 or G07 according to the setting of general machine parameter "ICORNER".

7.5 MIRROR IMAGE (G10, G11. G12, G13, G14)

G10: cancel mirror image
G11: mirror image on X axis
G12: mirror image on Y axis
G13: mirror image on Z axis
G14: mirror image on any axis (X..C), or in several at the same time.
Examples : G14 W   G14 X Z A B

When the CNC works with mirror images, it executes the movements programmed in the axes which have mirror image selected, with the sign changed.

Example :



The following subroutine defines the machining of part “a”.

```
G91  G01  X30  Y30  F100
               Y60
          X20  Y-20
          X40
     G02  X0   Y-40 I0    J-20
     G01  X-60
          X-30 Y-30
```

The programming of all parts would be :

Execution of subroutine	; machines “a”
G11	; mirror image on X axis.
Execution of subroutine	; machines “b
G10 G12	; mirror image on Y axis.
Execution of subroutine	; machines “c”
G11	; mirror image on X and Y axes.
Execution of subroutine	; machines “d”
M30	; end of program.

Functions G11, G12, G13, and G14 are modal and incompatible with G10.

G11, G12, and G13 can be programmed in the same block, because they are not incompatible with each other. Function G14 must be programmed alone in the block.

If function G73 (pattern rotation) is also active in a mirror image program, the CNC first applies the mirror image function and then the pattern rotation.

If while one of the mirror imaging functions (G11, G12, G13, and G14) is active, a new coordinate origin (part zero) is preset with G92, this new origin will not be affected by the mirror imaging function.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G10.

7.6 SCALING FACTOR (G72)

By using function **G72** you can enlarge or reduce programmed parts.

In this way, you can produce families of parts which are similar in shape but of different sizes with a single program.

Function G72 should be programmed on its own in a block. There are two formats for programming G72 :

Scaling factor applied to all axes.
Scaling factor applied to one or more axes.

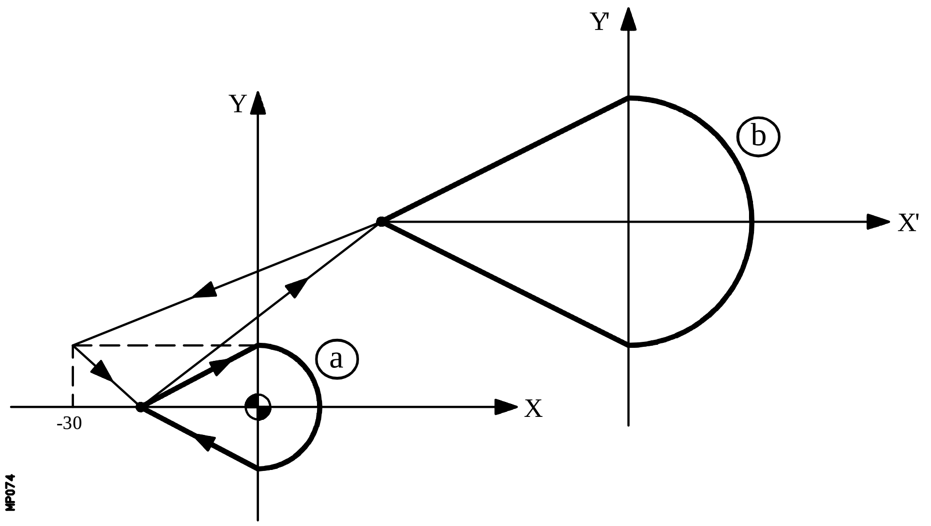
7.6.1 SCALING FACTOR APPLIED TO ALL AXES

The programming format is as follows :

G72 S5.5

Following G72 all coordinates programmed are multiplied by the value of the scaling factor defined by S until a new G72 scaling factor definition is read or the definition is cancelled.

Programming example (starting point X-30 Y10)



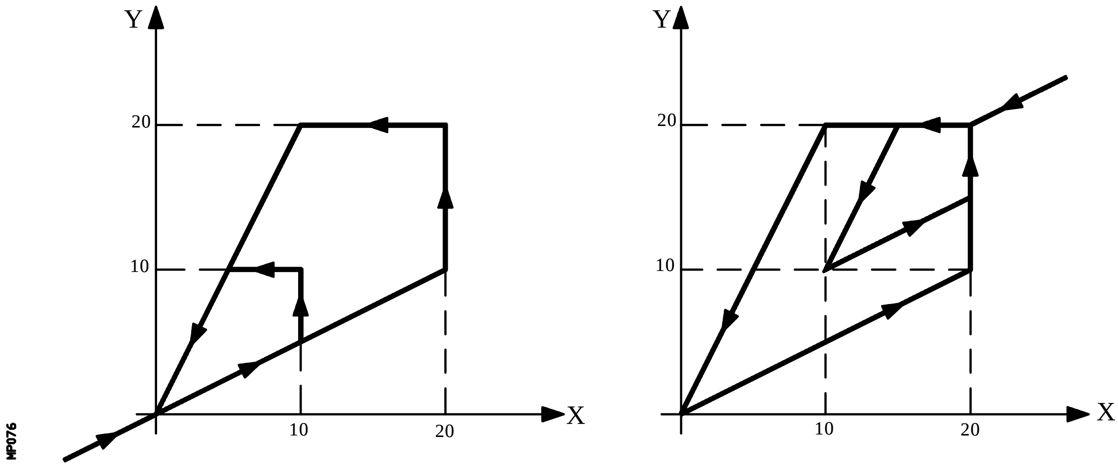
The following subroutine defines the machining of the part.

```
G90   X-19 Y0
      G01  X0    Y10  F150
      G02  X0    Y-10 I0      J-10
      G01  X-19 Y0
```

The programming of the parts would be :

Execution of subroutine	; machines "a"
G92 X-79 Y-30	; coordinate preset (zero offset)
G72 S2	; applies scaling factor 2
Execution of subroutine	; machines "b"
G72 S1	; cancels scaling factor
M30	; end of program

Examples of application of the scaling factor.




```
        G90  G00     X0      Y0
N10     G91      G01 X20     Y10
                             Y10
                     X-10
N20                  X-10    Y-20
        G72 S0.5
        (RPT N10,20)

          M30
```

```
        G90      G00     X20     Y20
N10     G91      G01     X-10
                         X-10    Y-20
                         X20     Y10
N20                              Y10
        G72 S0.5                 ; scaling factor
(RPT N10,20) ; repeats from
                                 ; block 10
                                 ; to block 20
        M30
```

Function G72 is modal and is cancelled when another scaling factor with a value of S1 is programmed, or on power-up, after executing M02, M30 or after EMERGENCY or RESET.

7.6.2 *SCALING FACTOR APPLIED TO ONE OR MORE AXES*

The programming format is :

G72 X...C 5.5

After **G72** the axis or axes and the required scaling factor are programmed.

All blocks programmed after G72 are treated by the CNC as follows :

The CNC calculates the movement of all the axes in relation to the programmed path and compensation.

It then applies the scaling factor indicated to the calculated movement of the corresponding axis or axes.

If the scaling factor is applied on one or more axes, the CNC will apply the scaling factor indicated both to the movement of the corresponding axis or axes and to their feedrate.

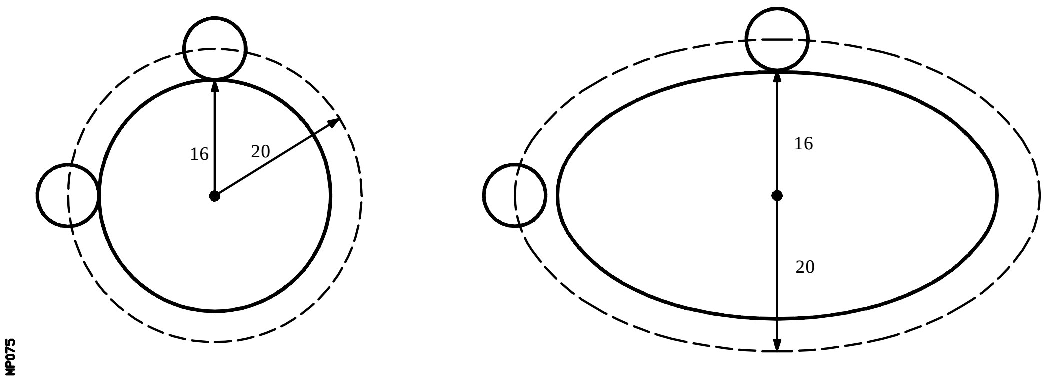
If, within the same program, both scaling factor types are applied, the one applied to all the axes and the one for one or several axes, the CNC applies a scaling factor equal to the product of the two scaling factors programmed for this axis to the axis or axes affected by both types.

Function G72 is modal and will be cancelled when the CNC is turned on, after executing M02, M30 or after an EMERGENCY or RESET.

Note: When simulating without moving the axes, this scaling factor is ignored

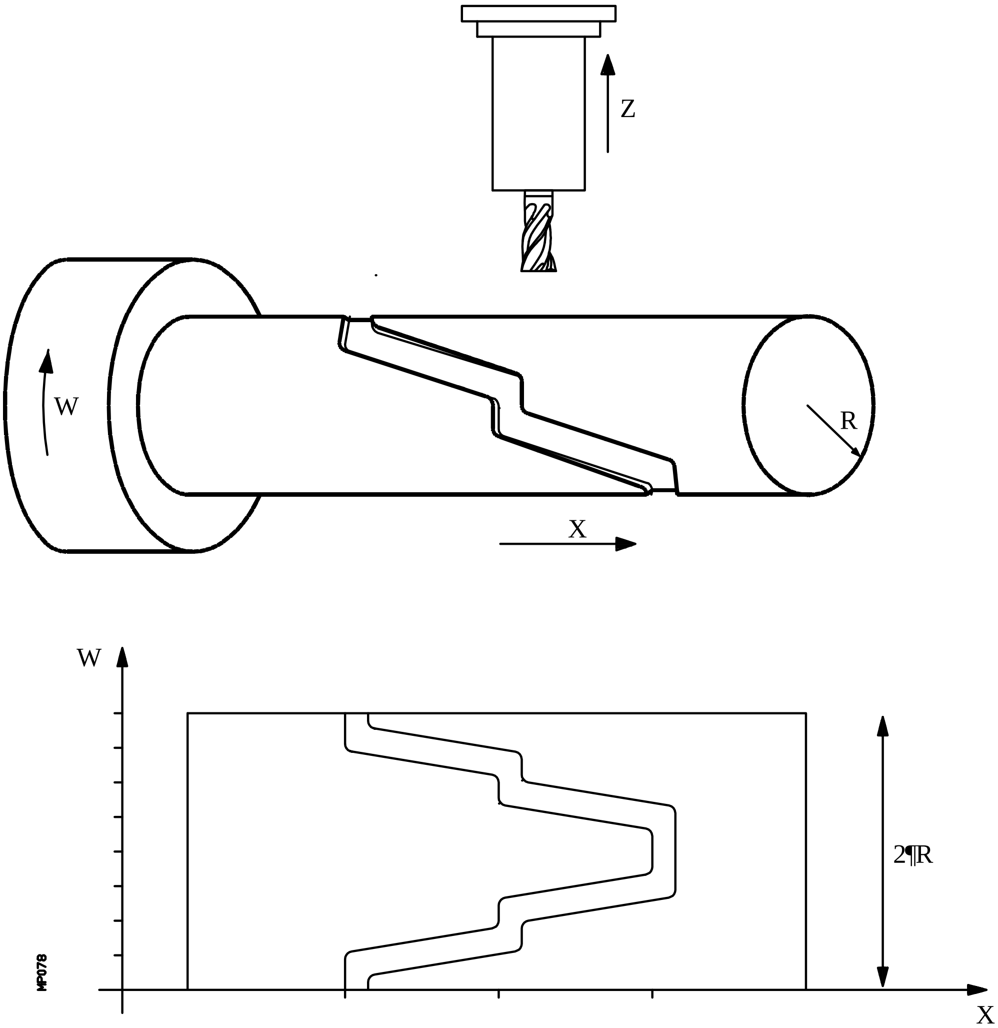
Example:

Application of the scaling factor to a plane axis, working with tool radius compensation.



As it can be observed, the tool path does not coincide with the required path, as the scaling factor is applied to the calculated movement.

However, if a scaling factor equal to 360/(2¶R) is applied to a rotary axis, R being the radius of the cylinder on which you wish to machine, this axis can be considered linear, and any figure with tool radius compensation can be programmed on the cylindrical surface.



7.7 PATTERN ROTATION (G73)

Function **G73** enables you to turn the system of coordinates, taking either the coordinates origin or the programmed rotation center as the active rotation center.

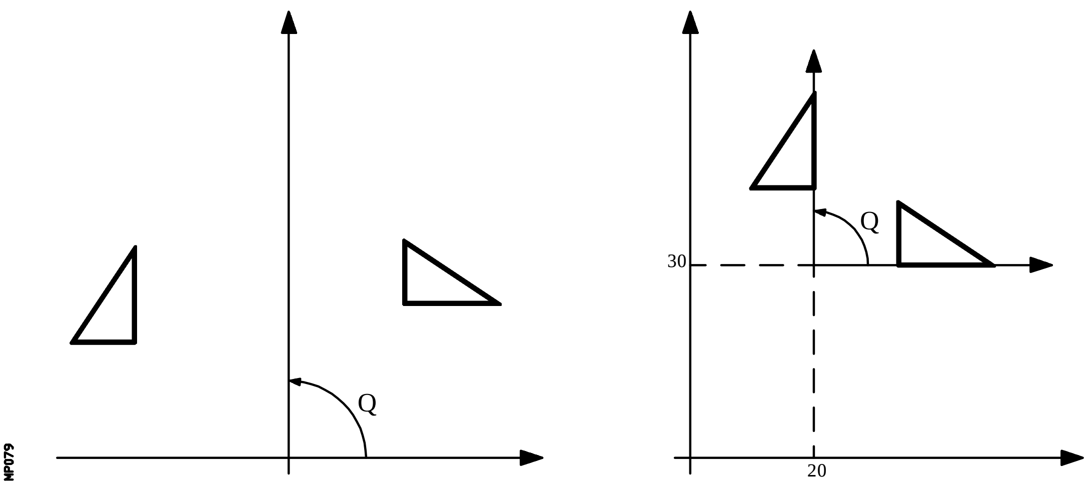
The format which defines the rotation is the following :

G 73 Q+/5.5 I±5.5 J±5.5

In which :

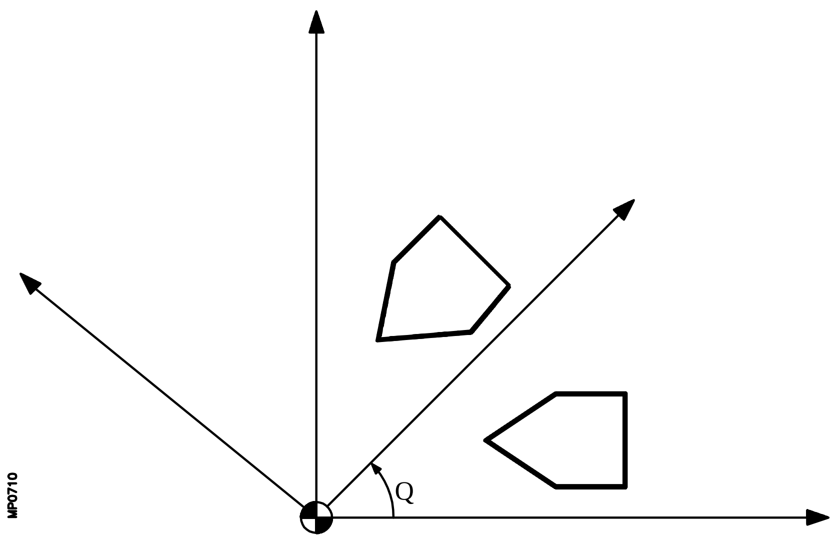
Q : indicates the angle of rotation in degrees
I,J : are optional and define the abscissa and ordinate respectively of the rotation center. If they are not defined, the coordinate origin will be taken as the rotation center.

Values I and J are defined in absolute coordinates and referred to the coordinate origin of the work plane. These coordinates are affected by the active scaling factor and mirror images.



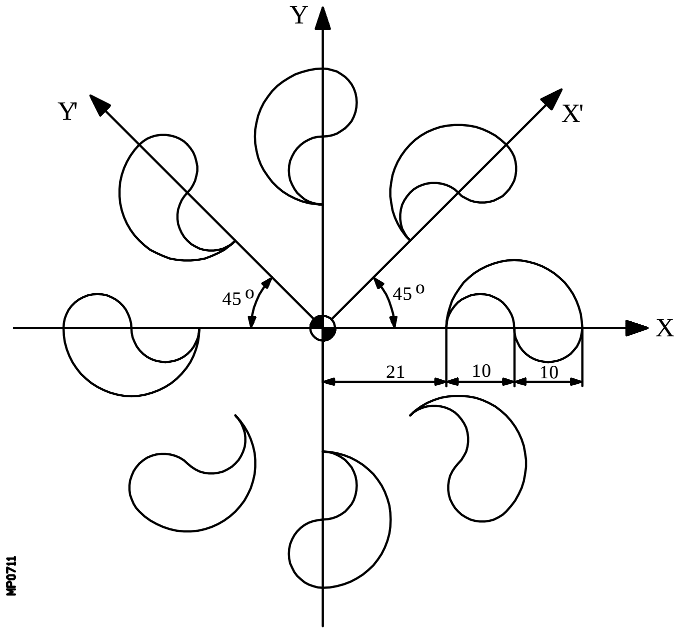
G73 Q90 G73 Q90 I20 J30

You should remember that G73 is incremental i.e. the different Q values programmed add up.



Function G73 should be programmed on its own in a block.

Example :



Assuming that the starting point is X0 Y0, you get :

```
N10  G01  X21  Y0    F300  ; positioning at starting point
     G02  Q0   I5    J0
     G03  Q0   I5    J0
          Q180I-10   J0
N20  G73  Q45              ; pattern rotation
     (RPT N10,20)    N7    ; repeat blocks 10 thru 20 seven times
     M30                   ; end of program
```

In a program which rotates the coordinate system, if any mirror image function is also active the CNC first applies the mirror image function and then the turn.

The pattern rotation function can be cancelled either by programming G73 (on its own, without angle value) or via G16, G17, G18, or G19, or on power-up, after executing M02, M30 or after EMERGENCY or RESET.

7.8 SLAVED AXIS/CANCELLATION OF SLAVED AXIS

The CNC enables two or more axes to be coupled together. The movement of all axes is subordinated to the movement of the axis to which they were coupled.

There are three possible ways of coupling axes :

Mechanical coupling. This is imposed by the manufacturer of the machine, and is selected via the axis machine parameter “GANTRY”.

By means of the PLC. This enables the coupling and uncoupling of each axis through logic input on the CNC “SYNCHRO1”, “SYNCHRO2”, “SYNCHRO3”, “SYNCHRO4”, and “SYNCHRO5”. Each axis is coupled to the one indicated in the axis machine parameter “SYNCHRO”.

By means of the program. This enables electronic coupling and uncoupling between two or more axes, through functions G77 and G78.

7.8.1 *SLAVED AXIS (G77)*

Function G77 allows the selection of both the master axis and the slaved axis (axes). The programming format is as follows :

G77 < Axis 1 > < Axis 2 > < Axis 3 > < Axis 4 > < Axis 5>

In which < Axis 2 > < Axis 3 > < Axis 4 > < Axis 5> indicate the slave axes you wish to couple to the master axis < Axis 1 >. You have to define < Axis 1 > and < Axis 2 >, the programming of the rest of the axes being optional.

Example :

```
G77 X Y U   ; couples Y and U axes to X axis
```

The following rules should be observed when doing electronic axis couplings :

You may use one or two different electronic couplings.

```
G77 X Y U   ; couples Y and U axes to X axis
G77 V Z     ; couples Z axis to V axis
```

You cannot couple one axis to two others at the same time.

```
G77 V Y     ; couples Y axis to V axis
G77 X Y     ; gives an error signal, because Y axis is coupled to V axis.
```

You can couple several axes to one in successive steps.

```
G77 X Z     ; couples Z axis to X axis
G77 X U     ; couples U axis to X axis —> Z U coupled to X
G77 X Y     ; couples Y axis to X axis —> Y Z U coupled to X
```

A pair of axes which are already coupled to each other cannot be coupled to another axis.

```
G77 Y U     ; couples U axis to Y axis
G77 X Y     ; gives an error signal, because Y axis is coupled to U axis.
```

7.8.2 SLAVED AXIS CANCELLATION (G78)

Function G78 enables you to uncouple all the axes which are coupled (slaved), or only uncouple indicated axes.

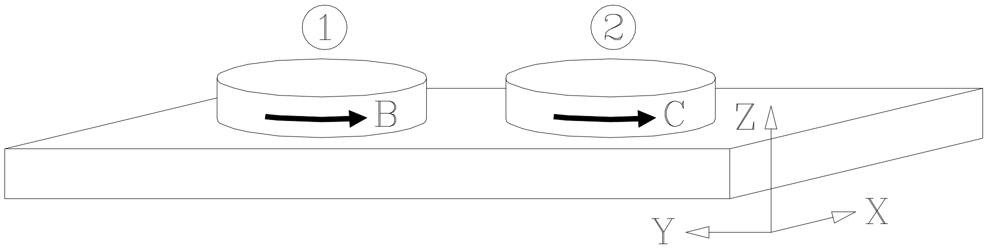
G78	Uncouples all slaved axes.
G78 <Axis 1><Axis 2><Axis 3><Axis 4>	Only uncouples indicated axes.

Example :

```
G77 X Y U ; slaves Y and U axes to X axis
G77 V Z   ; slaves Z axis to V axis
G78 Y     ; uncouples Y axis, but U stays slaved to X and Z to V.
G78       ; uncouples all axes.
```

7.9 *AXES TOGGLE. G28-G29*

With this feature, on machines having two machining tables, it is possible to use a single part-program to make the same parts on both tables.



With function G28 the axes can be toggled from one to the other in such way that after that instruction all the movements associated with the first axis next to G28 will take place on the second axis next to G28 and vice versa.

Programming format: G28 (axis 1) (axis 2)

To cancel the toggle, execute function G29 followed by one of the axes to be toggled back.

Up to three pairs of axes may be toggled at the same time.

The main axes cannot be toggled in the following cases: While tracing, while function G48 or G49 is active or when the "C" axis is active on a lathe.

On power-up, after executing an M30 or after an emergency or reset, the axes are toggled back as long as G48 or G49 is not active.

Example. Let us suppose that the part program is defined for table 1.

Execute the part-program on table 1	
G28 BC	Toggle the "B" and "C" axes
Zero offset	for machining on table 2
Execute the part-program	It will be executed on table 2
	In the meantime, replace the part made on table 1 with a new one
G29 B	Toggle the "B" and "C" axes back
Cancel the zero offset	for machining on table 1
Execute the part-program	It will be executed on table 1
	In the meantime, replace the part made on table 2 with a new one

8. *TOOL COMPENSATION*

The CNC has a tool offset table, its number of components being defined via the general machine parameter "NTOFFSET". The following is specified for each tool offset :

* Tool radius, in work units, in R±5.5 format
* Tool length, in work units, in L±5.5 format.
* Wear of tool radius, in work units, in I±5.5 format. The CNC adds this value to the theoretical radius (R) to calculate the real radius (R+I).
* Wear of tool length, in work units, in K±5.5 format. The CNC adds this value to the theoretical length (L) to calculate the real length (L+K).

When tool radius compensation is required (G41 or G42), the CNC applies the sum of R+I values of the selected tool offset as the compensation value.

When tool length compensation is required (G43), the CNC applies the sum of L+K values of the selected tool offset as the compensation value.

8.1 *TOOL RADIUS COMPENSATION (G40, G41, G42)*

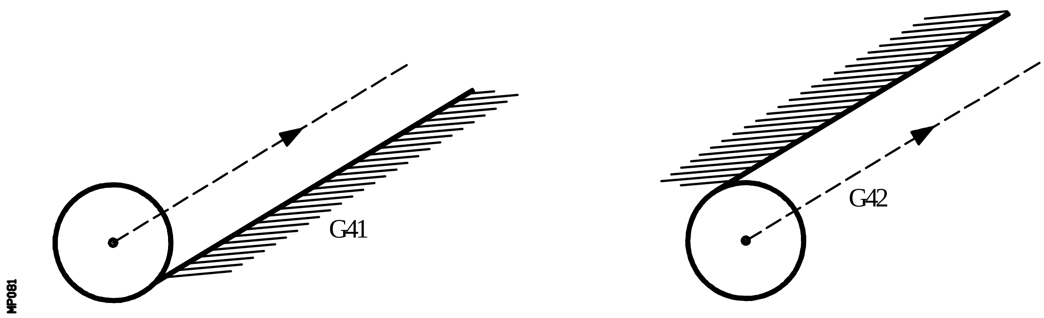
In normal milling operations, it is necessary to calculate and define the path of the tool taking its radius into account so that the required dimensions of the part are achieved.

Tool radius compensation allows the direct programming of part contouring and of the tool radius without taking the dimensions of the tool into account.

The CNC automatically calculates the path the tool should follow based on the contour of the part and the tool radius value stored in the tool offset table.

There are three preparatory functions for tool radius compensation:

G40 Cancelling of tool radius compensation
G41 Tool radius compensation to the left of the part.
G42 Tool radius compensation to the right of the part.



G41. The tool is to the left of the part, depending on the machining direction.

G42. The tool is to the right of the part, depending on the machining direction.

Tool values **R, L, I, K** should be stored in the tool offset table before starting machining, or should be loaded at the beginning of the program via assignments to variables **TOR, TOL, TOI, TOK.**

Once the plane in which compensation will be applied has been chosen via codes G16, G17, G18, or G19, this is put into effect by **G41** or **G42**, assuming the value of the tool offset selected via code **D**, or (in its absence) by the tool offset shown in the tool table for the selected tool (**T**).

Functions G41 and G42 are modal and incompatible to each other. They are cancelled by G40, G04 (interruption of block preparation), G53 (programming with reference to machine zero), G74 (home search), machining canned cycles (G81, G82, G83, G84, G85, G86, G87, G88, G89) and also on power-up, after executing M02, M30 or after EMERGENCY or RESET.

8.1.1 *ACTIVATING TOOL RADIUS COMPENSATION*

Once the plane in which tool radius compensation has been selected (via G16, G17, G18, or G19), functions G41 or G42 must be used to activate it.

G41 Compensation of tool radius compensation to the left.
G42 Compensation of tool radius compensation to the right.

In the same block (or a previous one) in which G41 or G42 is programmed, functions **T, D,** or only **T** must be programmed so that the tool offset value to be applied can be selected from the tool offset table. If no tool offset is selected, the CNC takes D0 with R0 L0 I0 K0.

When the new selected tool has an M06 associated to it and this M06, in turn, has a subroutine associated to it; the CNC will activate the tool radius compensation at the first movement block of that subroutine.

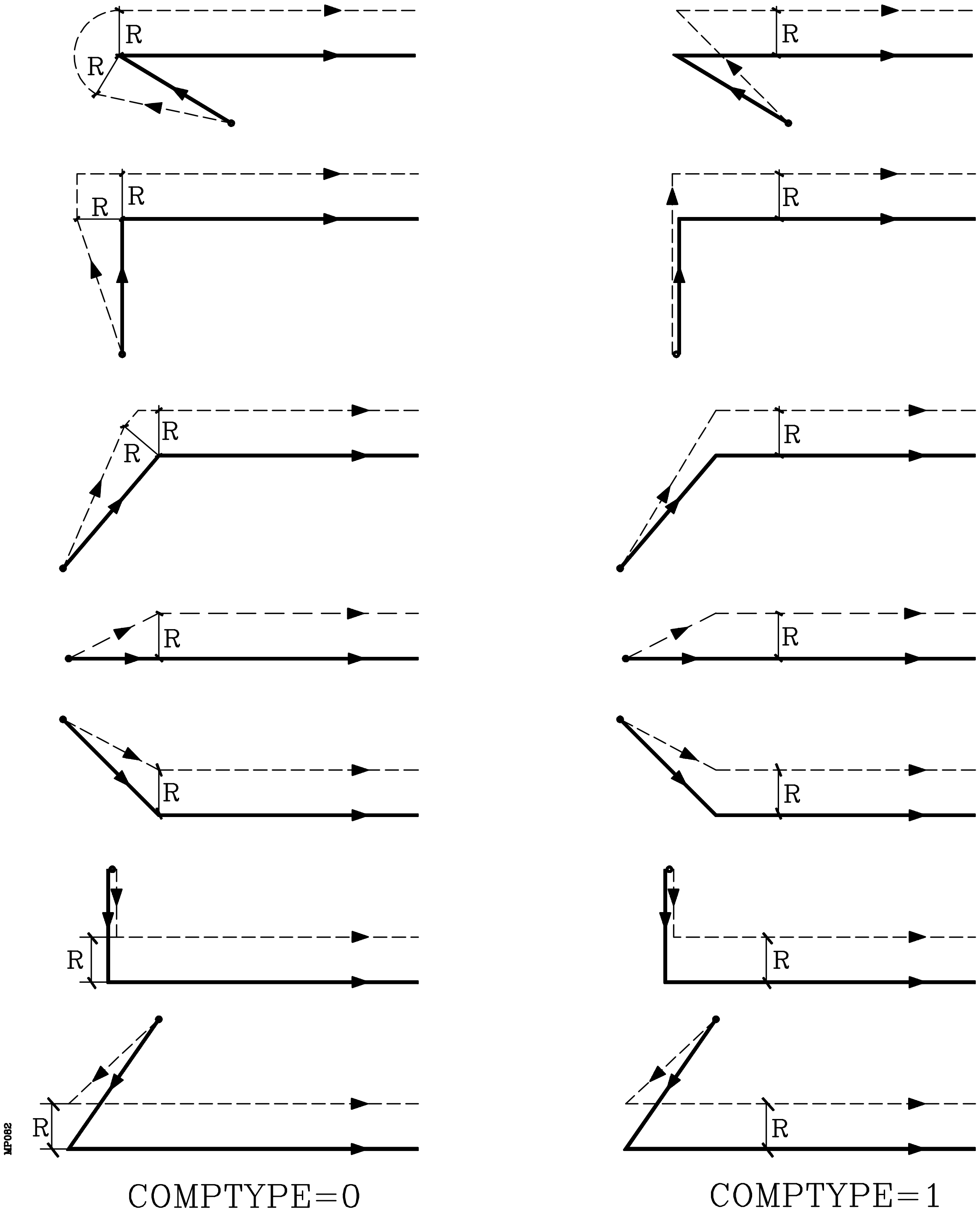
If that subroutine has a G53 programmed in a block (position values referred to Machine Reference Zero, home), the CNC will cancel any tool radius compensation (G41 or G42) selected previously.

The selection of tool radius compensation (G41 or G42) can only be made when functions G00 or G01 are active (straight-line movements).

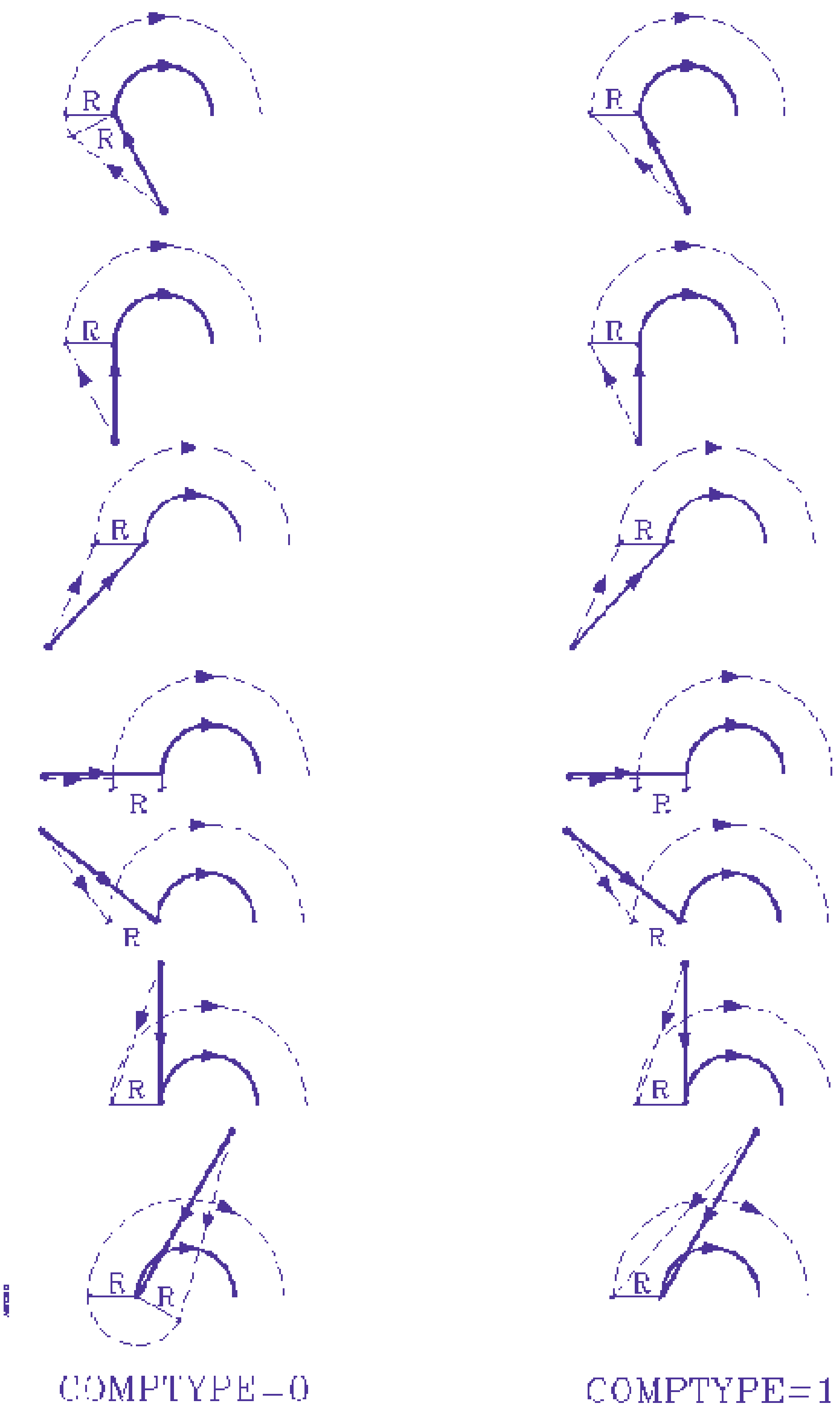
If the compensation is selected while G02 or G03 are active, the CNC will display the corresponding error message.

The following pages show different cases of starting tool radius compensation, in which the programmed path is represented by a solid line and the compensated path with a dotted line.

STRAIGHT-STRAIGHT path



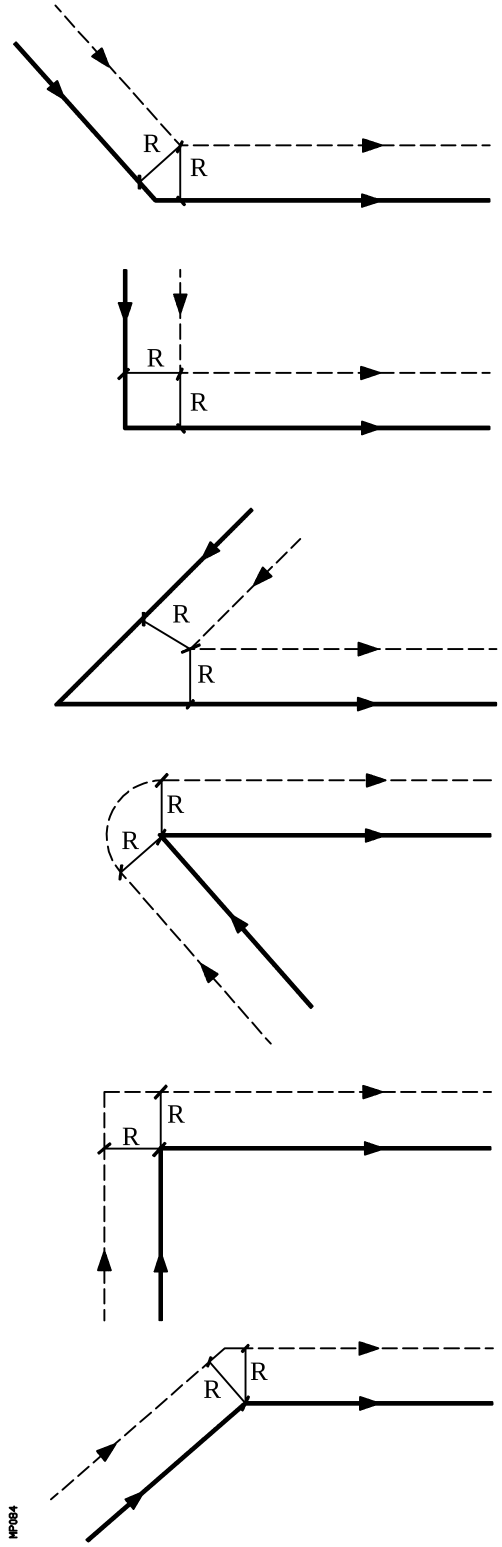
STRAIGHT-CURVED path

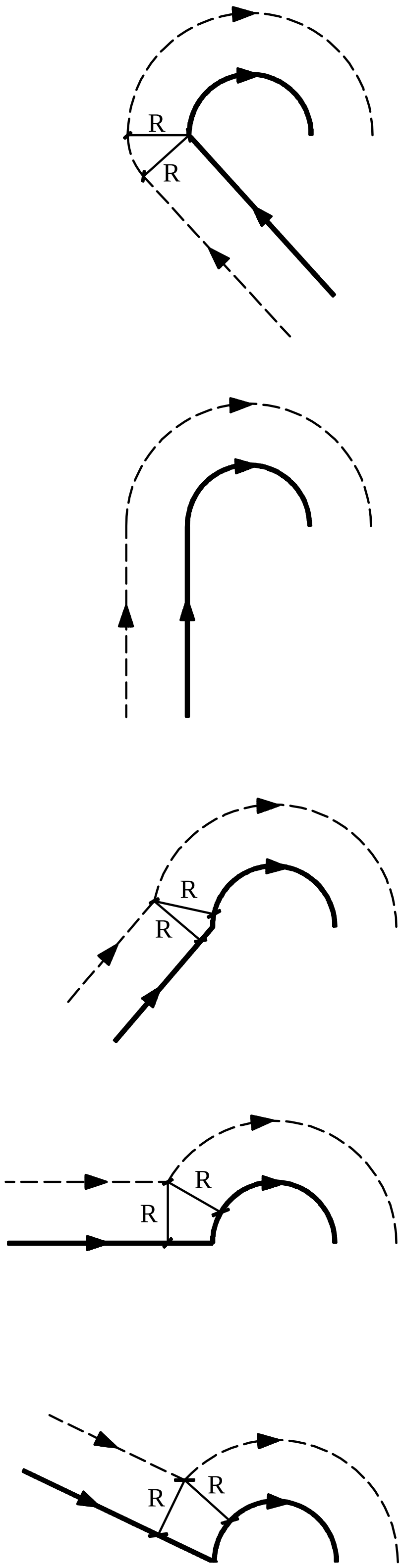


8.1.2 TOOL RADIUS COMPENSATION SECTIONS

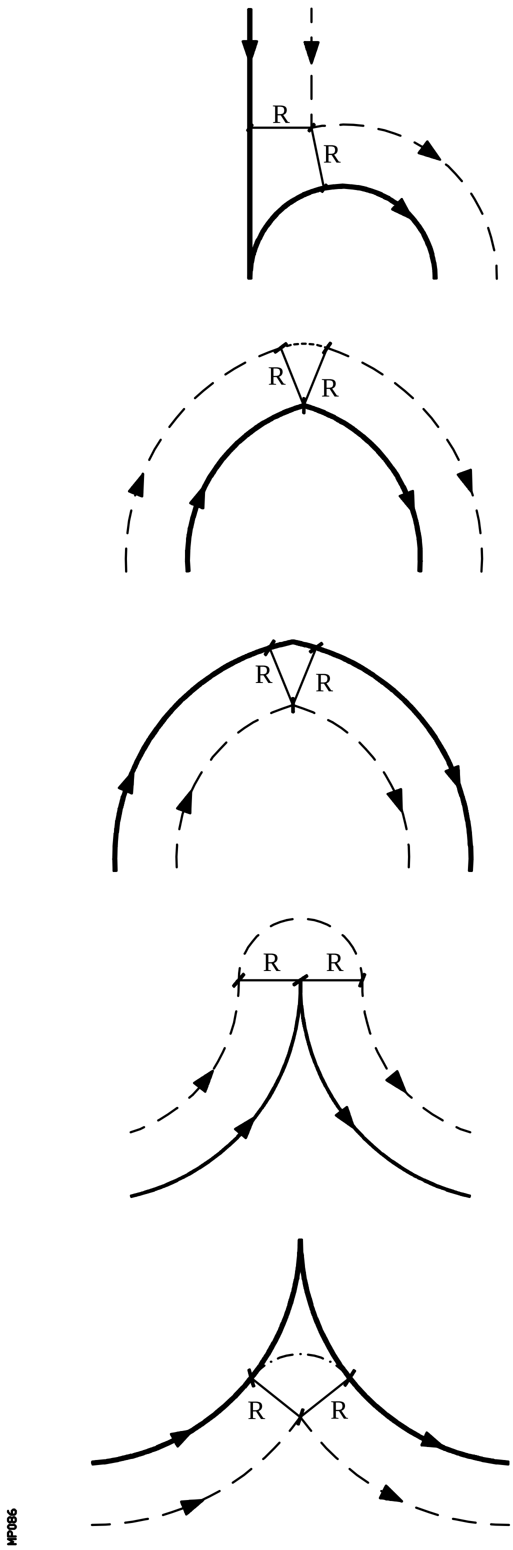
The diagrams (below) show the different paths followed by a tool controlled by a programmed CNC with tool radius compensation.

The programmed path is represented by a solid line and the compensated path by a dotted line.





MP085


R
R
R
R
R
R
R
R
R
R
MP086

The CNC reads up to 20 blocks ahead of the one it is executing, with the aim of calculating in advance the path to be followed.

When the CNC works with compensation it needs to know the next programmed movement to calculate the path to be followed. For this reason, no more than 17 consecutive blocks can be programmed without movement.

8.1.3 CANCELLING TOOL RADIUS COMPENSATION

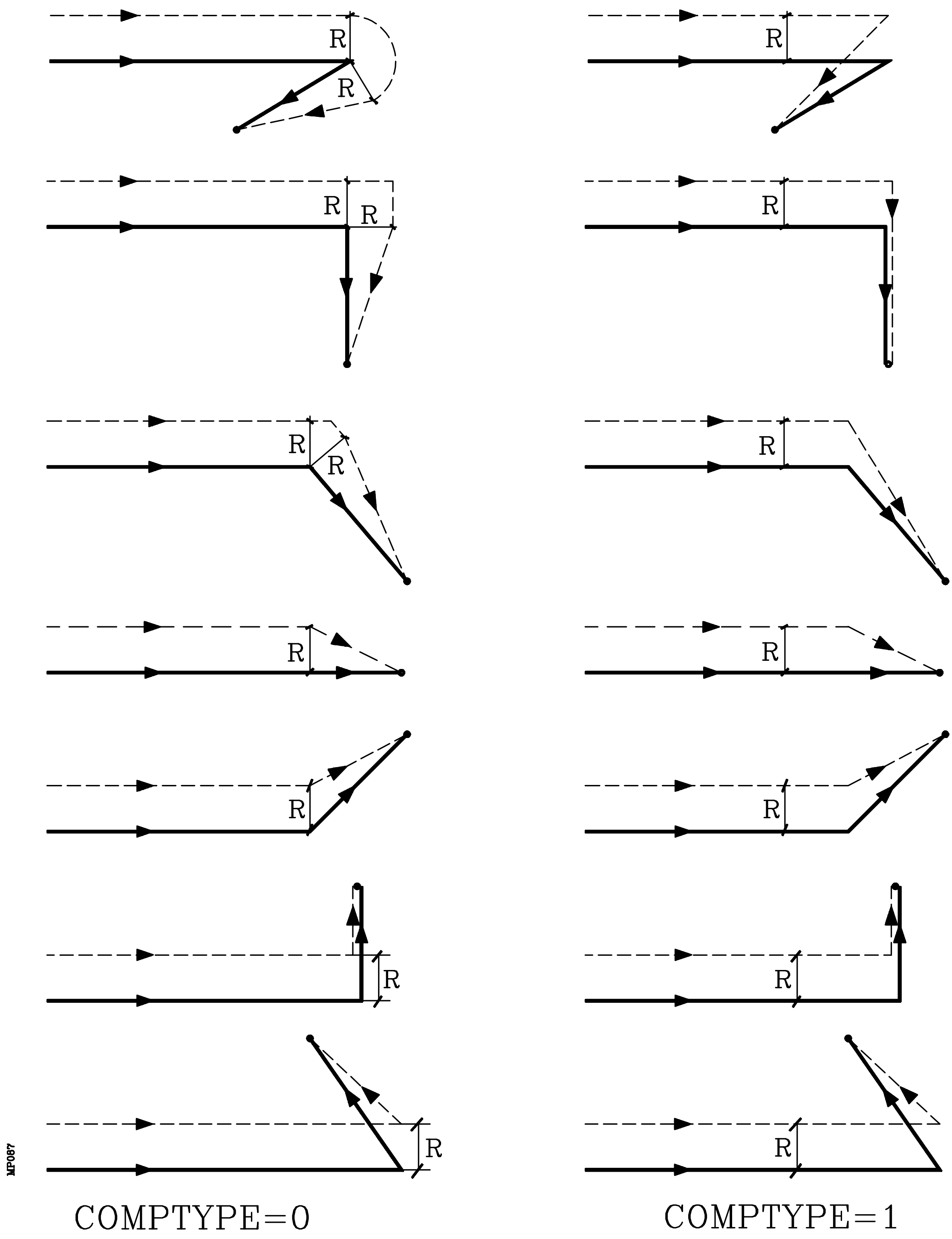
Tool radius compensation is cancelled by using function **G40**.

It should be remembered that cancelling radius compensation (G40) can only be done in a block in which a straight-line movement is programmed (G00 or G01).

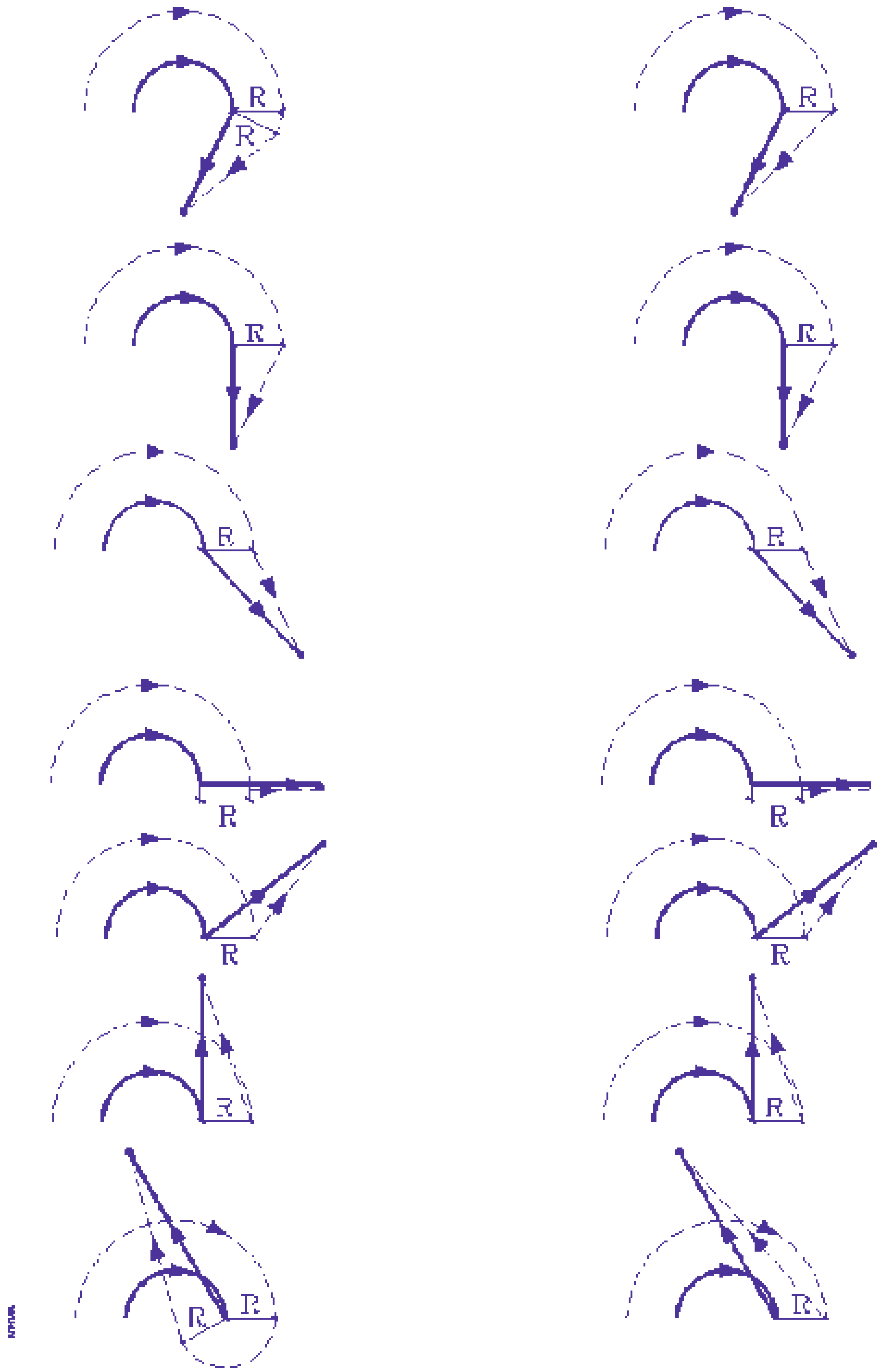
If G40 is programmed while functions G02 or G03 are active, the CNC displays the corresponding error message.

The following pages show different cases of cancelling tool radius compensation, in which the programmed path is represented by a solid line and the compensated path with a dotted line.

STRAIGHT-STRAIGHT path

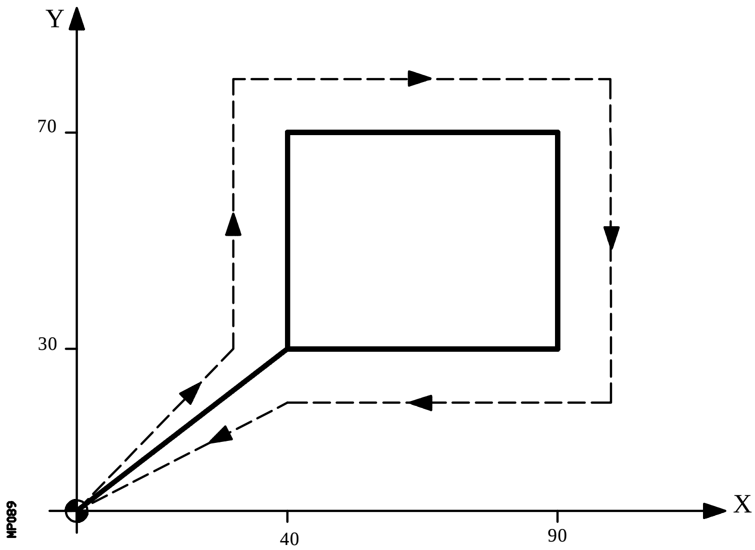


CURVED-STRAIGHT path



COMPTYPE=0 COMPTYPE=1

Example of machining with radius compensation

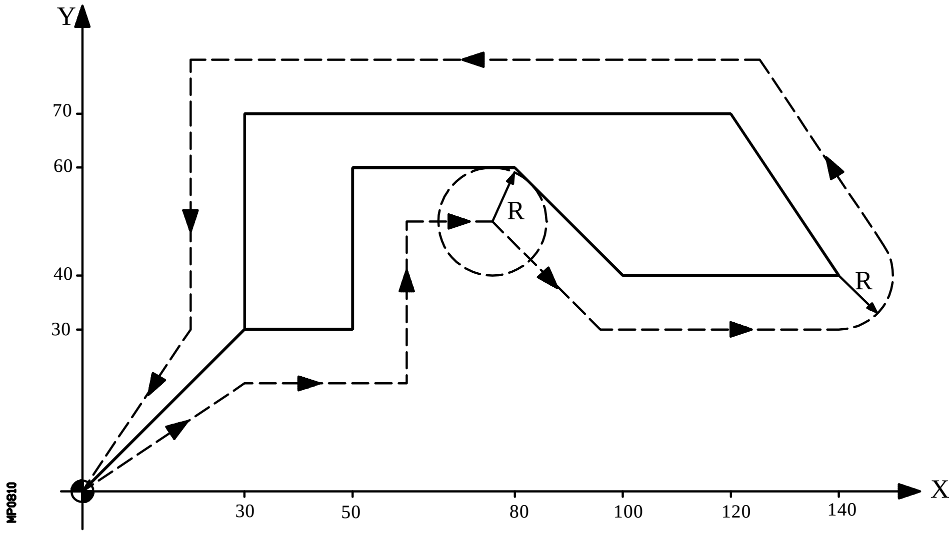


The programmed path is represented by a solid line and the compensation path by a dotted line.

Tool radius	: 10mm.
Tool number	: T1
Tool offset number	: D1

G92		X0	Y0	Z0	; position coordinate preset
G90	G17	S0.5	**T1 D1**	M03	; tool, tool offset, spindle start at S100
G41	G01	X40	Y30	F125	; activate compensation
			Y70		
		X90			
			Y30		
		X40			
G40	G01	X0	Y0		; cancel compensation
M30					

Example of machining with radius compensation:

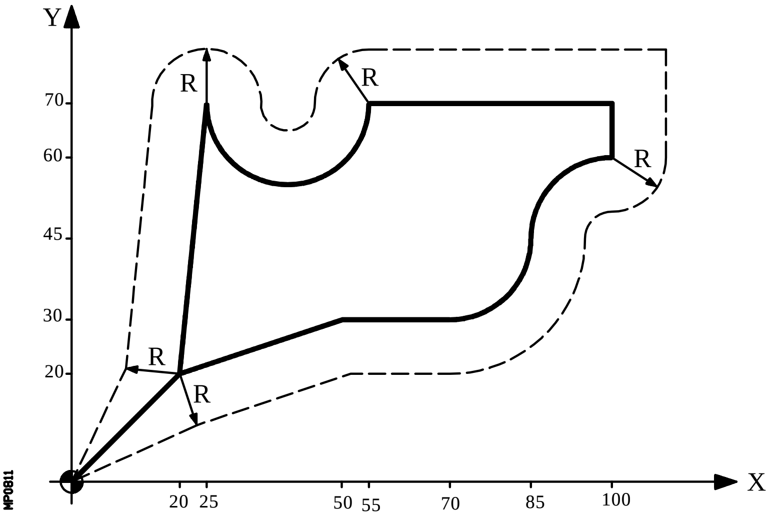


The programmed path is represented by a solid line and the compensation path by a dotted line.

Tool radius : 10mm.
Tool number : T1
Tool offset number : D1

```
G92  X0   Y0   Z0                    ; coordinate preset
G90  G17  G01  F150 S100 T1 D1 M03   ; tool, tool offset, spindle,..
G42  X30  Y30                        ; activate compensation
     X50
          Y60
     X80
     X100 Y40
     X140
     X120 Y70
     X30
          Y30
G40  G00  X0   Y0                    ; cancel compensation
M30
```

Example of machining with radius compensation :



The programmed path is represented by a solid line and the compensation path by a dotted line.

```
Tool radius          : 10mm.
Tool number          : T1
Tool offset number   : D1

G92  X0   Y0   Z0                    ;  coordinate preset
G90  G17  G01  F150 S100  T1 D1 M03  ;  tool, tool offset, spindle,..
G42  X20  Y20                        ;  activate compensation
     X50  Y30
     X70
G03  X85  Y45  I0   J15
G02  X100 Y60  I15  J0
G01       Y70
     X55
G02  X25  Y70  I-15 J0
G01  X20  Y20
G40  G00  X0   Y0   M5               ;  cancel compensation
M30
```

8.2 TOOL LENGTH COMPENSATION (G43, G44, G15)

With this function it is possible to compensate possible differences in length between the programmed tool and the tool being used.

The tool length compensation is applied on to the axis indicated by function G15 or, in its absence, to the axis perpendicular to the main plane.

If G17, tool length compensation on the Z axis.
If G18, tool length compensation on the Y axis
If G19, tool length compensation on the X axis.

Whenever one of functions G17, G18 or G19 is programmed, the CNC assumes as new longitudinal axis (upon which tool length compensation will be applied) the one perpendicular to the selected plane.

On the other hand, if function G15 is executed while functions G17, G18 or G19 are active, the new longitudinal axis (selected with G15) will replace the previous one.

The function codes used in length compensation are as follows:

G43 Activate tool length compensation.
G44 Cancelling tool length compensation.

Function G43 only indicates that a longitudinal compensation is to be applied. The CNC starts applying it when the longitudinal (perpendicular) axis starts moving.

Example:	G92 X0 Y0 Z50	; Preset
	G90 G17 G01 F150 S100 **T1** **D1** M03	; Tool, Tool offset, etc.
	G43 X20 Y20	; Selects compensation
	X70	
	Z30	; **Applies compensation**

When G43 is programmed, the CNC compensates the length in accordance with the value of the tool offset selected with code **D**, or (in its absence) the tool offset shown in the tool table for the selected tool (**T**).

Tool values **R, L, I, K** must be stored in the tool offset table before starting machining, or must be loaded at the beginning of the program via assignments to variables **TOR, TOL, TOI, TOK.**

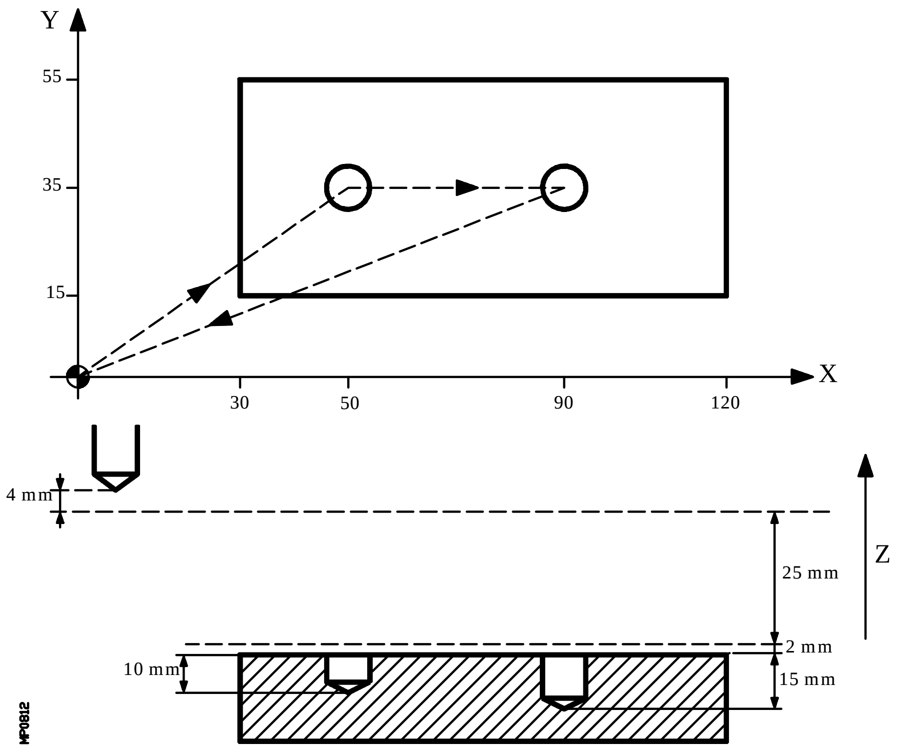
In the event of no tool offset being selected, the CNC takes D0 with values R0 L0 I0 K0.

Function G43 is modal and can be canceled via G44 and G74 (home search). If general machine parameter "ILCOMP=0", it is also canceled on power-up, after executing M02, M30 or after EMERGENCY or RESET.

G53 (programming with respect to machine zero) temporarily cancels G43 only while executing a block which contains a G53.

Length compensation can be used together with canned cycles, although here care should be taken to apply this compensation before starting the cycle.

Example of machining with length compensation :



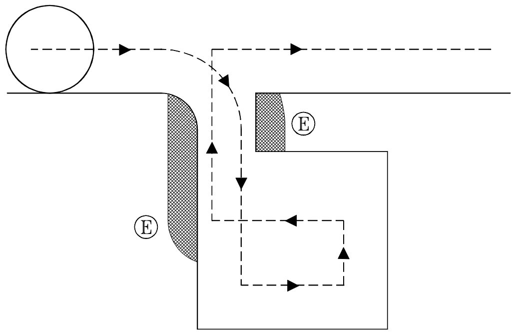
It is assumed that the tool used is 4mm shorter than the programmed one.

```
Tool length          : -4mm.
Tool number          : T1
Tool offset number   : D1

G92  X0   Y0   Z0                    ; coordinate preset
G91  G00  G05  X50  Y35  S500 M03
G43            Z-25          T1 D1   ; activate compensation
G01  G07       Z-12 F100
G00            Z12
               X40
G01            Z-17
G00  G05  G44  Z42  M05              ; cancel compensation
G90  G07       X0   Y0
M30
```

8.3 COLLISION DETECTION (G41 N, G42 N)

With this option, it is possible to check in advance the blocks to be executed in order to detect loops (a profile intersecting itself) or collisions in the programmed profile. The operator may set the number of blocks to be analyzed (up to 50 blocks).

The example shows the machining errors (E) due to a collision in the programmed profile. This type of errors may be prevented by using collision detection.



When detecting a loop or a collision, the blocks that cause it will not be executed and a warning will appear for each loop or collision eliminated.

Possible cases: a step on a straight path, a step on a circular path whose tool radius compensation is too large.

The information contained in the eliminated blocks, not being the movement in the active plane, will be executed (including the movement of other axes).

Block detection is set and activated by the tool radius compensation functions G41 and G42 with a new parameter: N (G41 N and G42 N) to turn the feature on and set the number of blocks to be analyzed.

Possible values between N3 and N50. Without "N" or with N0 , N1 and N2, it acts as in previous versions (backward compatible).

For CAD-generated programs consisting of numerous short blocks, the N values should be low (about 5) to avoid slowing block processing time too much.

When this function is active, the history of active G codes shows: G41 N or G42 N.

9. CANNED CYCLES

These canned cycles can be performed on any plane, the depth being along the axis selected as longitudinal via function G15 or, in its absence, along the axis perpendicular to this plane.

The CNC offers the following machining canned cycles :

G69 Complex deep hole drilling

G81 Drilling cycle

G82 Drilling cycle with dwell

G83 Simple deep hole drilling

G84 Tapping cycle

G85 Reaming cycle

G86 Boring cycle with withdrawal in G00

G87 Rectangular pocket milling cycle

G88 Circular pocket milling cycle

G89 Boring cycle with withdrawal in G01

It also offers the following functions that can be used with the machining canned cycles:

G79 Modification of the canned cycle parameters
G98 Return to the starting plane at the end of the canned cycle
G99 Return to the reference plane at the end of the canned cycle.

9.1 DEFINITION OF A CANNED CYCLE

A canned cycle is defined by the G function indicating the canned cycle and its corresponding parameters.

A canned cycle cannot be defined in a block which has nonlinear movements (G02, G03, G08, G09, G33 or G34).

Also, a canned cycle cannot be executed while function G02, G03, G33 or G34 is active. The CNC will issue the corresponding error message.

However, once a canned cycle has been defined in a block and following blocks, functions G02, G03, G08 or G09 can be programmed.

9.2 CANNED CYCLE AREA OF INFLUENCE

Once a canned cycle has been defined it remains active, and all blocks programmed after this block are under its influence while it is not cancelled.

In other words, every time a block is executed in which some axis movement has been programmed, the CNC will carry out (following the programmed movement) the machining operation which corresponds to the active canned cycle.

If, in a movement block within the area of influence of a canned cycle, the number of times a block is executed (repetitions) "N" is programmed at the end of the block, the CNC repeats the programmed positioning and the machining operation corresponding to the canned cycle the indicated number of times.

If a number of repetitions (times) “N0” is programmed, the machining operation corresponding to the **canned cycle will not be performed. The CNC will only carry out the programmed movement.**

If, within the area of influence of a canned cycle, there is a block which does not contain any movement, the machining operation corresponding to the defined canned cycle will not be performed, except in the calling block.

G81	Definition and execution of the canned cycle (drilling).
G90 G1 X100	The X axis moves to X100, where the hole is to be drilled.
G91 X10 N3	The CNC runs the following operation 3 times. * Incremental move to X10. * Runs the cycle defined above.
G91 X20 N0	Incremental move only to X20 (no drilling).

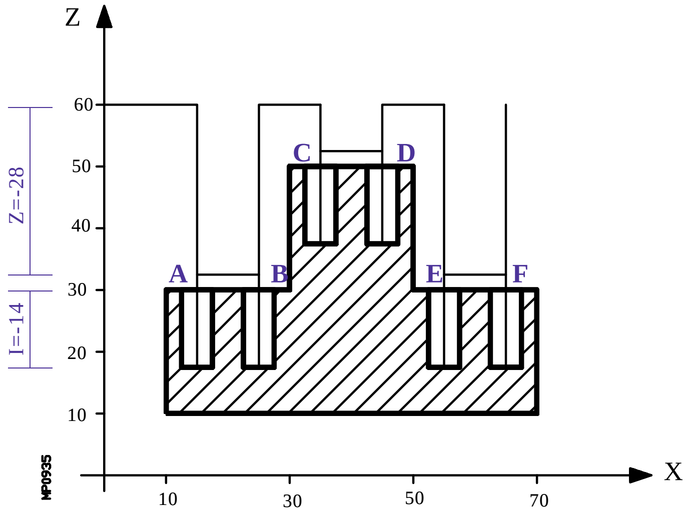
9.2.1. G79. MODIFICATION OF CANNED CYCLE PARAMETERS

The CNC allows one or several parameters of an active canned cycle to be modified by programming the G79 function, without any need for redefining the canned cycle. This is possible only inside the influence area of the canned cycle

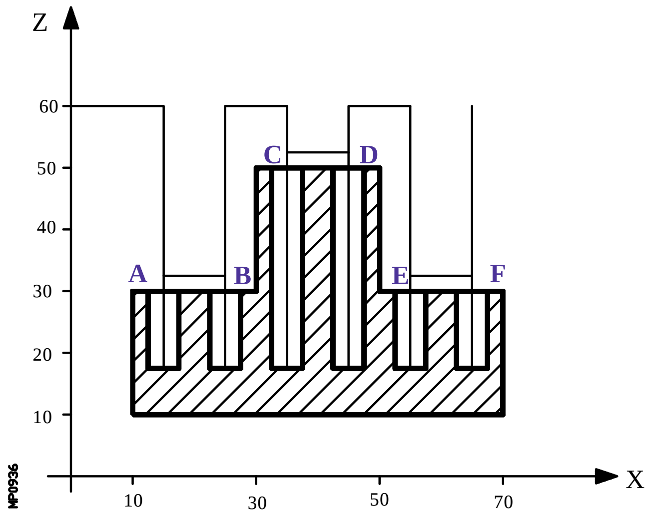
The CNC will continue to maintain the canned cycle active and will perform the following machinings of the canned cycle with the updated parameters.

The G79 function must be programmed alone in a block, and this block must not contain any more information.

Next 2 programming examples are shown assuming that the work plane is formed by the X and Y axes, and that the longitudinal axis (perpendicular) is the Z axis:




```
T1
M6
G00  G90  X0   Y0   Z60             ; Starting point
G81  G99  G91  X15  Y25  Z-28  I-14 ; Defines drilling cycle. Drills in A
G98  G90  X25                       ; Drills in B
G79  Z52                            ; Modifies reference plane and machining depth
G99  X35                            ; Drills in C
G98  X45                            ; Drills in D
G79  Z32                            ; Modifies reference plane and machining depth
G99  X55                            ; Drills in E
G98  X65                            ; Drills in F
M30
```




```
T1
M6
G00  G90  X0   Y0   Z60             ; Starting point
G81  G99  G90  X15  Y25  Z32   I18  ; Defines drilling cycle. Drills in A
G98  X25                            ; Drills in B
G79  Z52                            ; Modifies reference plane
G99  X35                            ; Drills in C
G98  X45                            ; Drills in D
G79  Z32                            ; Modifies reference plane
G99  X55                            ; Drills in E
G98  X65                            ; Drills in F
M30
```

9.3 CANNED CYCLE CANCELLATION

A canned cycle can be cancelled via :

- Function **G80**, which can be programmed in any block.
- After defining a new canned cycle. This will cancel and replace any other which may be active.
- After executing M02, M30, or after EMERGENCY or RESET.
- When searching home with function G74.
- Selecting a new work plane via functions G16, G17, G18, or G19.

9.4 GENERAL CONSIDERATIONS

1. A canned cycle can be defined at any point in a program, i.e., it can be defined both in the main program and in a subroutine.

2. Calls to subroutines can be made from a block within the influence of a canned cycle without implying the cancellation of the canned cycle.

3. The execution of a canned cycle will not alter the history of previous "G" functions.

4. Nor will the spindle turning direction be altered. A canned cycle can be entered with any turning direction (M03 or M04), leaving in the same direction in which the cycle was entered.

   Should a canned cycle be entered with the spindle stopped, it will start in a clockwise direction (M03), and maintain the same turning direction until the cycle is completed.

5. Should it be required to apply a scaling factor when working with canned cycles, it is advisable that this scale factor be common to all the axes involved.

6. The execution of a canned cycle cancels radius compensation (G41 and G42). It is equivalent to G40.

7. If tool length compensation (G43) is to be used, this function must be programmed in the same block or in the one before the definition of the canned cycle.

   The CNC applies the tool length compensation when the longitudinal (perpendicular) axis starts moving. Therefore, it is recommended to position the tool outside the canned cycle area when defining function G43 for the canned cycle.

8. The execution of any canned cycle will alter the global parameter **P299**.

9.5 MACHINING CANNED CYCLES

In all machining cycles there are three coordinates along the longitudinal axis to the work plane which, due to their importance, are discussed below:

Initial plane coordinate. This coordinate is given by the position which the tool occupies with respect to machine zero when the cycle is activated.

Reference plane coordinate. This is programmed in the cycle definition block and represents an approach coordinate to the part. It can be programmed in absolute coordinates or in incremental, in which case it will be referred to the initial plane.

Machining depth coordinate. This is programmed in the cycle definition block. It can be programmed in absolute coordinates or in incremental coordinates, in which case it will be referred to the reference plane.

There are two functions which allow to select the type of withdrawal of the longitudinal axis after machining.

G98 Selects the withdrawal of the tool as far as the initial plane, once the indicated machining has been done.

G99 Selects the withdrawal of the tool as far as the reference plane, once the indicated machining has been done.

These functions can be used both in the cycle definition block and the blocks which are under the influence of the canned cycle. The initial plane will always be the coordinate which the longitudinal axis had when the cycle was defined.

The structure of a canned cycle definition block is as follows:

G**	Starting point	Parameters	F S T D M	N****

It is possible to program the starting point in the canned cycle definition block (except the longitudinal axis), both in polar coordinates and in Cartesian coordinates.

After defining the point at which it is required to carry out the canned cycle (optional), the functions and parameters corresponding to the canned cycle will be defined, and afterwards, if required, the complementary functions F S T D M are programmed.

If a number of block repetitions is programmed, the CNC will repeat the programmed positionings and the canned cycle machining operations the indicated number of times.

When programming, at the end of the block, the number of times a block is to be executed "N", the CNC performs the programmed move and the machining operation corresponding to the active canned cycle the indicated number of times.

If "N0" is programmed, it will not execute the machining operation corresponding to the canned cycle. The CNC will only execute the programmed move.

The general operation for all the cycles is as follows:

* If the spindle was in operation previously, its turning direction is maintained. If it was not in movement, it will start by turning clockwise (M03).

* Positioning (if programmed) at the starting point for the programmed cycle.

* Rapid movement of the longitudinal axis from the initial plane to the reference plane.

* Execution of the programmed machining cycle.

* Rapid withdrawal of the longitudinal axis to the initial plane or reference plane, depending on whether G98 or G99 has been programmed.

A detailed explanation is given of each machining canned cycle, assuming in all cases that the work plane is made up of the X and Y axes and that the longitudinal axis is the Z axis.

Programming in other planes

The program format is always the same. It does not depend on the work plane. Parameters XY indicate the coordinate of the work plane (X: abscissa, Y: ordinate) and penetration takes place along the longitudinal axis.

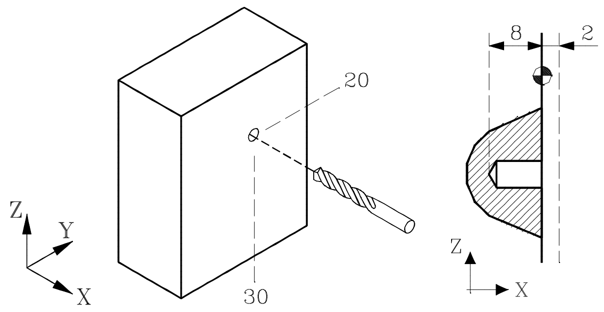
The following examples indicate how to drill holes in X and Y in both directions.

Function G81 defines the drilling canned cycle. The following parameters are defined:

- X coordinate of the point to be drilled along the abscissa axis
- Y coordinate of the point to be drilled along the ordinate axis
- I drilling depth.
- K dwell at the bottom.

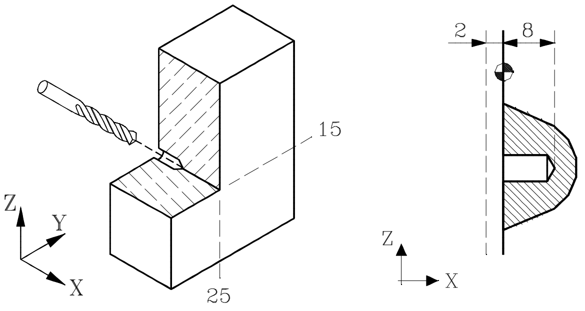
In the following examples, the part surface has a "0" coordinate. Make holes 8 mm deep and the reference coordinate is 2mm off the part surface.

Example 1:



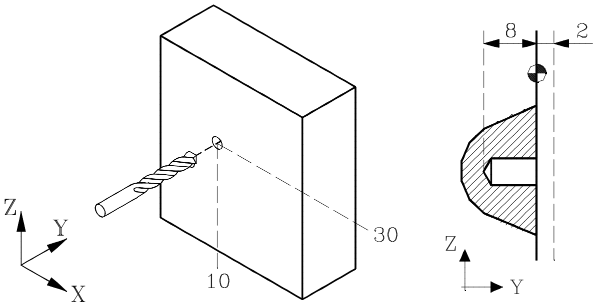

```
G19
G1 X25 F1000 S1000 M3
G81 X30 Y20 Z2 I-8 K1
```

Example 2:



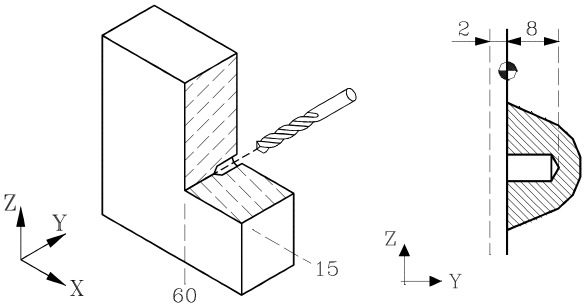
G19
G1 X-25 F1000 S1000 M3
G81 X25 Y15 Z-2 I8 K1

Example 3:



G18
G1 Y25 F1000 S1000 M3
G81 X30 Y10 Z2 I-8 K1

Example 4:



G18
G1 Y-25 F1000 S1000 M3
G81 X15 Y60 Z-2 I8 K1

9.5.1 G69. COMPLEX DEEP HOLE DRILLING CYCLE

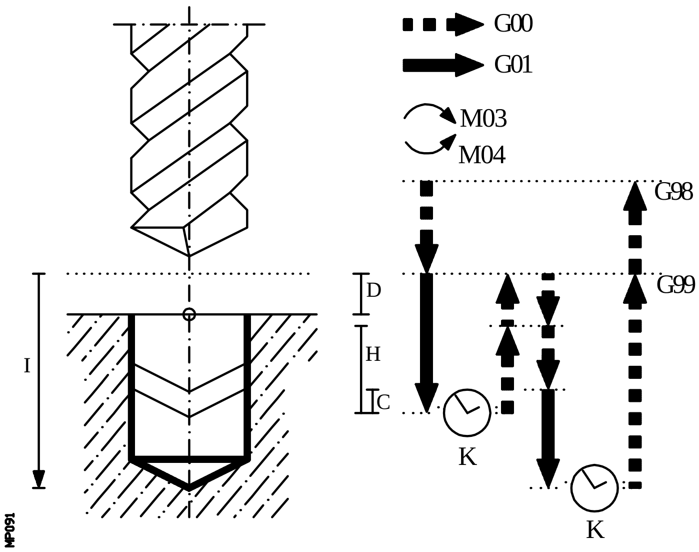
This cycle makes successive drilling steps until the final coordinate is reached.

The tool withdraws a fixed amount after each drilling operation, it being possible to select that every J drillings it withdraws to the reference plane.

A dwell can also be programmed after every drilling.

Working in Cartesian coordinates, the basic structure of the block is as follows:

G69 G98/G99 X Y Z I B C D H J K L R



G98 The tool withdraws to the Initial Plane, once the hole has been drilled.

G99 The tool withdraws to the Reference Plane, once the hole has been drilled.

XY±5.5 These are optional and define the movement of the axes of the main plane to position the tool at the machining point.

This point can be programmed in Cartesian coordinates or in polar coordinates, and the coordinates may be absolute or incremental, according to whether the machine is operating in G90 or G91.

Z±5.5 Defines the reference plane coordinate. It can be programmed in absolute coordinates or incremental coordinates, in which case it will be referred to the initial plane.

If this is not programmed, the CNC will take the position occupied by the tool at that moment as the reference plane.

I±5.5 Defines the total drilling depth. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the reference plane.

B5.5 Defines the drilling step in the axis longitudinal to the main plane.

C5.5 Defines to what distance from the previous drilling step, the longitudinal axis will travel in rapid feed (G00) in its approach to the part to make another drilling step.

If this is not programmed, the value of 1 mm (0.040 inch) will be taken. If programmed with a value of 0, the CNC will display the corresponding error.

D5.5 Defines the distance between the reference plane and the surface of the part where the drilling is to be done.

In the first drilling, this amount will be added to "B" drilling step. If it is not programmed, a value of 0 will be taken.

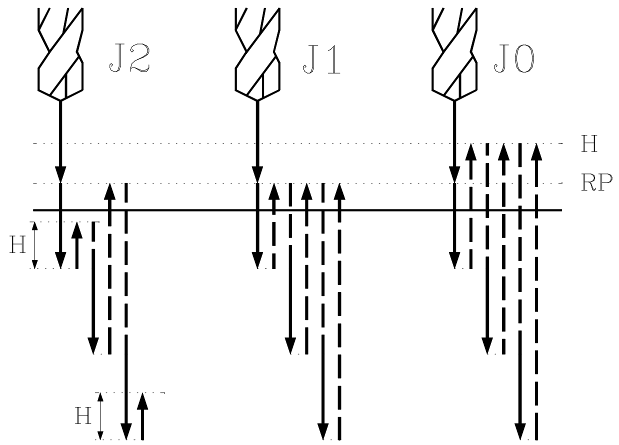
H±5.5 Distance or position the longitudinal axis returns to, in rapid (G00), after each drilling peck.

A "J" value other then "0" means the distance and if "J=0" indicates the relief position or absolute position it returns to.

When not programmed, the longitudinal axis will return to the reference plane.

J4 Defines after how many drilling pecks the tool returns to the reference plane in G00. A value between 0 and 9999 may be programmed.

When not programmed or programmed with a "0" value, it returns to the position indicated by H (relief position) after each drilling peck.



With "J >1" it will return the distance indicated by "H" and every "J" steps to the reference plane (RP).

With J1, it will return to the reference plane (RP) after each peck .

With J0, it will return to the relief position indicated by H.

K5 Defines the dwell time, in hundredths of a second, after each drilling step, until the withdrawal begins. Should this not be programmed, the CNC will take a value of K0.

L5.5 Defines the minimum value which the drilling step can acquire. This parameter is used with R values other than 1mm (0.040 inch). If this is not programmed or programmed with a value of 0, a value of 1 will be taken.

R5.5 Factor which reduces the drilling step "B". If this is not programmed or programmed with a value of 0, a value of 1 will be taken.

If R equals 1, all the drilling steps will be the same and the programmed value "B".

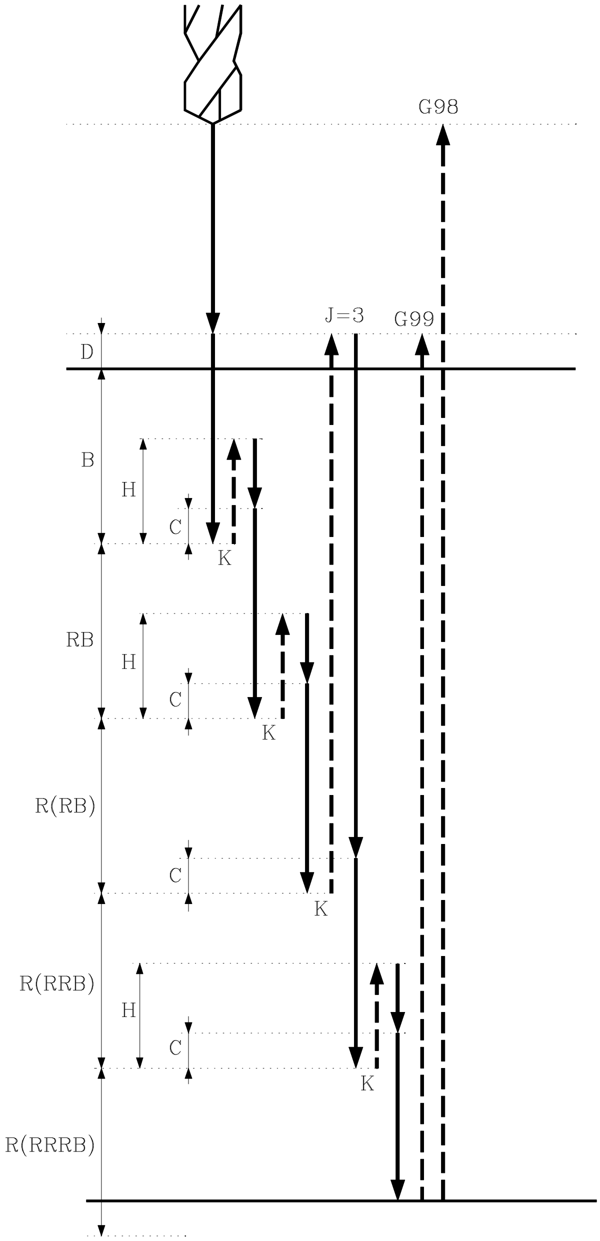
If R is not equal to 1, the first drilling step will be "B", the second, "R B", the third "R (RB)", and so on, i.e., after the second step, the new step will be the product of factor R by the previous step.

If R is selected with a value other than 1, the CNC will not allow smaller steps than that programmed in L.

Basic operation:

1. If the spindle was in operation previously, its turning direction is maintained. If it was not in movement, it will start by turning clockwise (M03).

2. Rapid movement of the longitudinal axis from the initial plane to the reference plane.



3. First drilling operation. Movement at working feedrate of the longitudinal axis to the programmed incremental depth in “B+D”.

   This movement will be carried out either in G07 or G50 depending on the value assigned to the longitudinal axis "INPOSW2(P51)"

   If P51 =0, in G7 (square corner) If P51=1, in G50 (controlled round corner).

4. Drilling loop. The following steps will be repeated until the machining depth coordinate programmed in I is reached.

   4.1. Dwell K in hundredths of a second, if this has been programmed.

   4.2. Withdrawal of the longitudinal axis in rapid (G00) as far as the reference plane, if the number of drillings programmed in J were made, otherwise it withdraws the distance programmed in “H”.

   4.3. Longitudinal axis approach in rapid (G00) as far as a distance “C” of the previous drilling step.

   4.4. Another drilling step. Movement of the longitudinal axis, at the working feedrate (G01) until the next incremental drilling according to “B and R”.

   This movement will be carried out in either in G07 or in G50 depending on the value assigned to the parameter of the longitudinal axis "INPOSW2(P51)".

   If P51=0 in G7 (square corner). If P51=1, in G50 (controlled round corner).

5. Dwell time K in hundredths of a second, if this has been programmed.

6. Withdrawal at rapid feedrate (G00) of the longitudinal axis to the initial or reference plane, depending on whether G98 or G99 has been programmed.

If a scaling factor is applied to this cycle, it should be borne in mind that this scaling factor will only affect the reference plane coordinates and drilling depth.

Therefore, and due to the fact that parameter “D” is not affected by the scaling factor, the surface coordinate of the part will not be proportional to the programmed cycle.

Programming example supposing that the work plane is formed by the X and Y axes, that the longitudinal axis is the Z axis and that the starting point is X0 Y0 Z0:

```
T1
M6
G0 G90 X0 Y0 Z0 ..................................................... ; Starting point
G69 G98 G91 X100 Y25 Z-98 I-52 B12 C2 D2
                H5 J2 K150 L3 R0.8 F100 S500 M8 ....... ; Canned cycle definition
G80 ............................................................................ ; Canned cycle cancellation
G90 X0 Y0 ................................................................. ; Positioning
M30 .......................................................................... ; End of program
```

9.5.2. G81 DRILLING CANNED CYCLE

This cycle drills at the point indicated until the final programmed coordinate is reached.

It is possible to program a dwell at the bottom of the drill hole.

Working in Cartesian coordinates, the basic structure of the block is as follows:

G81 G98/G99 X Y Z I K

G98 The tool withdraws to the Initial Plane, once the hole has been drilled.

G99 The tool withdraws to the Reference Plane, once the hole has been drilled.

XY±5.5 These are optional and define the movement of the axes of the main plane to position the tool at the machining point.

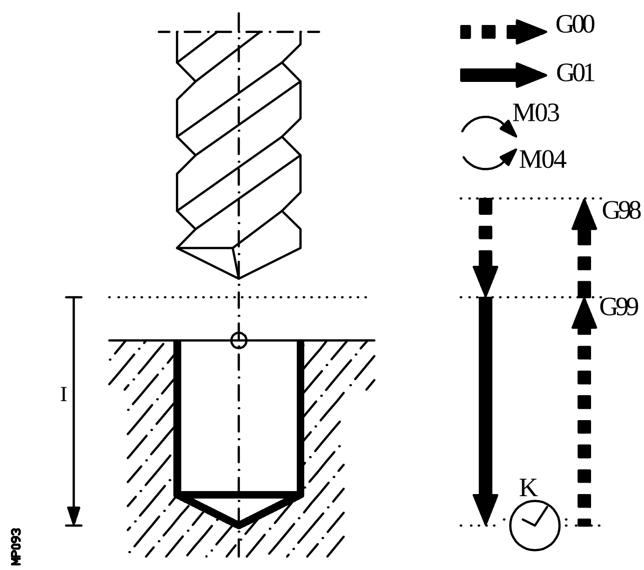
This point can be programmed in Cartesian coordinates or in polar coordinates, and the coordinates may be absolute or incremental, according to whether the machine is operating in G90 or G91.

Z±5.5 Defines the reference plane coordinate. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the initial plane.

If this is not programmed, the CNC will take the position occupied by the tool at that moment as the reference plane.

I±5.5 Defines drilling depth. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the reference plane.

K5 Defines the dwell time, in hundredths of a second, after each drilling step, until the withdrawal begins. Should this not be programmed, the CNC will take a value of K0.



Basic operation:

1. If the spindle was in operation previously, its turning direction is maintained. If it was not in movement, it will start by turning clockwise (M03).

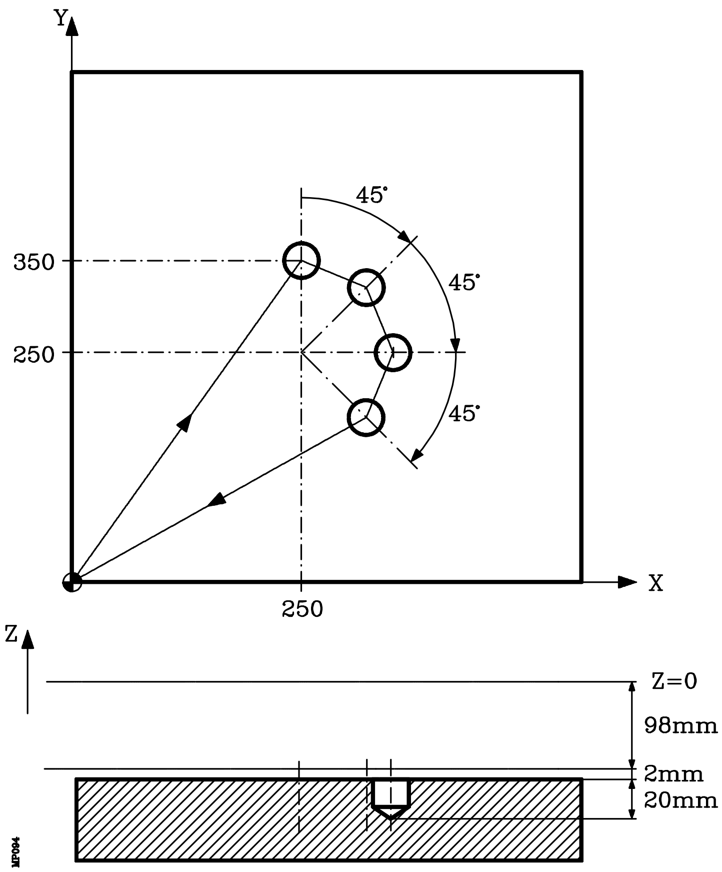
2. Rapid movement of the longitudinal axis from the initial plane to the reference plane.

3. The hole is drilled. Movement at working feedrate of the longitudinal axis to the programmed machining depth I.

4. Dwell time K in hundredths of a second, if this has been programmed.

5. Withdrawal at rapid feedrate (G00) of the longitudinal axis to the initial or reference plane, depending on whether G98 or G99 has been programmed.

Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is the Z axis and that the starting point is X0 Y0 Z0:



T1
M6
G0 G90 X0 Y0 Z0 .... ; Starting point
G81 G98 G00 G91 X250 Y350 Z-98 I-22 F100 S500 .... ; Positioning and definition of canned cycle
G93 I250 J250 .... ; Sets polar coordinate origin
Q-45 N3 .... ; Turn and canned cycle, 3 times
G80 .... ; Cancels canned cycle
G90 X0 Y0 .... ; Positioning
M30 .... ; End of program

9.5.3. G82. DRILLING CANNED CYCLE WITH DWELL

This cycle drills at the point indicated until the final programmed coordinate is reached.

Then it executes a dwell at the bottom of the drill hole.

Working in Cartesian coordinates, the basic structure of the block is as follows:

G82 G98/G99 X Y Z I K

G98 The tool withdraws to the Initial Plane, once the hole has been drilled.

G99 The tool withdraws to the Reference Plane, once the hole has been drilled.

XY±5.5 These are optional and define the movement of the axes of the main plane to position the tool at the machining point.

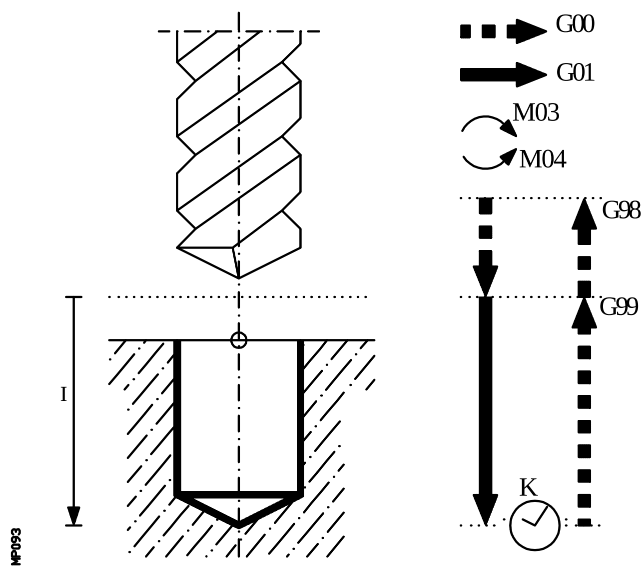
This point can be programmed in Cartesian coordinates or in polar coordinates, and the coordinates may be absolute or incremental, according to whether the machine is operating in G90 or G91.

Z±5.5 Defines the reference plane coordinate. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the initial plane.

If this is not programmed, the CNC will take the position occupied by the tool at that moment as the reference plane.

I±5.5 Defines drilling depth. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the reference plane.

K5 Defines the dwell time, in hundredths of a second, after each drilling step, until the withdrawal begins. Should this not be programmed, the CNC will take a value of K0.



Basic operation:

1. If the spindle was in operation previously, its turning direction is maintained. If it was not in movement, it will start by turning clockwise (M03).

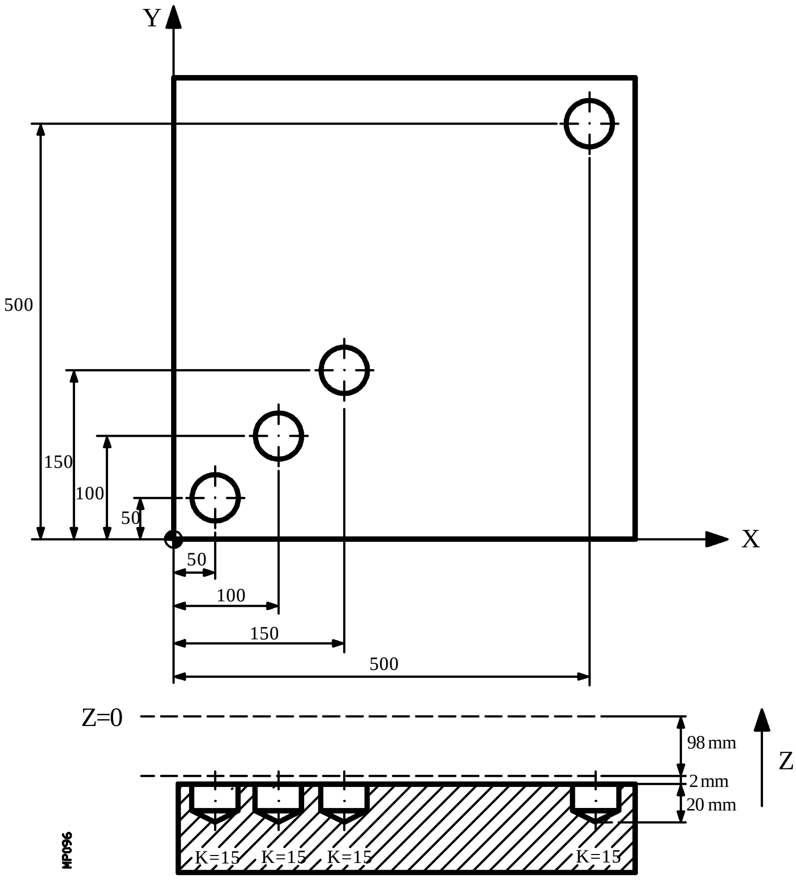
2. Rapid movement of the longitudinal axis from the initial plane to the reference plane.

3. The hole is drilled. Movement at working feedrate of the longitudinal axis to the bottom of the machined hole, programmed in I.

4. Dwell time K in hundredths of a second.

5. Withdrawal at rapid feedrate (G00) of the longitudinal axis to the initial or reference plane, according to whether G98 or G99 has been programmed.

Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is the Z axis and that the starting point is X0 Y0 Z0:



T1
M6
G0 G90 X0 Y0 Z0 .... ; Starting point
G82 G99 G00 G91 X50 Y50 Z-98 I-22 K150 F100 S500 N3 .... ; 3 machining positions
G98 G90 G00 X500 Y500 .... ; Positioning and canned cycle
G80 .... ; Cancels canned cycle
G90 X0 Y0 .... ; Positioning
M30 .... ; End of program

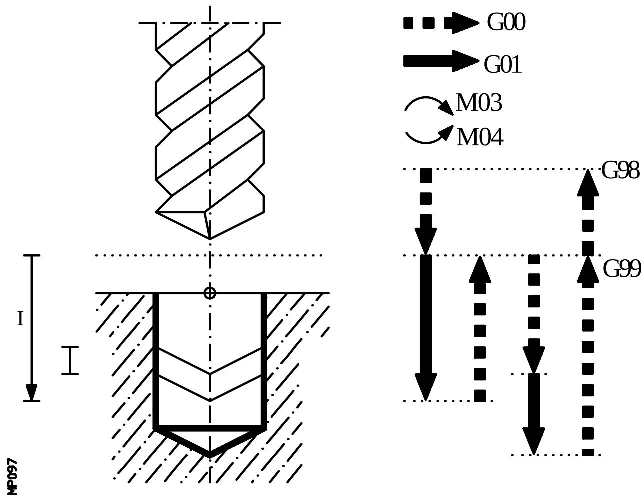
9.5.4. G83. SIMPLE DEEP HOLE DRILLING

This cycle performs successive drilling steps until the final programmed coordinate is reached.

The tool withdraws as far as the reference plane after each drilling step.

Working in cartesian coordinates, the basic structure of the block is as follows:

G83 G98/G99 X Y Z I J



G98 The tool withdraws to the Initial Plane, once the hole has been drilled.

G99 The tool withdraws to the Reference Plane, once the hole has been drilled.

XY±5.5 These are optional and define the movement of the axes of the main plane to position the tool at the machining point.

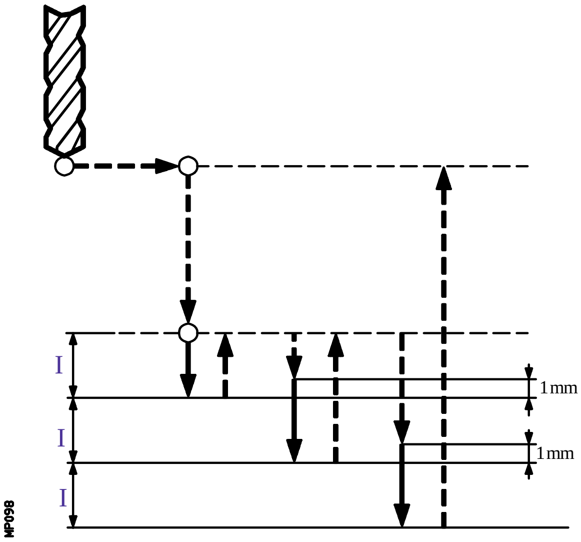
This point can be programmed in cartesian coordinates or in polar coordinates, and the coordinates may be absolute or incremental, according to whether the machine is operating in G90 or G91.

Z±5.5 Defines the reference plane coordinate. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the initial plane.

If this is not programmed, the CNC will take the position occupied by the tool at that moment as the reference plane.

I±5.5 Defines the value of each drilling step according to the axis longitudinal to the main plane.

J4 Defines the number of steps which the drill is to make. This can be programmed with a value between 1 and 9999.



Basic operation

1. If the spindle was in operation previously, its turning direction is maintained. If it was not in movement, it will start by turning clockwise (M03).

2. Rapid movement of the longitudinal axis from the initial plane to the reference plane.

3. First drilling. Movement at working feedrate of the longitudinal axis to the programmed incremental depth in “I”.

   This movement will be carried out either in G07 or G50 depending on the value assigned to the longitudinal axis "INPOSW2(P51)"

   If P51 =0, in G7 (square corner) otherwise, in G50 (controlled round corner).

4. Drilling loop. The following steps will be repeated “J-1” times as in the previous step the first programmed drilling was done.

   4.1. Withdrawal of the longitudinal axis in rapid (G00) to the reference plane.

   4.2. Longitudinal axis approach in rapid (G00):

   If INPOSW2=0 up to 1mm from the previous drilling peck.
   Otherwise, up to "INPOSW2 +0.02 MM of the previous drilling peck.

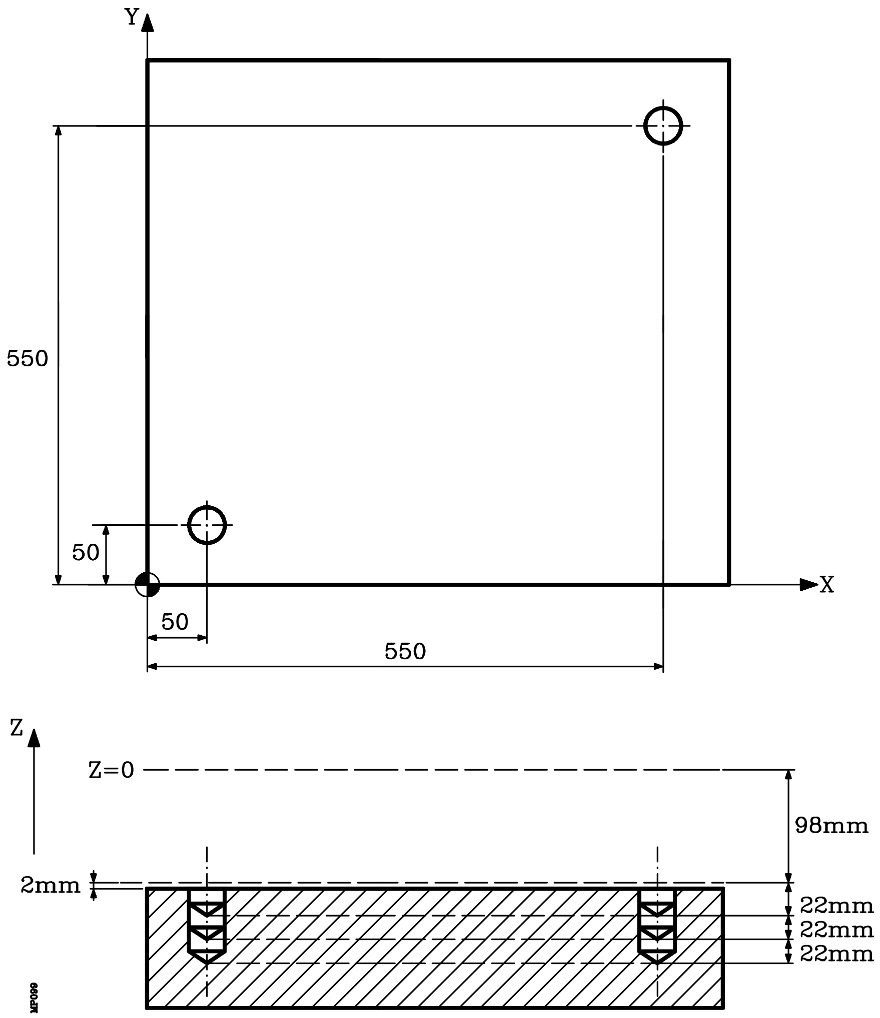
   4.3. Another drilling step. Movement of the longitudinal axis, at working feedrate (G01) the incremental depth programmed in “I”.

   If INPOSW2= 0 in G7 Otherwise, in G50

5. Withdrawal at rapid feedrate (G00) of the longitudinal axis to the initial or reference plane, depending on whether G98 or G99 has been programmed.

If a scaling factor is applied to this cycle, drilling will be performed proportional to that programmed, with the same step "I" programmed, but varying the number of steps "J".

Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is the Z axis and that the starting point is X0 Y0 Z0:



T1
M6
G0 G90 X0 Y0 Z0 .... ; Starting point
G83 G99 G00 G90 X50 Y50 Z-98 I-22 J3 F100 S500 M4 .... ; Positioning and canned cycle setting
G98 G00 G91 X500 Y500 .... ; Positioning and canned cycle
G80 .... ; Cancels canned cycle
G90 X0 Y0 .... ; Positioning
M30 .... ; End of program

9.5.5. G84. TAPPING CANNED CYCLE

This cycle taps at the point indicated until the final programmed coordinate is reached. The general logic output "**TAPPING**" (M5517) will stay active during this cycle.

Due to the fact that the tapping tool turns in two directions (one when tapping and the other when withdrawing from the thread), by means of the machine parameter of the spindle "SREVM05" it is possible to select whether the change in turning direction is made with the intermediate spindle stop, or directly.

General machine parameter "STOPAP(P116)" indicates whether general inputs /STOP, /FEEDHOL and /XFERINH are enabled or not while executing function G84.

It is possible to program a dwell before each reversal of the spindle turning direction, i.e., at the bottom of the thread hole and when returning to the reference plane.

Working in cartesian coordinates, the basic structure of the block is as follows:

G84 G98/G99 X Y Z I K R J

G98 The tool withdraws to the Initial Plane, once the hole has been tapped.

G99 The tool withdraws to the Reference Plane, once the hole has been tapped.

XY±5.5 These are optional and define the movement of the axes of the main plane to position the tool at the machining point.

This point can be programmed in cartesian coordinates or in polar coordinates, and the coordinates may be absolute or incremental, according to whether the machine is operating in G90 or G91.

Z±5.5 Defines the reference plane coordinate. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the initial plane.

If this is not programmed, the CNC will take the position occupied by the tool at that moment as the reference plane.

I±5.5 Defines tapping depth. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the reference plane.

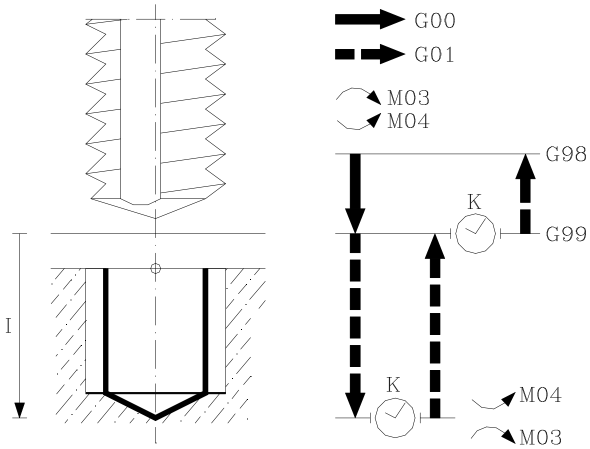
K5 Defines the dwell time, in hundredths of a second, after each drilling step, until the withdrawal begins. Should this not be programmed, the CNC will take a value of K0.

R Defines the type of tapping cycle to be performEd: normal if "R0" and rigid if "R1".

J 5.5 When rigid tapping, the returning feedrate will be J times the tapping feedrate. When not programmed or programmed J1, they will both be the same.

To perform a rigid tapping cycle, the spindle must be installed so it can work in closed loop; i.e. with encoder and servo drive.

During rigid tapping the CNC interpolates the longitudinal axis with the spindle rotation.



Basic operation

1. If the spindle was in operation previously, its turning direction is maintained. If it was not in movement, it will start by turning clockwise (M03).

2. Rapid movement of the longitudinal axis from the initial plane to the reference plane.

3. Movement of the longitudinal axis and at the working feedrate, to the bottom of the machined section, producing the threaded hole. The canned cycle will execute this movement and all later movements at 100% of F feedrate and the programmed S speed.

   If rigid tapping is selected (parameter R=1), the CNC will activate the general logic output “RIGID” (M5521) to indicate to the PLC that a rigid tapping block is being executed.

4. Spindle stop (M05). This will only be performed when the spindle meachine parameter **“SREVM05”** is selected and parameter "K" has a value other than "0"..

5. Dwell, if parameter “K” has been programmed.

6. Spindle turning direction reversal.

7. Withdrawal, at J times the working feedrate, of the longitudinal axis to the reference plane. Once this coordinate has been reached, the canned cycle will assume the selected FEEDRATE OVERRIDE and the SPINDLE OVERRIDE.

   If rigid tapping is selected (parameter R=1), the CNC will activate the general logic output “RIGID” (M5521) to indicate to the PLC that a rigid tapping block is being executed.

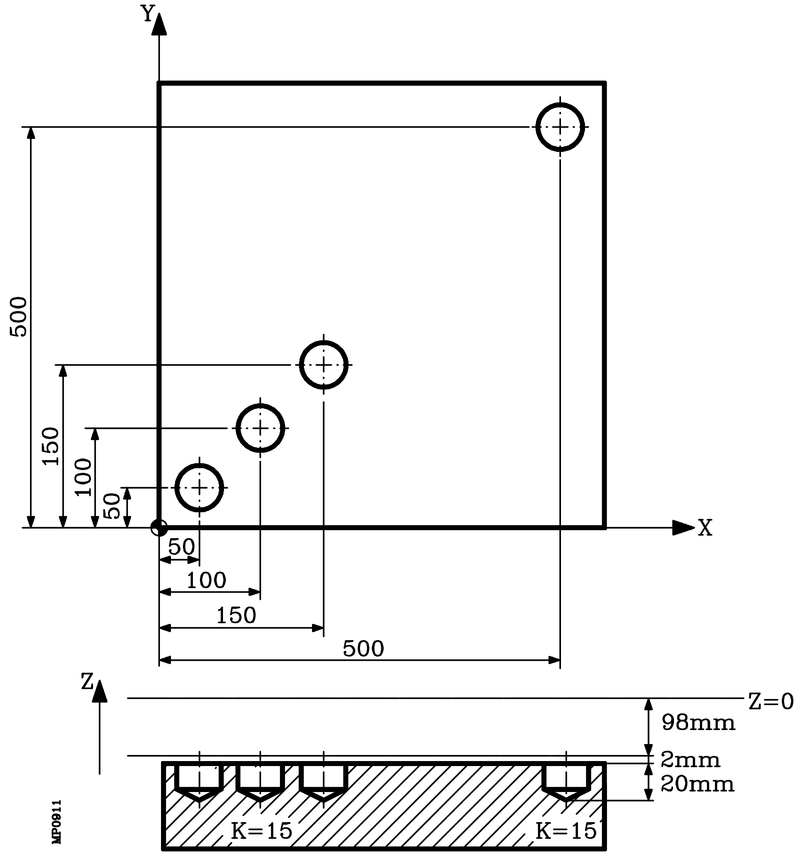
8. Spindle stop (M05). This will only be performed if the spindle meachine parameter **“SREVM05”** is selected.

9. Dwell, if parameter “K” has been programmed.

10. Spindle turning direction reversal.

11. Withdrawal, at rapid feedrate (G00), of the longitudinal axis as far as the initial plane if G98 has been programmed.

Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is the Z axis and that the starting point is X0 Y0 Z0:



T1
M6
G0 G90 X0 Y0 Z0 .... ; Starting point
G84 G99 G00 G91 X50 Y50 Z-98 I-22 K150 F350 S500 N3 .... ; 3 machining positions
G98 G00 G90 X500 Y500 .... ; Positioning and canned cycle
G80 .... ; Cancels canned cycle
G90 X0 Y0 .... ; Positioning
M30 .... ; End of program

9.5.6. G85. REAMING CYCLE

This cycle reams at the point indicated until the final programmed coordinate is reached.

It is possible to program a dwell at the bottom of the machined hole.

Working in cartesian coordinates, the basic structure of the block is as follows:

G85 G98/G99 X Y Z I K

G98 The tool withdraws to the Initial Plane, once the hole has been reamed.

G99 The tool withdraws to the Reference Plane, once the hole has been reamed.

XY±5.5 These are optional and define the movement of the axes of the main plane to position the tool at the machining point.

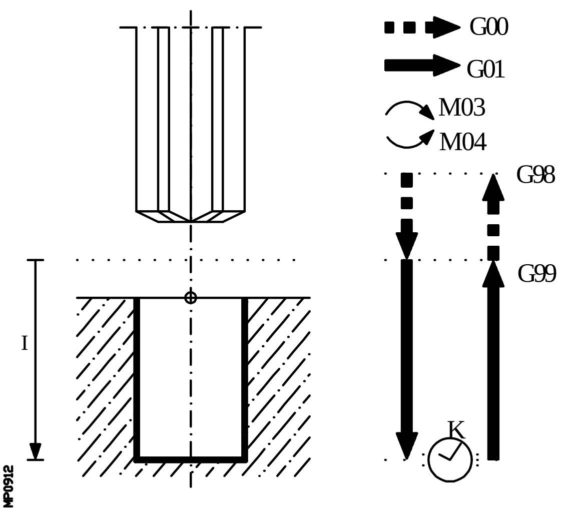
This point can be programmed in cartesian coordinates or in polar coordinates, and the coordinates may be absolute or incremental, according to whether the machine is operating in G90 or G91.

Z±5.5 Defines the reference plane coordinate. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the initial plane.

If this is not programmed, the CNC will take the position occupied by the tool at that moment as the reference plane.

I±5.5 Defines reaming depth. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the reference plane.

K5 Defines the dwell time, in hundredths of a second, after each drilling step, until the withdrawal begins. Should this not be programmed, the CNC will take a value of K0.



Basic operation

1. If the spindle was in operation previously, its turning direction is maintained. If it was not in movement, it will start by turning clockwise (M03).
2. Rapid movement of the longitudinal axis from the initial plane to the reference plane.
3. Movement at the working feedrate (G01) of the longitudinal axis to the bottom of the machined hole, and reaming.
4. Dwell, if parameter "K" has been programmed.
5. Withdrawal at working feedrate, of the longitudinal axis as far as the reference plane.
6. Withdrawal, at rapid feedrate (G00), of the longitudinal axis as far as the initial plane if G98 has been programmed.

Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is the Z axis and that the starting point is X0 Y0 Z0:

```
T1
M6
G0 G90 X0 Y0 Z0 ........................................................ ;Starting point
G85 G98 G91 X250 Y350 Z-98 I-22 F100 S500 ........... ;Canned cycle definition
G80 .................................................................... ;Canned cycle cancellation
G90 X0 Y0 ............................................................ ;Positioning
M30 .................................................................... ;End of program
```

9.5.7. G86. BORING CYCLE WITH WITHDRAWAL IN RAPID (G00)

This cycle bores at the point indicated until the final programmed coordinate is reached.

It is possible to program a dwell at the bottom of the machined hole.

Working in cartesian coordinates, the basic structure of the block is as follows:

G86 G98/G99 X Y Z I K

G98 The tool withdraws to the Initial Plane, once the hole has been bored.

G99 The tool withdraws to the Reference Plane, once the hole has been bored.

XY±5.5 These are optional and define the movement of the axes of the main plane to position the tool at the machining point.

This point can be programmed in cartesian coordinates or in polar coordinates, and the coordinates may be absolute or incremental, according to whether the machine is operating in G90 or G91.

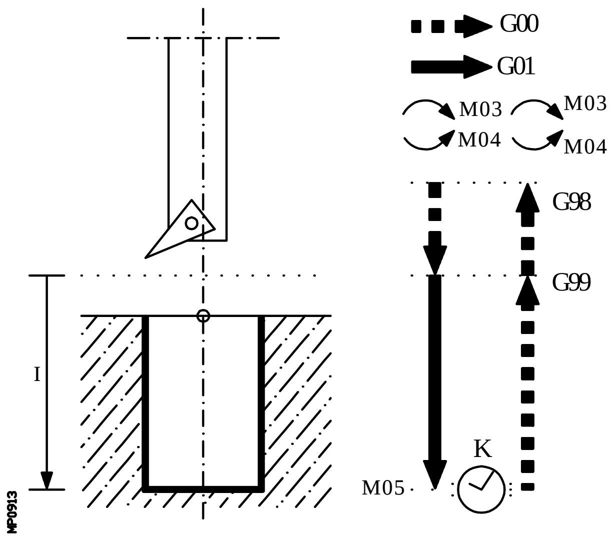
Z±5.5 Defines the reference plane coordinate. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the initial plane.

If this is not programmed, the CNC will take the position occupied by the tool at that moment as the reference plane.

I±5.5 Defines boring depth. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the reference plane.

K5 Defines the dwell time, in hundredths of a second, after each drilling step, until the withdrawal begins. Should this not be programmed, the CNC will take a value of K0.

Basic operation



1. If the spindle was in operation previously, its turning direction is maintained. If it was not in movement, it will start by turning clockwise (M03).

2. Rapid movement of the longitudinal axis from the initial plane to the reference plane.

3. Movement at the working feedrate (G01) of the longitudinal axis to the bottom of the machined hole, and boring.

4. Dwell, if parameter "K" has been programmed.

5. Spindle stop (M05).

6. Withdrawal, at rapid feedrate (G00), of the longitudinal axis as far as the initial plane or the reference plane, depending on whether G98 or G99 has been programmed.

7. When spindle withdrawal has been completed, it will start in the same direction in which it was turning before.

Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is the Z axis and that the starting point is X0 Y0 Z0:

```
T1
M6
G0 G90 X0 Y0 Z0 ........................................................ ;Starting point
G86 G98 G91 X250 Y350 Z-98 I-22 K20 F100 S500 ... ;Canned cycle definition
G80 ..................................................................... ;Canned cycle cancellation
G90 X0 Y0 ............................................................. ;Positioning
M30 ..................................................................... ;End of program
```

9.5.8. G87. RECTANGULAR POCKET CANNED CYCLE

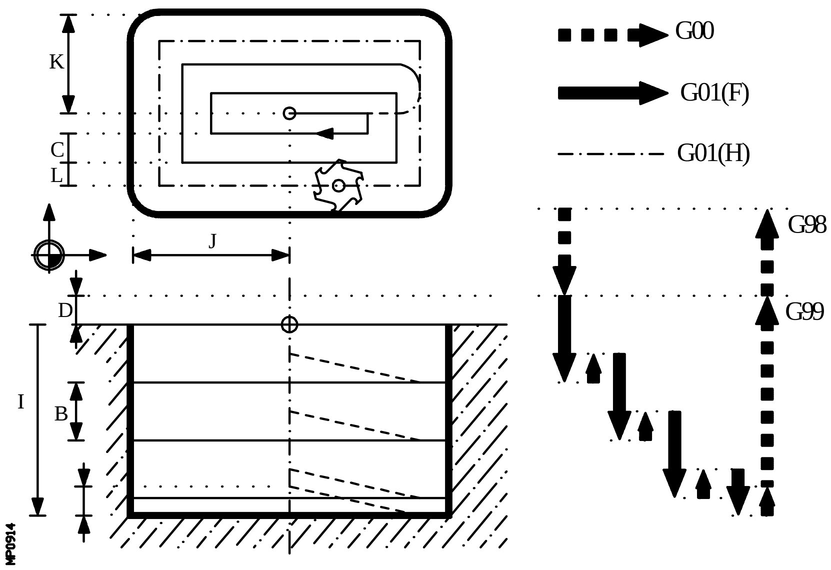
This cycle executes a rectangular pocket at the point indicated until the final programmed coordinate is reached.

It is possible to program, in addition to milling pass and feedrate, a final finishing step with its corresponding milling feedrate.

In order to obtain a good finish in the machining of the pocket walls, the CNC will apply a tangential entry and exit to the last milling step during each cutting operation.

Working in cartesian coordinates, the basic structure of the block is as follows:

G87 G98/G99 X Y Z I J K B C D H L V



G98 The tool withdraws to the Initial Plane, once the pocket has been made.

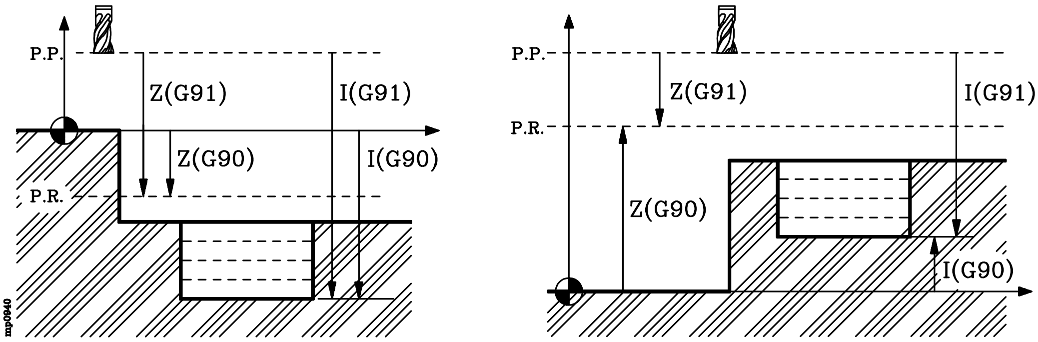
G99 The tool withdraws to the Reference Plane, once the pocket has been made.

XY±5.5 These are optional and define the movement of the axes of the main plane to position the tool at the machining point.

This point can be programmed in cartesian coordinates or in polar coordinates, and the coordinates may be absolute or incremental, according to whether the machine is operating in G90 or G91.

Z±5.5 Defines the reference plane coordinate.

When programmed in absolute coordinates, it will be referred to the part zero and when programmed in incremental coordinates, it will be referred to the starting plane (P.P.).

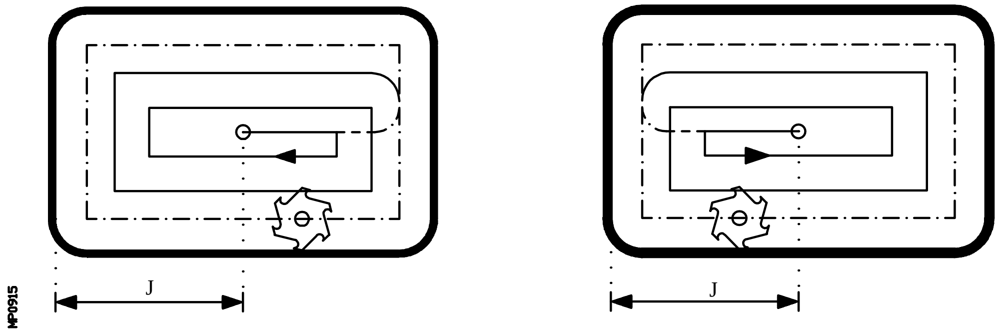


If this is not programmed, the CNC will take the position occupied by the tool at that moment as the reference plane. Thus, the starting plane (P.P.) and the reference plane (P.R.) wil be the same.

I±5.5 Defines machining depth.

When programmed in absolute coordinates, it will be referred to the part zero and when programmed in incremental coordinates, it will be referred to the starting plane (P.P.).

J±5.5 Defines the distance from the center to the edge of the pocket according to the abscissa axis. The sign indicates the pocket machining direction.



J with "+" sign J with "-" sign

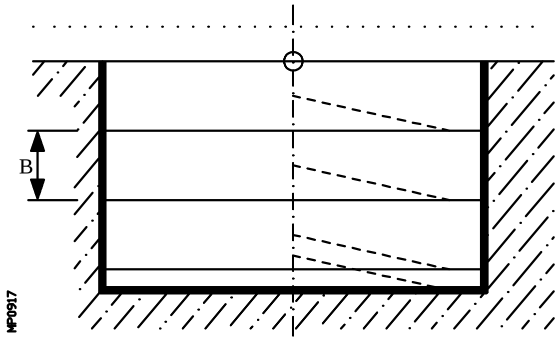
K5.5 Defines the distance from the center to the edge of the pocket according to the ordinate axis.

K

B±5.5 Defines the cutting depth according to the longitudinal axis.

- If this is programmed with a positive sign, the entire cycle will be executed with the same machining pass, this being equal to or less than that programmed.

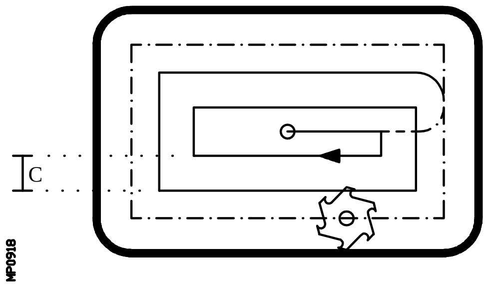
- If this is programmed with a negative sign, the entire pocket will be executed with the given pass, except for the last pass which will machine the rest.



C±5.5 Defines the milling pass along the main plane.

- If the value is positive, the entire cycle will be executed with the same milling step, this being equal to or less than that programmed.

- If the value is negative, the entire pocket will be executed with the given step, except for the last step which will machine whatever remains.



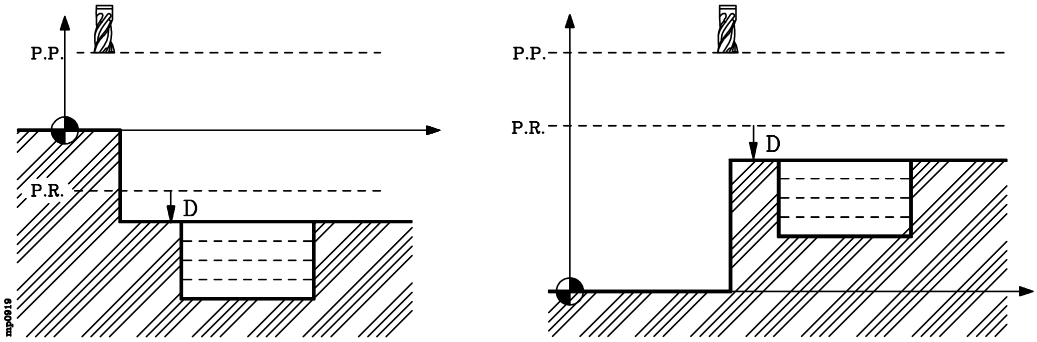
If this is not programmed, the CNC will assume 3/4 of the diameter of the diameter of the selected tool.

If programmed with a value greater than the tool diameter, the CNC will issue the corresponding error.

If programmed with a value of 0, the CNC will show the corresponding error.

D5.5 Defines the distance between the reference plane and the surface of the part where the pocket is to be made.

During the first deepening operation this amount will be added to incremental depth “B”. If this is not programmed, a value of 0 will be taken.

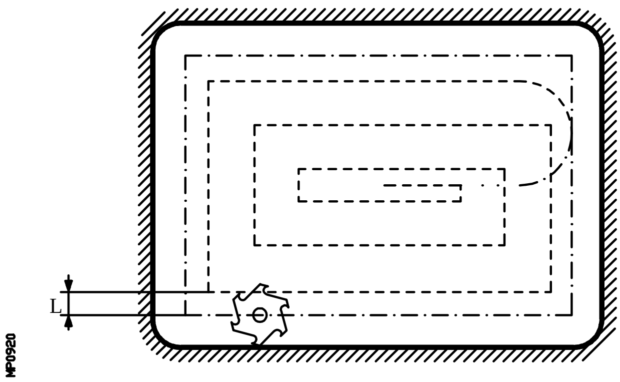


H.5.5 Defines the working feedrate during the finishing pass.

If this is not programmed or is programmed with a value of 0, the value of the working feedrate for machining will be taken.

L±5.5 Defines the value of the finishing pass, along the main plane.

- If the value is positive, the finishing pass is made on a square corner (G07).
- If the value is negative, the finishing pass is made on a rounded corner (G05).



If this is not programmed or is programmed with a value of 0 no finishing pass will be made.

V.5.5 Defines the tool penetrating feedrate.

If not programmed or programmed with a value of "0", the CNC will assume 50% of the feedrate in the plane (F).

Basic operation

1. If the spindle was in operation previously, its turning direction is maintained. If it was not in movement, it will start by turning clockwise (M03).

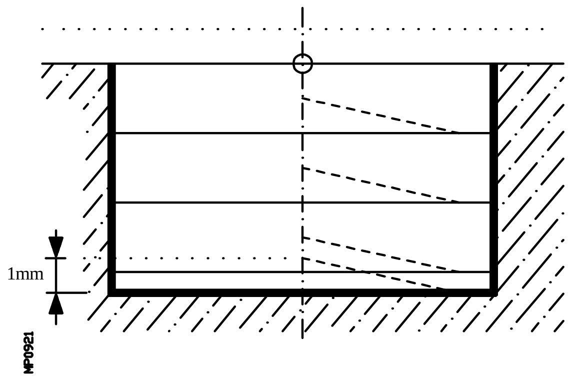
2. Rapid movement of the longitudinal axis from the initial plane to the reference plane.

3. First deepening operation. Movement of longitudinal axis at the feedrate indicated by "V" to the incremental depth programmed in “B+D”.

4. Milling at the working feedrate of the surface of the pocket in steps defined by means of “C” as far as a distance “L” (finishing pass) from the pocket wall.

5. Milling of the “L” finishing pass with the working feedrate defined in “H”.

6. Once the finishing pass has been completed, the tool withdraws at the rapid feedrate (G00) to the center of the pocket, the longitudinal axis being separated 1 mm (0.040 inch) from the machined surface.

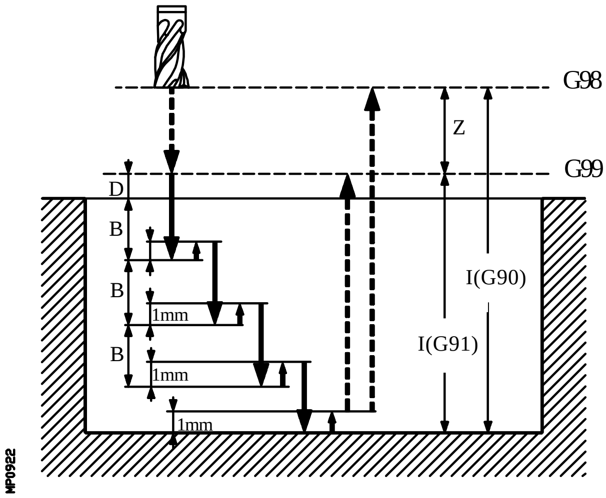


7. Further milling runs until the total depth of the pocket is reached.

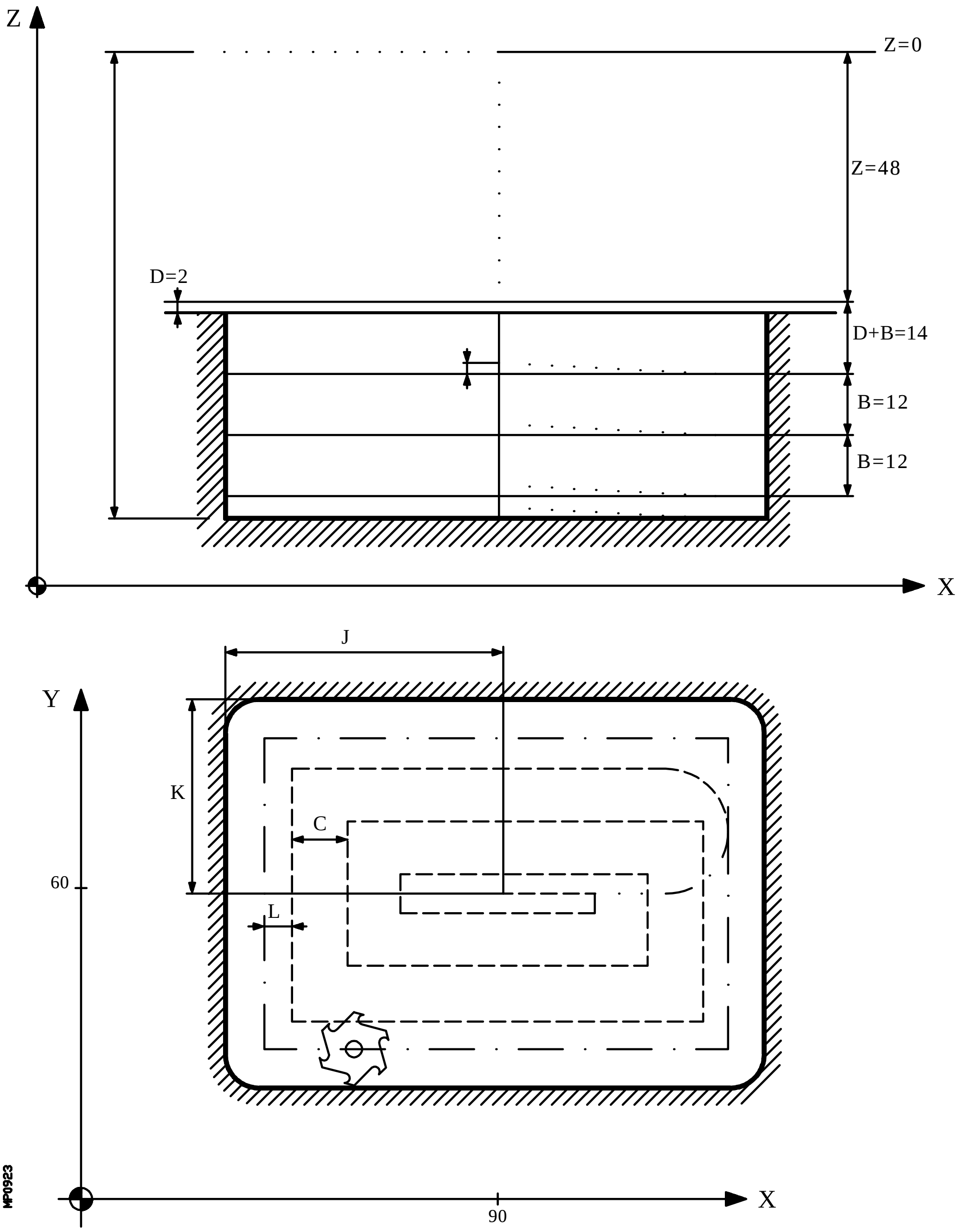
   - Movement of the longitudinal axis at the feedrate indicated by "V", up to a distance “B” from the previous surface.

   - Milling of a new surface following the steps indicated in paragraphs 4, 5 and 6.

8. Withdrawal at rapid feedrate (G00) of the longitudinal axis to the initial or reference plane, along depending on G98 or G99 has been programmed.

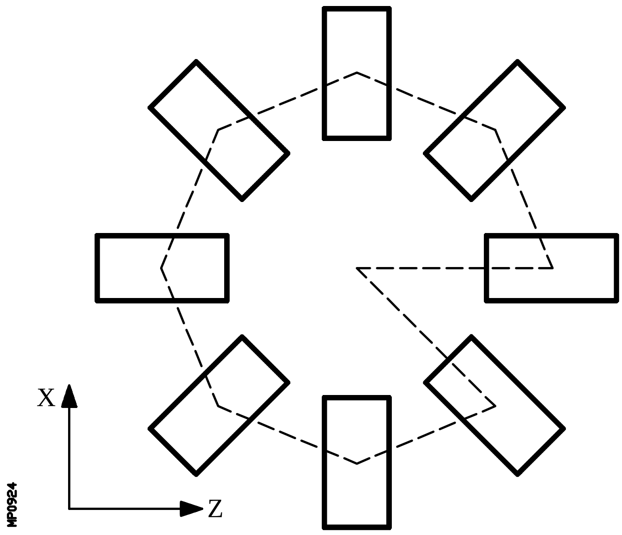


Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is the Z axis and that the starting point is X0 Y0 Z0:



(TOR1=6, TOT1=0)
T1 D1
M6
G0 G90 X0 Y0 Z0 .... ; Starting point
G87 G98 G00 G90 X90 Y60 Z-48 I-90 J52.5 K37.5 B12
C10 D2 H100 L5 V100 F300 S1000 T1 D1 M03 .... ; Canned cycle definition
G80 .... ;Cancels canned cycle
G90 X0 Y0 .... ; Positioning
M30 .... ; End of program

Programming example assuming that the starting point is X0 Y0 Z0.




```
    (TOR1=6, TOT1=0)
    T1 D1
    M6
    G0 G90 X0 Y0 Z0 ............................................ ; Starting point
    G18 ........................................................ ; Work plane
N10 G87 G98 G00 G90 X200 Y-48 Z0 I-90 J52.5 K37.5 B12
    C10 D2 H100 L5 V50 F300 .................................... ; Canned cycle definition
N20 G73 Q45 ................................................. ; Turn
    (RPT N10 N20) N7 ........................................... ; Repeat 7 times
    G80 ........................................................ ; Canned cycle Cancellation
    G90 X0 Y0 .................................................. ; Positioning
    M30 ........................................................ ; End of program
```

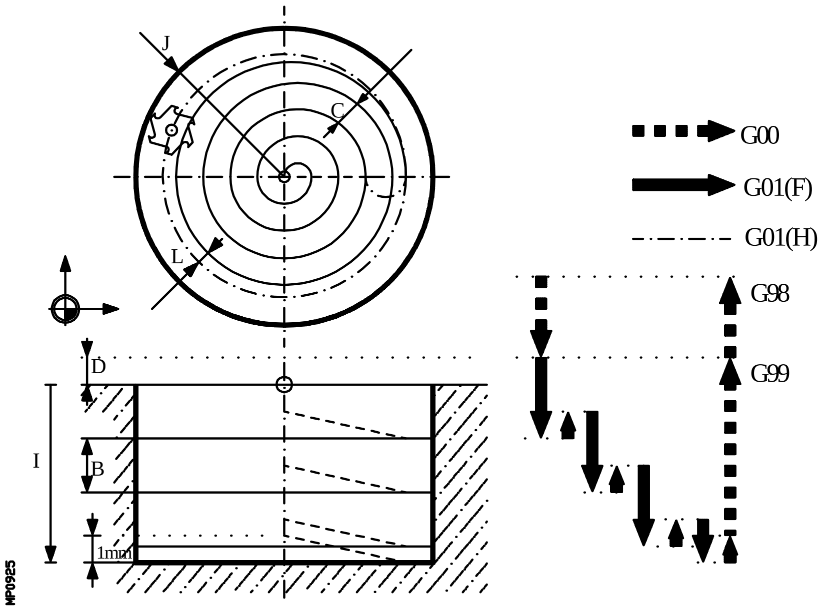
9.5.9. G88. CIRCULAR POCKET CANNED CYCLE

This cycle executes a circular pocket at the point indicated until the final programmed coordinate is reached.

It is possible to program, in addition to milling step and feedrate, a final finishing pass with its corresponding milling feedrate.

Working in cartesian coordinates, the basic structure of the block is as follows:

G88 G98/G99 X Y Z I J B C D H L V



G98 The tool withdraws to the Initial Plane, once the pocket has been made.

G99 The tool withdraws to the Reference Plane, once the pocket has been made.

XY±5.5 These are optional and define the movement of the axes of the main plane to position the tool at the machining point.

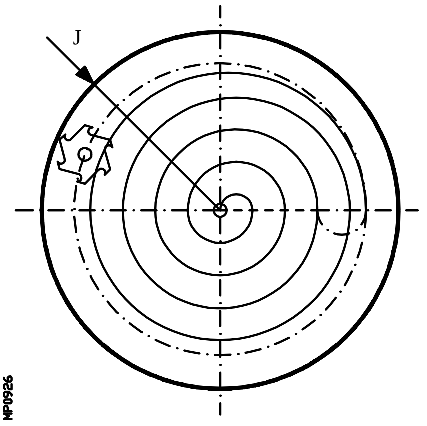
This point can be programmed in cartesian coordinates or in polar coordinates, and the coordinates may be absolute or incremental, depending on whether the machine is operating in G90 or G91.

Z±5.5 Defines the reference plane coordinate. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the initial plane.

If this is not programmed, the CNC will take the position occupied by the tool at that moment as the reference plane.

I±5.5 Defines machining depth. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the reference plane.

J±5.5 Defines the radius of the pocket. The sign indicates the pocket machining direction.



J with "+" sign J with "-" sign

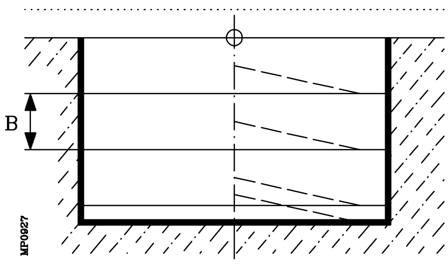
B±5.5 Defines the cutting pass along the longitudinal axis to the main plane.

- If this value is positive, the entire cycle will be executed with the same machining pass, this being equal to or less than that programmed.

- If this value is negative, the entire pocket will be executed with the given pass, except for the last pass which will machine the rest.

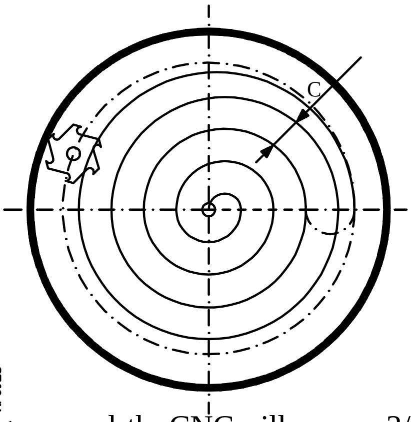
C±5.5 Defines the milling pass along the main plane.

- If the value is positive, the entire cycle will be executed with the same milling pass,



this being equal to or less than that programmed.

- If the value is negative, the entire pocket will be executed with the given pass, except for the last pass which will machine whatever remains.



MP0928

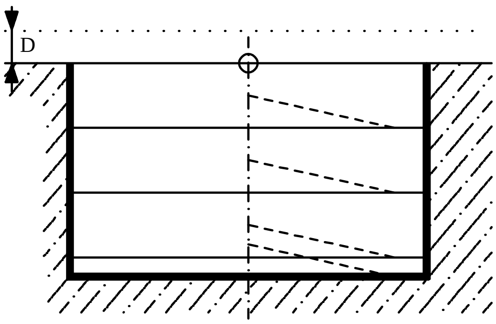
If this is not programmed, the CNC will assume 3/4 of the diameter of the diameter of the selected tool.

If programmed with a value greater than the tool diameter, the CNC will issue the corresponding error.

If programmed with a value of 0, the CNC will show the corresponding error.

D5.5 Defines the distance between the reference plane and the surface of the part where the pocket is to be made.

During the first deepening operation this amount will be added to incremental depth "B". If this is not programmed, a value of 0 will be taken.

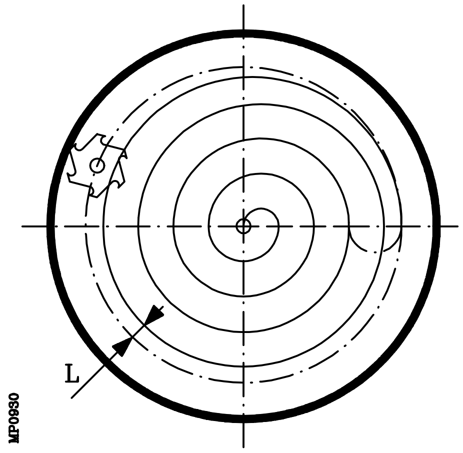


MP0929

H.5 Defines the working feedrate during the finishing pass.

If this is not programmed or is programmed with a value of 0, the value of the working feedrate for machining will be taken.

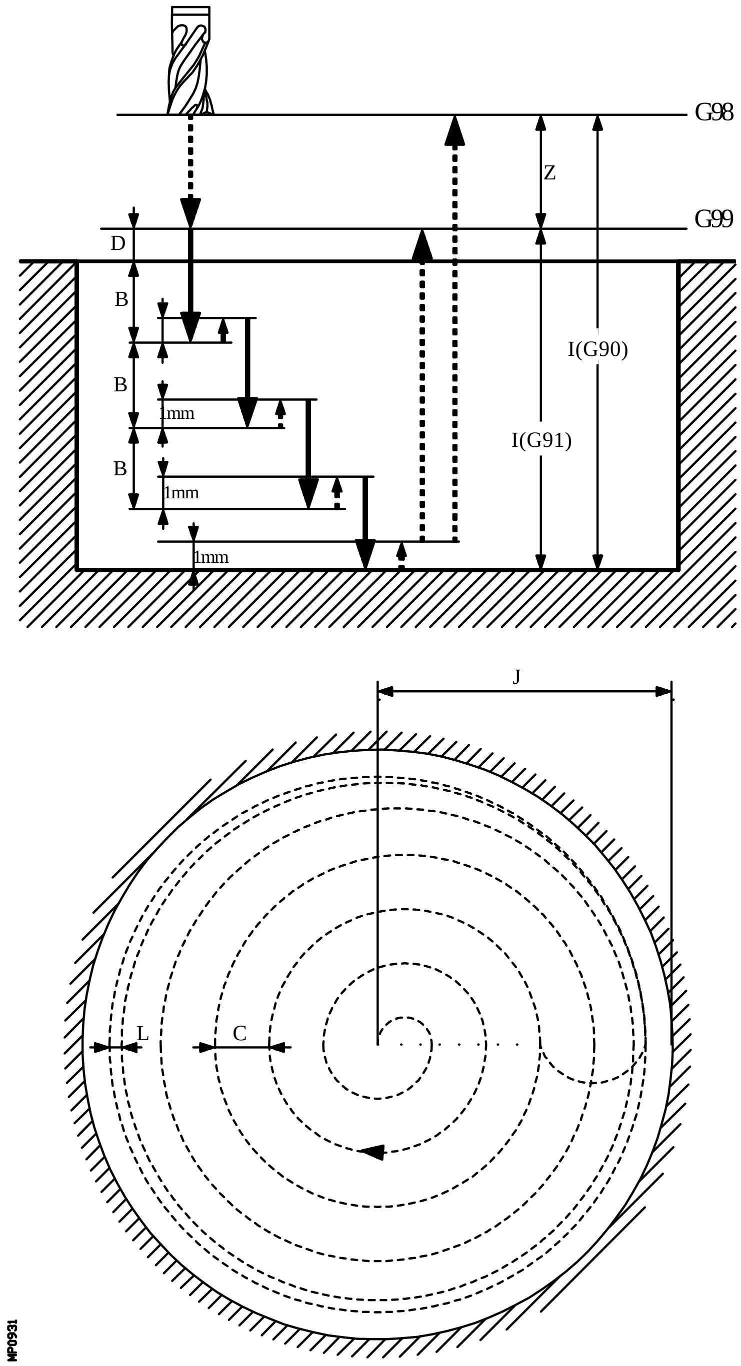
L5.5 Defines the value of the finishing pass, along the main plane.



If this is not programmed or is programmed with a value of 0 no finishing pass will be made.

V.5.5 Defines the tool penetrating feedrate.

If not programmed or programmed with a value of "0", the CNC will assume 50% of the feedrate in the plane (F).


G98
Z
G99
D
B
B
B
1mm
1mm
1mm
I(G90)
I(G91)
J
L
C
MP0931

Basic operation

1. If the spindle was in operation previously, its turning direction is maintained. If it was not in movement, it will start by turning clockwise (M03).

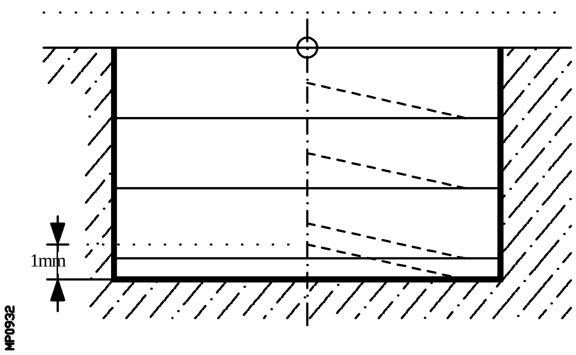
2. Rapid movement (G00) of the longitudinal axis from the initial plane to the reference plane.

3. First deepening operation. Movement of longitudinal axis at the feedrate indicated by "V" to the incremental depth programmed in "B+D".

4. Milling at the working feedrate of the surface of the pocket in steps defined by means of "C" as far as a distance "L" (finishing pass) from the pocket wall.

5. Milling of the "L" finishing pass with the working feedrate defined in "H".

6. Once the finishing pass has been completed, the tool withdraws at the rapid feedrate (G00) to the center of the pocket, the longitudinal axis being separated 1 mm (0.040 inch) from the machined surface.



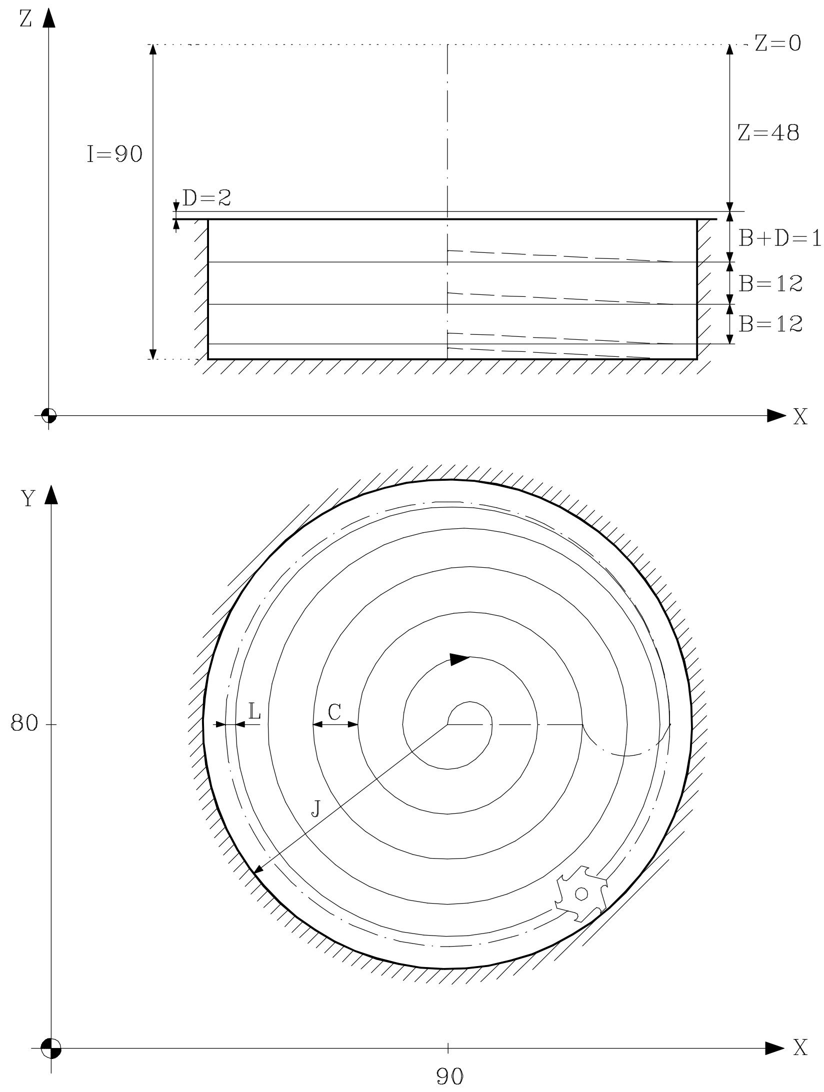
7. Further milling runs until the total depth of the pocket is reached.

   - Movement of the longitudinal axis at the feedrate indicated by "V", up to a distance "B" from the previous surface.

   - Milling of a new surface following the steps indicated in paragraphs 4, 5 and 6.

8. Withdrawal at rapid feedrate (G00) of the longitudinal axis to the initial or reference plane, depending on whether G98 or G99 has been programmed.

Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is the Z axis and that the starting point is X0 Y0 Z0:



(TOR1=6, TOT1=0)
T1 D1
M6
G0 G90 X0 Y0 Z0 .... ; Starting point
G88 G98 G00 G90 X90 Y80 Z-48 I-90 J70 B12 C10
D2 H100 L5 V100 F300 S1000 T1 D1 M03 .... Canned cycle definition
G80 .... ; Canned cycle cancellation
G90 X0 Y0 .... ; Positioning
M30 .... ; End of program

9.5.10. G89. BORING CYCLE WITH WITHDRAWAL AT WORKING FEEDRATE (G01)

This cycle bores at the point indicated until the final programmed coordinate is reached.

It is possible to program a dwell at the bottom of the machined hole.

Working in cartesian coordinates, the basic structure of the block is as follows:

G89 G98/G99 X Y Z I K

G98 The tool withdraws to the Initial Plane, once the hole has been bored.

G99 The tool withdraws to the Reference Plane, once the hole has been bored.

XY±5.5 These are optional and define the movement of the axes of the main plane to position the tool at the machining point.

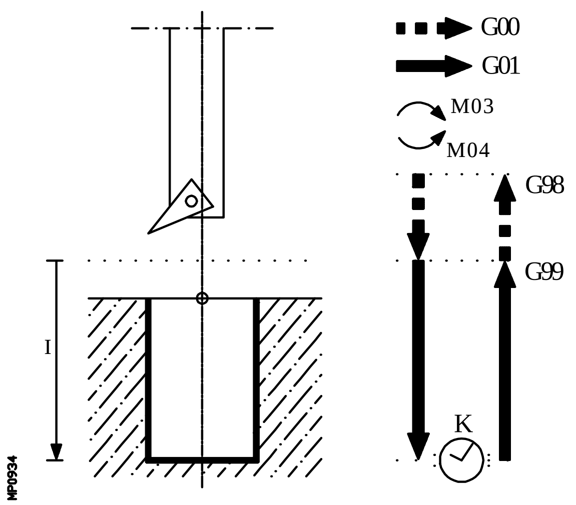
This point can be programmed in cartesian coordinates or in polar coordinates, and the coordinates may be absolute or incremental, along whether the machine is operating in G90 or G91.

Z±5.5 Defines the reference plane coordinate. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the initial plane.

If this is not programmed, the CNC will take the position occupied by the tool at that moment as the reference plane.

I±5.5 Defines boring depth. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the reference plane.

K5 Defines the dwell time, in hundredths of a second, after each boring step, until the withdrawal begins. Should this not be programmed, the CNC will take a value of K0.



Basic operation

1. If the spindle was in operation previously, its turning direction is maintained. If it was not in movement, it will start by turning clockwise (M03).

2. Rapid movement of the longitudinal axis from the initial plane to the reference plane.

3. Movement at the working feedrate (G01) of the longitudinal axis to the bottom of the machined hole, and boring.

4. Spindle stop (M05).

5. Withdrawal at working feedrate of the longitudinal axis to the reference plane.

6. Withdrawal, at rapid feedrate (G00), of the longitudinal axis as far as the initial plane if G98 has been programmed.

Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is the Z axis and that the starting point is X0 Y0 Z0:

```
T1
M6
G0 G90 X0 Y0 Z0 ......................................................... ;Starting point
G89 G98 G91 X250 Y350 Z-98 I-22 K20 F100 S500 ... ;Canned cycle definition
G80 .................................................................................. ;Canned cycle cancellation
G90 X0 Y0 ....................................................................... ;Positioning
M30 .................................................................................. ;End of program
```

10. *MULTIPLE MACHINING*

Multiple functions are defined as a series of functions which allow a machining operation to be repeated along a given path.

The programmer will select the type of machining, which can be a canned cycle or a subroutine (which must be programmed as a modal subroutine) defined by the user.

Machining subroutines are defined by the following functions:

G60: multiple machining in a straight line pattern.
G61: multiple machining in a rectangular pattern.
G62: multiple machining in a grid pattern.
G63: multiple machining in a circular pattern.
G64: multiple machining in an arc pattern.
G65: multiple machining in an arc-chord pattern.

These functions can be performed on any work plane and must be defined every time they are used, as they are not modal.

It is absolutely essential for the machining which it is required to repeat to be active. In other words, these functions will only make sense if they are under the influence of a canned cycle or under the influence of a modal subroutine.

To perform multiple machining, follow these steps:

1.- Move the tool to the first point of the multiple machining operation.

2- Define the canned cycle or modal subroutine to be repeated at all the points.

3.- Define the multiple operation to be performed.

All machining operations programmed with these functions will be done under the same working conditions (T,D,F,S) which were selected when defining the canned cycle or modal subroutine.

Once the multiple machining operation has been performed, the program will recover the history it had before starting this machining, even when the canned cycle or modal subroutine will remain active. Now feedrate F corresponds to the feedrate programmed for the canned cycle or modal subroutine.

Likewise, the tool will be positioned at the last point where the programmed machining operation was done.

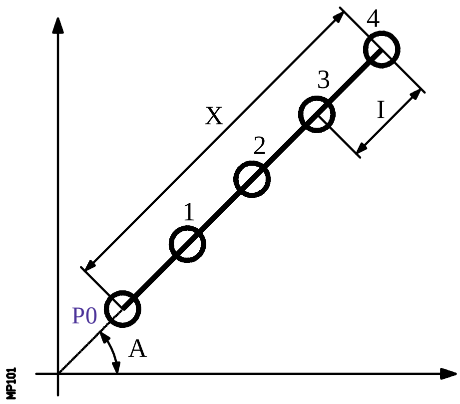
If multiple machining of a modal subroutine is performed in the Single Block mode, this subroutine will be performed complete (not block by block) after each programmed movement.

A detailed explanation is given on the next page of multiple machining operations, assuming in each case, that the work plane is formed by X and Y axes.

10.1 *G60: MULTIPLE MACHINING IN A STRAIGHT LINE PATTERN*

The programming format of this cycle is as follows:

G60 A	**X I**	**P Q R S T U V**
	X K	
	I K	



A(+/-5.5) Defines the angle which forms the machining path with the abscissa axis. It is expressed in degrees and if not programmed, the value A=0 will be taken.

X(5.5) Defines the length of the machining path.

I(5.5) Defines the pitch between machining operations.

K(5) Defines the number of total machining operations in the section, including the machining definition point.

Due to the fact that machining may be defined with any two points of the **X I K** group, the CNC allows the following definition combinations: **XI**, **XK**, **IK**.

Nevertheless, if format **XI** is defined, care should be taken to ensure that the number of machining operations is an integer number, otherwise the CNC will show the corresponding error code.

P,Q,R,S,T,U,V These parameters are optional and are used to indicate at which points or between which those programmed points it is not required to machine.

Thus, programming P7 indicates that it is not required to do machining at point 7, and programming Q10.013 indicates that machining is not required from point 10 to 13, or expressed in another way, that no machining is required at points 10, 11, 12 and 13.

When it is required to define a group of points (Q10.013), care should be taken to define the final point with three digits, as if Q10.13 is programmed, multiple machining understands Q10.130.

The programming order for these parameters is P Q R S T U V, it also being necessary to maintain the order in which the points assigned to these are numbered, i.e., the numbering order of the points assigned to Q must be greater than that assigned to P and less than that assigned to R.

Example:

Proper programming	P5.006 Q12.015 R20.022
Improper programming	P5.006 Q20.022 R12.015

If these parameters are not programmed, the CNC understands that it must perform machining at all the points along the programmed path.

Basic operation:

1. Multiple machining calculates the next point of those programmed where it is wished to machine.

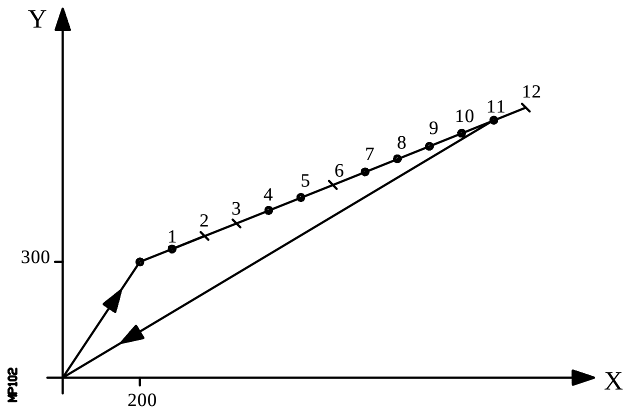
2. Rapid traverse (G00) to this point.

3. Multiple machining will perform the canned cycle or modal subroutine selected after this movement.

4. The CNC will repeat steps 1-2-3 until the programmed path has been completed.

   After completing multiple machining, the tool will be positioned at the last point along the programmed path where machining was performed.

Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is Z and that the starting point is X0 Y0 Z0.



G81 G98 G00 G91 X200 Y300 Z-8 I-22 F100 S500	;Canned cycle positioning and definition
G60 A30 X1200 I100 P2.003 Q6 R12	;Defines multiple machining
G80	;Cancels canned cycle
G90 X0 Y0	;Positioning
M30	;End of program

It is also possible to write the multiple machining definition block in the following ways:

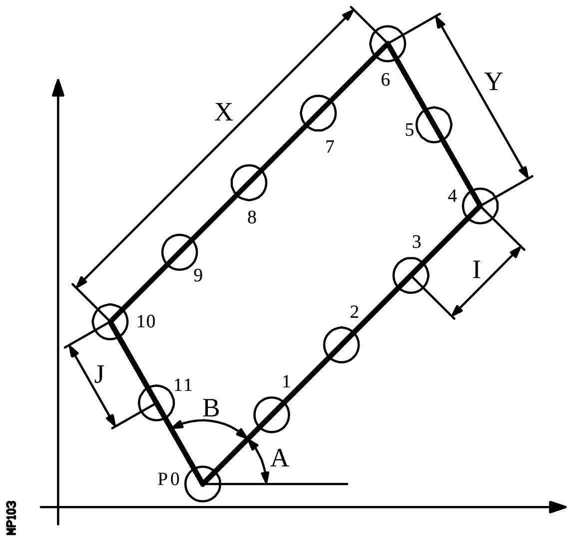
G60 A30 **X1200 K13** P2.003 Q6 R12

G60 A30 **I100** **K13** P2.003 Q6 R12

10.2 G61: MULTIPLE MACHINING IN A RECTANGULAR PATTERN

The programming format of this cycle is as follows:

G61 A B	X I	Y J	P Q R S T U V
	X K	Y D	
	I K	J D	



A(+/-5.5) Defines the angle formed by the machining path with the abscissa axis. It is expressed in degrees and if not programmed, the value A=0 will be taken.

B(+/-5.5) Defines the angle formed by the two machining paths. It is expressed in degrees and if not programmed, the value B=90 will be taken.

X(5.5) Defines the length of the machining path according to the abscissa axis.

I(5.5) Defines the pitch between machining operations according to the abscissa axis.

K(5) Defines the number of total machining operations in the abscissa axis, including the machining definition point.

Due to the fact that machining may be defined according to the abscissa axis with any two points of the **X I K** group, the CNC allows the following definition combinations: **XI**, **XK**, **IK**.

Nevertheless, if format **XI** is defined, care should be taken to ensure that the number of machining operations is an integer number, otherwise the CNC will show the corresponding error code.

Y(5.5) Defines the length of the machining path according to the ordinate axis.

J(5.5) Defines the pitch between machining operations according to the ordinate axis.

D(5) Defines the number of total machining operations in the ordinate axis, including the machining definition point.

Due to the fact that machining may be defined according to the ordinate axis with any two points of the **Y J D** group, the CNC allows the following definition combinations: **YJ**, **YD**, **JD**.

Nevertheless, if format **YJ** is defined, care should be taken to ensure that the number of machining operations is an integer number, otherwise the CNC will show the corresponding error code.

P,Q,R,S,T,U,V These parameters are optional and are used to indicate at which points or between which of those programmed points it is not required to machine.

Thus, programming P7 indicates that it is not required to do machining at point 7, and programming Q10.013 indicates that machining is not required from point 10 to 13, or expressed in another way, that no machining is required at points 10, 11, 12 and 13.

When it is required to define a group of points (Q10.013), care should be taken to define the final point with three digits, as if Q10.13 is programmed, multiple machining understands Q10.130.

The programming order for these parameters is P Q R S T U V, it also being necessary to maintain the order in which the points assigned to these are numbered, i.e., the numbering order of the points assigned to Q must be greater than that assigned to P and less than that assigned to R.

Example:

Proper programming P5.006 Q12.015 R20.022
Improper programming P5.006 Q20.022 R12.015

If these parameters are not programmed, the CNC understands that it must perform machining at all the points along the programmed path.

Basic operation:

1. Multiple machining calculates the next point of those programmed where it is wished to machine.

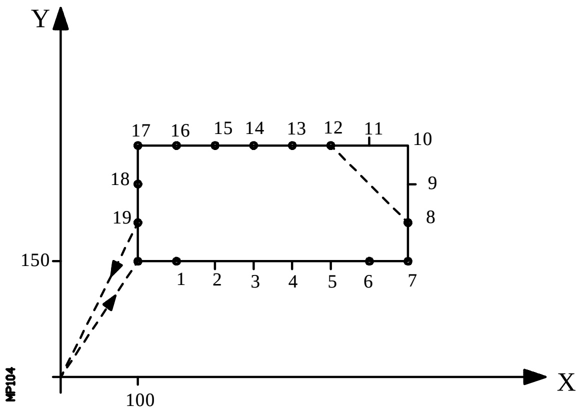
2. Rapid traverse (G00) to this point.

3. Multiple machining will perform the canned cycle or modal subroutine selected after this movement.

4. The CNC will repeat steps 1-2-3 until the programmed path has been completed.

   After completing multiple machining, the tool will be positioned at the last point along the programmed path where machining was performed.

Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is Z and that the starting point is X0 Y0 Z0.



G81 G98 G00 G91 X100 Y150 Z-8 I-22 F100 S500	;Canned cycle positioning and definition
G61 X700 I100 Y180 J60 P2.005 Q9.011	;Defines multiple machining
G80	;Cancels canned cycle
G90 X0 Y0	;Positioning
M30	;End of program

It is also possible to write the multiple machining definition block in the following ways:

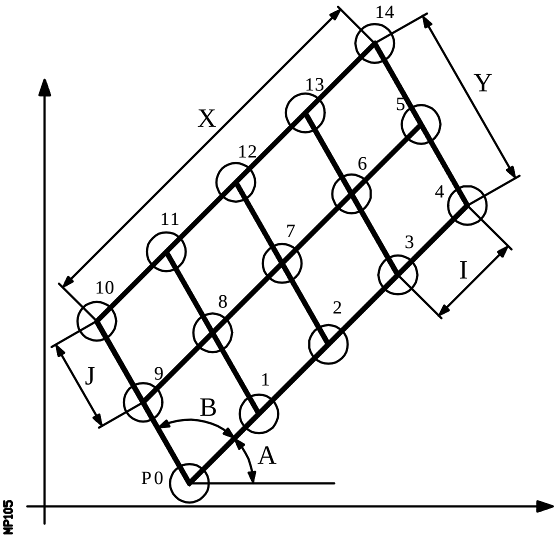
G61 **X700 K8 J60 D4** P2.005 Q9.001

G61 **I100 K8 Y180 D4** P2.005 Q9.011

10.3 G62: MULTIPLE MACHINING IN A GRID PATTERN

The programming format of this cycle is as follows:

G62 A B	X I	Y J	P Q R S T U V
	X K	Y D	
	I K	J D	



A(+/-5.5) Defines the angle formed by the machining path with the abscissa axis. It is expressed in degrees and if not programmed, the value A=0 will be taken.

B(+/-5.5) Defines the angle formed by the two machining paths. It is expressed in degrees and if not programmed, the value B=90 will be taken.

X(5.5) Defines the length of the machining path according to the abscissa axis.

I(5.5) Defines the pitch between machining operations according to the abscissa axis.

K(5) Defines the number of total machining operations in the abscissa axis, including the machining definition point.

Due to the fact that machining may be defined according to the abscissa axis with any two points of the **X I K** group, the CNC allows the following definition combinations: **XI**, **XK**, **IK**.

Nevertheless, if format **XI** is defined, care should be taken to ensure that the number of machining operations is an integer number, otherwise the CNC will show the corresponding error code.

Y(5.5) Defines the length of the machining path according to the ordinate axis.

J(5.5) Defines the pitch between machining operations according to the ordinate axis.

D(5) Defines the number of total machining operations in the ordinate axis, including the machining definition point.

Due to the fact that machining may be defined according to the ordinate axis with any two points of the **Y J D** group, the CNC allows the following definition combinations: **YJ**, **YD**, **JD**.

Nevertheless, if format **YJ** is defined, care should be taken to ensure that the number of machining operations is an integer number, otherwise the CNC will show the corresponding error code.

P,Q,R,S,T,U,V These parameters are optional and are used to indicate at which points or between which of those programmed points it is not required to machine.

Thus, programming P7 indicates that it is not required to do machining at point 7, and programming Q10.013 indicates that machining is not required from point 10 to 13, or expressed in another way, that no machining is required at points 10, 11, 12 and 13.

When it is required to define a group of points (Q10.013), care should be taken to define the final point with three digits, as if Q10.13 is programmed, multiple machining understands Q10.130.

The programming order for these parameters is P Q R S T U V, it also being necessary to maintain the order in which the points assigned to these are numbered, i.e., the numbering order of the points assigned to Q must be greater than that assigned to P and less than that assigned to R.

Example:

Proper programming P5.006 Q12.015 R20.022
Improper programming P5.006 Q20.022 R12.015

If these parameters are not programmed, the CNC understands that it must perform machining at all the points along the programmed path.

Basic operation:

1. Multiple machining calculates the next point of those programmed where it is wished to machine.

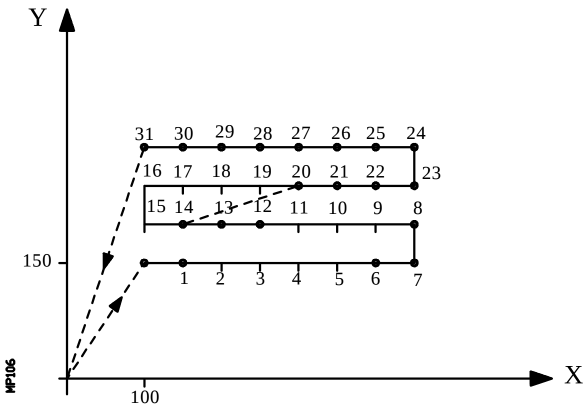
2. Rapid rapid traverse (G00) to this point.

3. Multiple machining will perform the canned cycle or modal subroutine selected after this movement.

4. The CNC will repeat steps 1-2-3 until the programmed path has been completed.

   After completing multiple machining, the tool will be positioned at the last point along the programmed path where machining was performed.

Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is Z and that the starting point is X0 Y0 Z0.



G81 G98 G00 G91 X100 Y150 Z-8 I-22 F100 S500 ;Canned cycle positioning and definition
G62 X700 I100 Y180 J60 P2.005 Q9.011 R15.019 ;Defines multiple machining
G80 ;Cancels canned cycle
G90 X0 Y0 ;Positioning
M30 ;End of program

It is also possible to write the multiple machining definition block in the following ways:

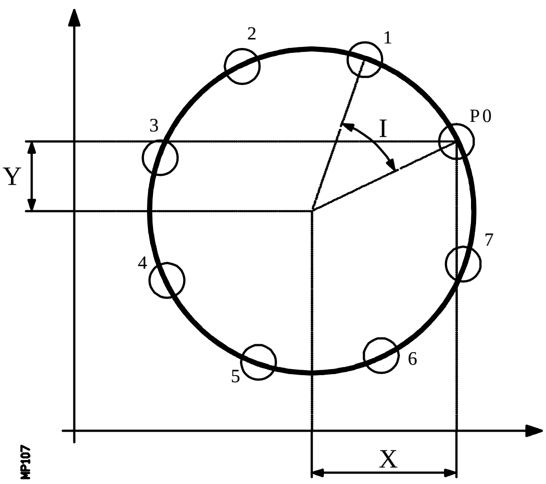
G61 **X700 K8 J60 D4** P2.005 Q9.001 R15.019

G61 **I100 K8 Y180 D4** P2.005 Q9.011 R15.019

10.4 G63: MULTIPLE MACHINING IN A CIRCULAR (BOLT-HOLE) PATTERN

The programming format of this cycle is as follows:

G63 X Y | I / K | C F P Q R S T U V



X(+/-5.5) Defines the distance from the starting point to the center along the abscissa axis.

Y(+/-5.5) Defines the distance from the starting point to the center along the ordinate axis.

With parameters X and Y the center of the circle is defined in the same way that I and J do this in circular interpolations (G02, G03).

I(+/-5.5) Defines the pitch angle between machining operations, if G00 or G01, the sign indicates the direction, "+" counter-clockwise, "-" clockwise.

K(5) Defines the number of total machining operations along the circle, including the machining definition point.

It will be enough to program I or K in the multiple machining definition block. Nevertheless, if K is programmed in a multiple machining operation in which movement between points is made in G00 or G01, machining will be done in the counter-clockwise direction.

C Indicates how movement is made between machining points. If it is not programmed, the value C=0 will be taken.

C=0: Movement is made in rapid feedrate (G00)
C=1: Movement is made in linear interpolation (G01).
C=2: Movement is made in clockwise circular interpolation (G02)
C=3: Movement is made in counter-clockwise circular interpolation (G03)

F(5.5) Defines the feedrate which is used for moving between points. Obviously, it will only apply for "C" values other than zero. If it is not programmed, the value F0 will be taken, maximum feedrate selected by the "MAXFEED" axis machine parameter.

P,Q,R,S,T,U,V These parameters are optional and are used to indicate at which points or between which of those programmed points it is not required to machine.

Thus, programming P7 indicates that it is not required to do machining at point 7, and programming Q10.013 indicates that machining is not required from point 10 to 13, or expressed in another way, that no machining is required at points 10, 11, 12 and 13.

When it is required to define a group of points (Q10.013), care should be taken to define the final point with three digits, as if Q10.13 is programmed, multiple machining understands Q10.130.

The programming order for these parameters is P Q R S T U V, it also being necessary to maintain the order in which the points assigned to these are numbered, i.e., the numbering order of the points assigned to Q must be greater than that assigned to P and less than that assigned to R.

Example:

Proper programming P5.006 Q12.015 R20.022
Improper programming P5.006 Q20.022 R12.015

If these parameters are not programmed, the CNC understands that it must perform machining at all the points along the programmed path.

Basic operation:

1. Multiple machining calculates the next point of those programmed where it is wished to machine.

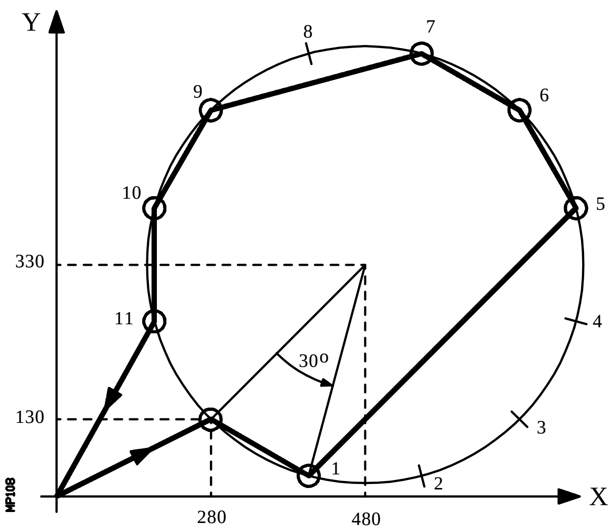
2. Movement programmed by "C" (G00,G01,G02 or G03) to this point.

3. Multiple machining will perform the canned cycle or modal subroutine selected after this movement.

4. The CNC will repeat steps 1-2-3 until the programmed path has been completed.

   After completing multiple machining, the tool will be positioned at the last point along the programmed path where machining was performed.

   Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is Z and that the starting point is X0 Y0 Z0.




```
G81 G98 G01 G91 X280 Y130 Z-8 I-22 F100 S500   ;Canned cycle positioning and definition
G63 X200 Y200 I30 C1 F200 P2.004 Q8            ;Defines multiple machining
G80                                            ;Cancels canned cycle
G90 X0 Y0                                      ;Positioning
M30                                            ;End of program
```

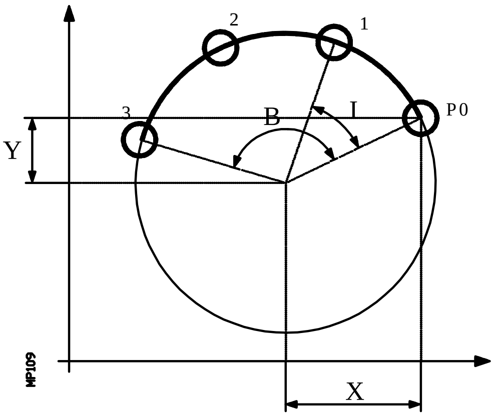
It is also possible to write the multiple machining definition block in the following ways:

G63 X200 Y200 **K12** C1 F200 P2.004 Q8

10.5 G64: MULTIPLE MACHINING IN AN ARC PATTERN

The programming format of this cycle is as follows:

G64 X Y B | I / K | C F P Q R S T U V



X(+/-5.5) Defines the distance from the starting point to the center along the abscissa axis.

Y(+/-5.5) Defines the distance from the starting point to the center along the ordinate axis.

With parameters X and Y the center of the circle is defined in the same way that I and J do this in circular interpolations (G02, G03).

B(5.5) Defines the angular stroke of the machining path and is expressed in degrees.

I(+/-5.5) Defines the pitch angle between machining operations, if G00 or G01, the sign indicates the direction, "+" counter-clockwise, "-" clockwise.

K(5) Defines the number of total machining operations along the circle, including the machining definition point.

It will be enough to program I or K in the multiple machining definition block. Nevertheless, if K is programmed in a multiple machining operation in which movement between points is made in G00 or G01, machining will be done in the counter-clockwise direction.

C Indicates how movement is made between machining points. If it is not programmed, the value C=0 will be taken.

C=0: Movement is made in rapid feedrate (G00)
C=1: Movement is made in linear interpolation (G01).
C=2: Movement is made in clockwise circular interpolation (G02)
C=3: Movement is made in counter-clockwise circular interpolation (G03)

F(5.5) Defines the feedrate which is used for moving between points. Obviously, it will only have value for "C" values other than zero. If it is not programmed, the value F0 will be taken, maximum feedrate selected by the "MAXFEED" axis machine parameter.

P,Q,R,S,T,U,V These parameters are optional and are used to indicate at which points or between which of those programmed points it is not required to machine.

Thus, programming P7 indicates that it is not required to do machining at point 7, and programming Q10.013 indicates that machining is not required from point 10 to 13, or expressed in another way, that no machining is required at points 10, 11, 12 and 13.

When it is required to define a group of points (Q10.013), care should be taken to define the final point with three digits, as if Q10.13 is programmed, multiple machining understands Q10.130.

The programming order for these parameters is P Q R S T U V, it also being necessary to maintain the order in which the points assigned to these are numbered, i.e., the numbering order of the points assigned to Q must be greater than that assigned to P and less than that assigned to R.

Example:

Proper programming P5.006 Q12.015 R20.022
Improper programming P5.006 Q20.022 R12.015

If these parameters are not programmed, the CNC understands that it must perform machining at all the points along the programmed path.

Basic operation:

1. Multiple machining calculates the next point of those programmed where it is wished to machine.

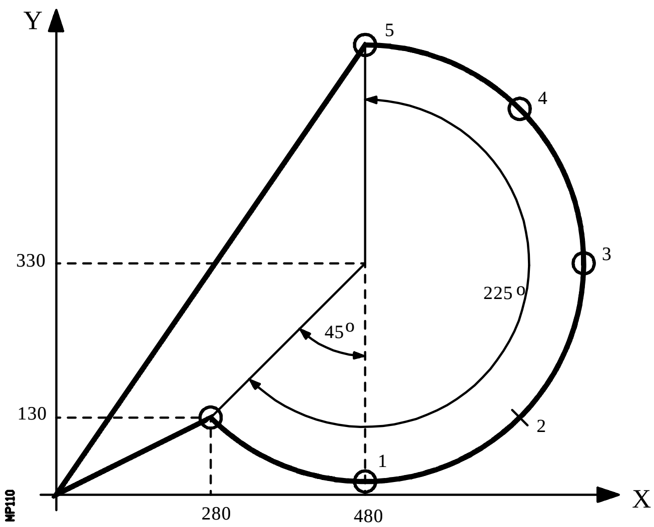
2. Movement programmed by "C" (G00,G01,G02 or G03) to this point.

3. Multiple machining will perform the canned cycle or modal subroutine selected after this movement.

4. The CNC will repeat steps 1-2-3 until the programmed path has been completed.

   After completing multiple machining, the tool will be positioned at the last point along the programmed path where machining was performed.

   Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is Z and that the starting point is X0 Y0 Z0.



G81 G98 G01 G91 X280 Y130 Z-8 I-22 F100 S500	;Canned cycle positioning and definition
G64 X200 Y200 B225 K6 C3 F200 P2	;Defines multiple machining
G80	;Cancels canned cycle
G90 X0 Y0	;Positioning
M30	;End of program

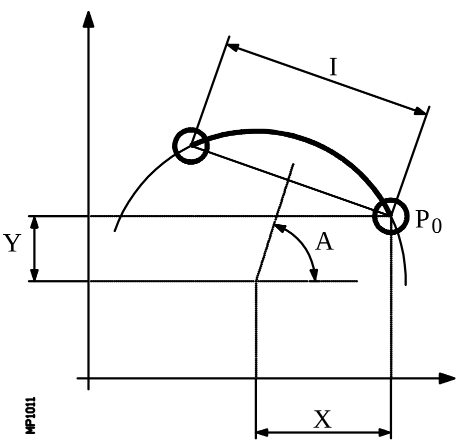
It is also possible to write the multiple machining definition block in the following ways:

G64 X200 Y200 B225 **K6** C3 F200 P2

10.6 G65: MACHINING PROGRAMMED BY MEANS OF AN ARC CHORD

This function allows activated machining to be performed at a point programmed by means of an arc chord. Only one machining operation will be performed, its programming format being:

G65 X Y | A / I | C F



X(+/-5.5) Defines the distance from the starting point to the center along the abscissa axis.

Y(+/-5.5) Defines the distance from the starting point to the center along the ordinate axis.

With parameters X and Y the center of the circle is defined in the same way that I and J do this in circular interpolations (G02, G03).

A(+/-5.5) Defines the angle formed by the perpendicular bisector of the chord with the abscissa axis and is expressed in degrees.

I(+/-5.5) Defines the chord length. When moving in G00 or G01, the sign indicates the direction, "+" counter-clockwise, "-" clockwise.

C Indicates how movement is made between machining points. If it is not programmed, the value C=0 will be taken.

C=0: Movement is made in rapid feedrate (G00)
C=1: Movement is made in linear interpolation (G01).
C=2: Movement is made in clockwise circular interpolation (G02)
C=3: Movement is made in counter-clockwise circular interpolation (G03)

F(5.5) Defines the feedrate which is used for moving between points. Obviously, it will only apply for "C" values other than zero. If it is not programmed, the value F0 will be taken, maximum feedrate selected by the "MAXFEED" axis machine parameter.

Basic operation:

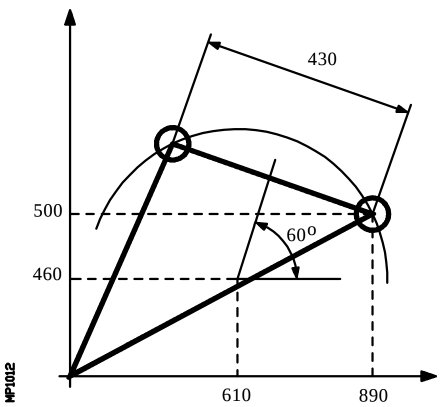
1. Multiple machining calculates the next point of those programmed where it is wished to machine.

2. Movement programmed by "C" (G00,G01,G02 or G03) to this point.

3. Multiple machining will perform the canned cycle or modal subroutine selected after this movement.

   After completing multiple machining, the tool will be positioned at the programmed point.

   Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is Z and that the starting point is X0 Y0 Z0.



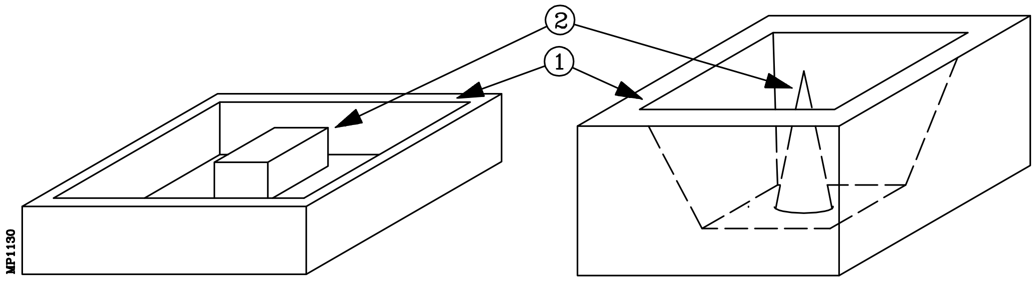

```
G81 G98 G01 G91 X890 Y500 Z-8 I-22 F100 S500   ;Canned cycle positioning and definition
G65 X280 Y-40 A60 C1 F200                      ;Defines multiple machining
G80                                            ;Cancels canned cycle
G90 X0 Y0                                      ;Positioning
M30                                            ;End of program
```

It is also possible to write the multiple machining definition block in the following ways:

G65 X-280 Y40 **I430** C1 F200

11. *IRREGULAR POCKET CANNED CYCLE (WITH ISLANDS)*

A pocket is composed by an external contour or profile (1) and a series of internal contours or profiles (2). These internal profiles are called islands.

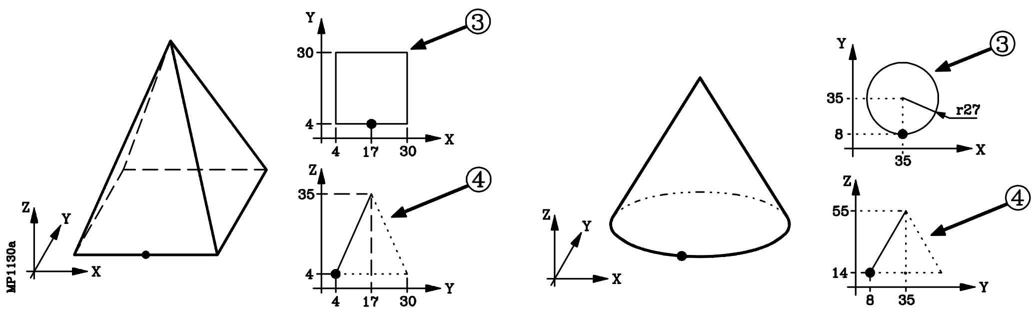


With this pocket canned cycle, 2D and 3D pockets may be machined.

2D pocket (Upper left-hand illustration).
Its inside and outside walls are vertical.
Its programming is detailed in the first part of this chapter.
To define the contours of a 2D pocket, the plane profile for all the contours must be defined.

3D pocket (Upper right-hand illustration).
When any of the inside or outside profiles and/or islands is not vertical.
Its programming is detailed in the second part of this chapter.

To define the contours of a 2D pocket, the plane profile (3) and the depth profile (4) for all the contours must be defined (even if they are vertical).



The call function for a 2D or 3D irregular pocket canned cycle is G66.

The machining of a pocket may consist of the following operations:

Drilling operation, prior to machining .... Only on 2D pockets
Roughing operation .... 2D and 3D pockets
Semi-finishing operation .... Only on 3D pockets
Finishing operation .... 2D and 3D pockets

11.1 2D POCKETS

The G66 function is not modal, therefore it must be programmed whenever it is required to perform a 2D pocket.

In a block defining an irregular pocket canned cycle, no other function can be programmed, its structure definition being:

G66 D H R I F K S E Q

D (0-9999) & H (0-9999) Label number of the first block (D) and last block (H) defining the drilling operation.
When not setting "H" only block "D" is executed.
When not setting "D" there is no drilling operation.

R (0-9999) & I (0-9999) Label number of the first block (R) and last block (I) defining the roughing operation.
When not setting "I" only block "R" is executed.
When not setting "R" there is no roughing operation.

F (0-9999) & K (0-9999) Label number of the first block (F) and last block (K) defining the finishing operation.
When not setting "K" only block "F" is executed.
When not setting "F" there is no finishing operation.

S (0-9999) & E (0-9999) Label number of the first block (S) and last block (E) defining the geometry of the profiles forming the pocket.
Both parameters must be set.

Q (0-9999) Number of the program containing the geometry definition, parameters S and E.
If it is in the same program, "Q" need not be defined.

Programming example:

```
      G00 G90 X100 Y200 Z50 F5000 T1 D2   ;Initial positioning
      M06
      G66 D100 R200 I210 F300  S400 E500  ;Definition of irregular pocket canned cycle
      M30                                 ;End of program

N100  G81 ............                    ;Defines the drilling operation
N200  ...................                 ;Starts the roughing operation
      G67 ............
N210  ...................                 ;End the roughing operation
N300  G68 ............                    ;Defines the finishing operation
N400  G0 G90 X300 Y50 Z3                  ;Starts the geometry description
      ...................
      ...................
N500 G2 G6 X300 Y50 I150 J0               ;End of geometry description
```

Basic operation:

1.- Drilling operation. Only if it has been programmed.

After analyzing the geometry of the pocket with islands, the tool radius and the angle of the path programmed in the roughing operation, the CNC will calculate the coordinates of the point where the selected drilling operation must be performed.

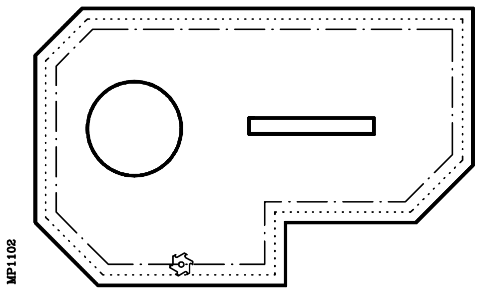
2.- Roughing operation. Only if it has been programmed.

It consists of several surface milling passes, until the total depth programmed has been reached. On each surface milling pass, the steps below will be followed depending on the type of machining that has been programmed:

Case A: When the machining paths are linear and maintain a certain angle with the abscissa axis.

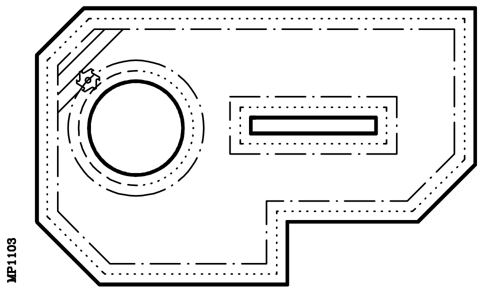
* It first contours the external profile of the part.

If the finishing operation has been selected on the cycle call, this contouring is performed leaving the finishing stock programmed for the finishing pass.

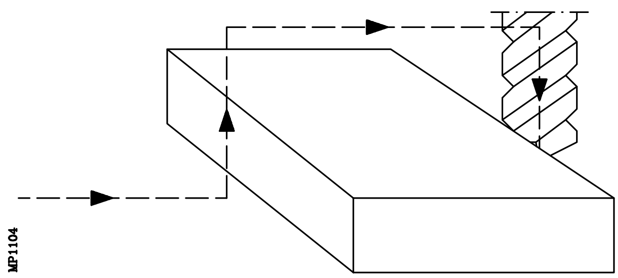


* Next the milling operation, with the programmed feed and steps.

If, while milling, an island is run into for the first time, it will be contoured.

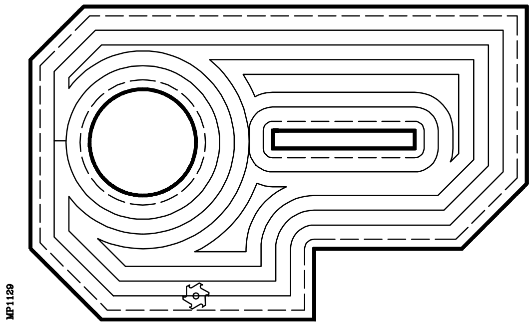


After the contouring and the remaining times, the tool will pass over the island, withdrawing along the longitudinal axis, to the reference plane, and will continue machining once the island has been cleared.



Case B: When the machining paths are concentric

* The roughing operation is carried out along paths concentric to the profile. The machining will be done as fast as possible avoiding (when possible) going over the islands.

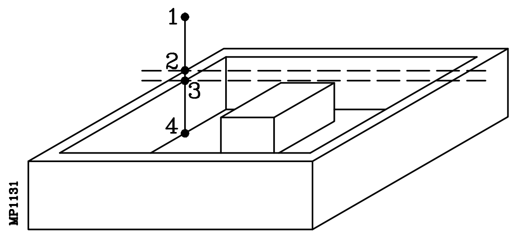


3.- Finishing operation. Only if it has been programmed.

This operation can be done on a single pass or on several, as well as following the profiles in the programmed direction or in the opposite.

The CNC will machine both the external profile and the islands, making tangential approaches and exits to these with a constant surface speed.

In the pocket canned cycle with islands, there are four coordinates along the longitudinal axis (selected with G15), which, due to their importance, are discussed below:



1.- Initial plane coordinate. This coordinate is given by the position which the tool occupies when the cycle is called.

2.- Reference plane coordinate. This represents an approach coordinate to the part, and must be programmed in absolute coordinates.

3.- Part surface coordinate. This is programmed in absolute coordinates and in the first profile definition block.

4.- Machining depth coordinate. This is programmed in absolute coordinates.

Conditions after finishing the cycle

Once the canned cycle has been completed, the active feedrate will be the last programmed feedrate, the one relating to the roughing or finishing operation. Likewise, the CNC will assume functions G00, G07, G40 and G90.

11.1.1 DRILLING OPERATION

This operation is optional and in order to be executed it is necessary to also program a roughing operation.

It is mainly used when the tool programmed in the roughing operation does not machine along the longitudinal axis, allowing, by means of this operation, the access of this tool to the surface to be roughed off.

It will be programmed in a block which will need to bear a label number in order to indicate to the canned cycle the block where the drilling operation is defined.

Example:	G66 **D100** R200 F300 S400 E500	; Definition of the irregular pocket cycle.
	N100 G81	; Definition of the drilling operation.

The drilling canned cycles that can be programmed are:

- **G69** Complex deep hole drilling canned cycle (with variable step).
- **G81** Drilling canned cycle.
- **G82** Drilling canned cycle with dwell.
- **G83** Simple deep hole drilling canned cycle (with constant step).

When defining the drilling operation, the corresponding definition parameters must be programmed together with the required function.

In a block of this type, only cycle definition parameters must be programmed, without defining XY positioning, as the canned cycle itself will calculate the coordinate of the point or points to be drilled according to the programmed profile and the roughing angle.

After the definition parameters, auxiliary F S T D M functions can be programmed, if so wished. No M function can be programmed if it has an associated subroutine.

It is possible to program the M06 function in this block (if it does not have an associated subroutine), to make the tool change. Otherwise, the CNC will show the corresponding error. If the M06 has an associated subroutine, the drilling tool "T" must be selected before calling the cycle.

Examples:

```
N100 G69 G98 G91 Z-4 I-90 B1.5 C0.5 D2 H2 J4 K100 F500 S3000 M3
N120 G81 G99 G91 Z-5 I-30 F400 S2000 T3 D3 M3
N220 G82 G99 G91 Z-5 I-30 K100 F400 S2000 T2 D2 M6
N200 G83 G98 G91 Z-4 I-5 J6 T2 D4
```

11.1.2 *ROUGHING OPERATION*

This is the main operation in the machining of an irregular pocket, and its programming is optional.

This operation will be carried out in either square corner (G07) or round corner (G05) as it is currently selected. However, the canned cycle will assign the G07 format to the necessary movements.

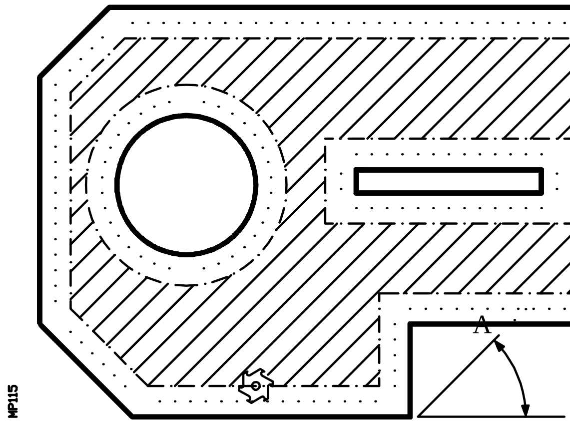
It will be programmed in a block which will need to bear a label number in order to indicate to the canned cycle the block where the roughing operation is defined.

Example: G66 D100 **R200** F300 S400 E500 ; Definition of the irregular pocket cycle.
N200 G67 .... ; Definition of the roughing operation.

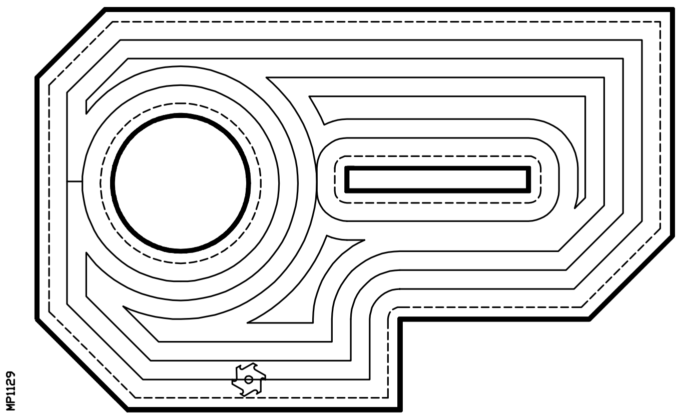
The function for the roughing operation is G67 and its programming format:

G67 A B C I R K V Q F S T D M

A(+/-5.5) Defines the angle which forms the roughing path with the abscissa axis.



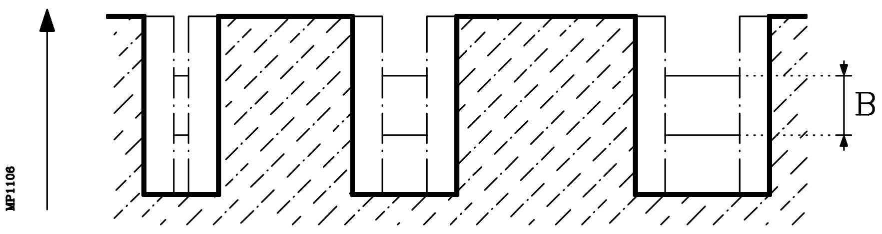
If parameter "A" is not programmed, the roughing operation is carried out following concentric paths. It will be machined as fast as possible since it does not have to go over the islands.



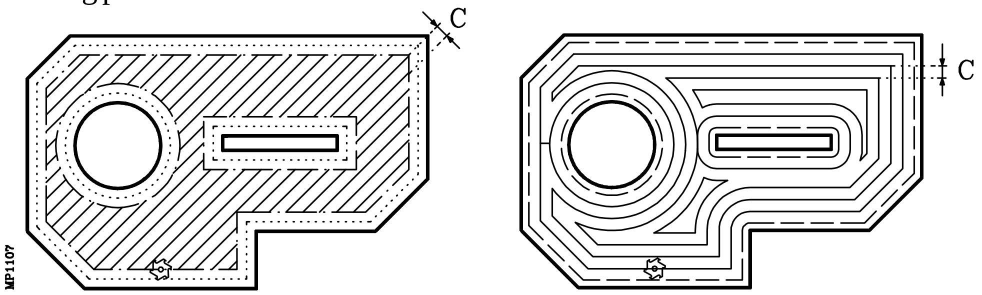
B(+/-5.5) Defines the machining pass along the longitudinal axis (depth of the roughing pass). It must be defined and it must have a value other than 0; otherwise, the roughing operation will be cancelled.

- If programmed with a positive sign, all the roughing will be performed with the same machining pass, and the canned cycle calculates a pass equal to or smaller than the programmed pass.

- If programmed with a negative sign, all the roughing will be performed with the programmed pass, and the canned cycle will adjust the last pass to obtain the total programmed depth.



C(+/-5.5) Defines the milling pass in roughing along the main plane, the entire pocket being performed with the given pass, and the canned cycle adjusts the last milling pass.

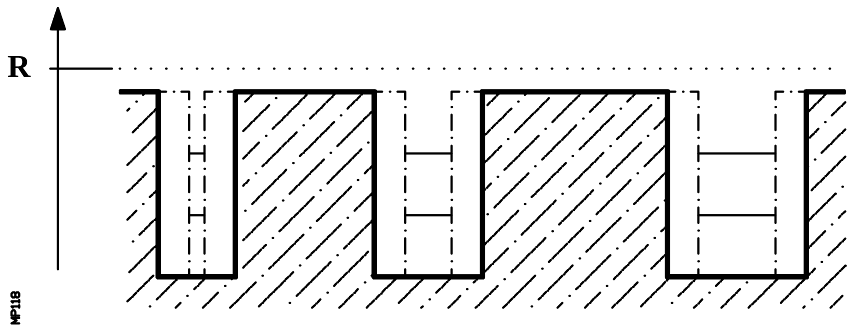


If it is not programmed or is programmed with either a value of 0, it will assume a value of 3/4 the diameter of the selected tool.

If programmed with a value greater than the tool diameter, the CNC will issue the corresponding error.

I(+/-5.5) Defines the total depth of the pocket and is programmed in absolute coordinates. It must be programmed.

R(+/-5.5) Defines the reference plane coordinate and is programmed in absolute coordinates. It must be programmed.



K(1) Defines the type of profile intersection to be used.

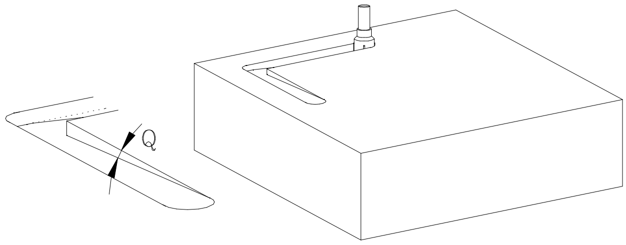
0 = Basic profile intersection.
1 = Advanced profile intersection.

If not programmed, a value of 0 will be assumed. Both intersection types will be discussed later on.

V (5.5) Defines the tool penetrating feedrate.

If not programmed or programmed with a value of "0", the CNC will assume 50% of the feedrate in the plane (F).

Q (5.5) Optional. Tool penetration angle.



If not programmed or programmed with a value of 90, it means that the tool penetrates vertically.

If programmed with a value lower than 0 or higher than 90, it will issue an error message indicating "wrong parameter value in canned cycle".

F (5.5) Optional. Defines the machining feedrate in the plane.

S (5.5) Optional. Defines the spindle speed.

T (4) Defines the tool used for the roughing operation. It must be programmed.

D (4) Optional. Defines the tool offset number.

M Optional. Up to 7 miscellaneous M functions can be programmed.

This operation allows M06 with an associated subroutine to be defined, and the tool change is performed before beginning the roughing operation.

11.1.3 FINISHING OPERATION

This is the last operation in the machining of an irregular pocket, and its programming is optional.

It will be programmed in a block which will need to bear a label number in order to indicate to the canned cycle the block where the finishing operation is defined.

Example: G66 D100 R200 **F300** S400 E500 ; Definition of the irregular pocket cycle.
N300 G68 .... ; Definition of the finishing operation.

The function for the finishing operation is G68 and its programming format:

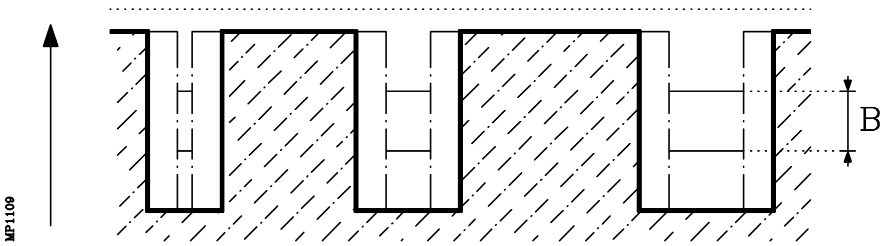
G68 B L Q I R K V F S T D M

B(±5.5) Defines the machining pass along the longitudinal axis (depth of the finishing pass).

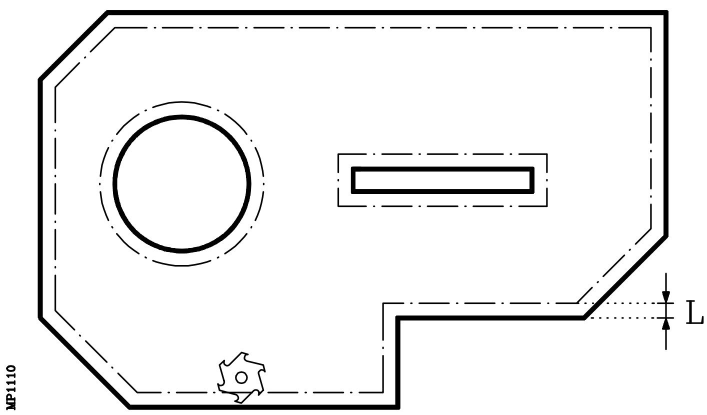
- If it is programmed with a value of 0, the CNC will perform a single finishing pass with the total depth of the pocket.

- If programmed with a positive sign, all the roughing will be performed with the same machining pass, and the canned cycle calculates a pass equal to or lower than the programmed pass.

- If programmed with a negative sign, all the roughing will be performed with the programmed pass, and the canned cycle will adjust the last pass to obtain the total programmed depth.



L(±5.5) Defines the value of the finishing stock which it is required to leave on the side walls of the pocket before the finishing operation.



- If programmed with a positive value, the finishing pass will be carried out in square corner (G07).

- If programmed with a negative value, the finishing pass will be carried out in round corner (G05).

- If programmed with a value of 0, no finishing pass will be carried out.

Q Indicates the direction of the finishing pass.The finishing pass on the islands is always carried out in the opposite direction.

Q = 0 The finishing pass is carried out in the same direction as the outside profile was programmed.

Q = 1 The finishing pass is carried out in the opposite direction to the one programmed.

Q = 2 Reserved.

Any other value will generate the corresponding error message. If parameter "Q" is not programmed, the cycle assumes Q0.

I(±5.5) Defines the total depth of the island and it is given in absolute coordinates.

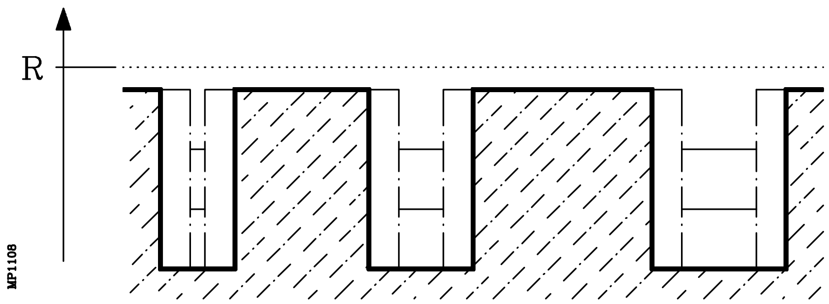
- If the island has a roughing operation, it is not necessary to define this parameter since it has been programmed in that operation. However, if programmed in both operations, the canned cycle will assume the particular depth indicated for each operation.

- If the island has no roughing operation, it is necessary to define this parameter.

R (±5.5) Defines the coordinate of the reference plane and it is given in absolute values.

- If the island has a roughing operation, it is not necessary to define this parameter since it has been programmed in that operation. However, if programmed in both operations, the canned cycle will assume the particular depth indicated for each operation.

- If the island has no roughing operation, it is necessary to define this parameter.



K(1) Defines the type of profile intersection to be used.

0 = Basic profile intersection.
1 = Advanced profile intersection.

If the island has a roughing operation, it is not necessary to define this parameter since it has been programmed in that operation. However, if programmed in both operations, the canned cycle will assume the one defined for the roughing operation.

If no roughing operation has been defined and this parameter is not programmed, the canned cycle will assume a K0 value. Both types of intersection are described later on.

V (5.5) Defines the tool penetrating feedrate.

If not programmed or programmed with a value of "0", the CNC will assume 50% of the feedrate in the plane (F).

F (5.5) Optional. Defines the machining feedrate in the plane.

S (5.5) Optional. Defines the spindle speed.

T (4) Defines the tool used for the roughing operation. It must be programmed.

D (4) Optional. Defines the tool offset number.

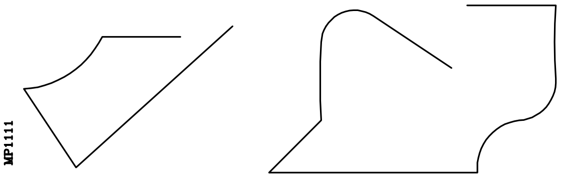
M Optional. Up to 7 miscellaneous M functions can be programmed.

This operation allows M06 with an associated subroutine to be defined, and the tool change is performed before beginning the roughing operation.

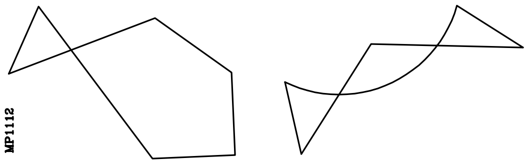
11.1.4 *PROFILE PROGRAMMING RULES*

When outside and inside profiles of an irregular pocket are programmed the following programming rules must be followed:

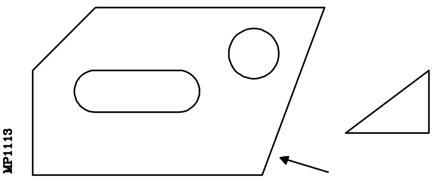
1.- All types of programmed profiles must be closed. The following examples cause a geometry error.



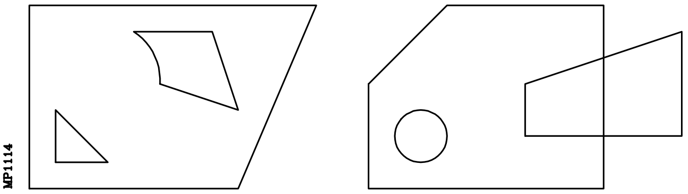
2.- No profile must intersect itself. The following examples cause a geometry error.



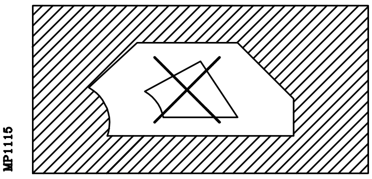
3.- When more than one outside profile has been programmed, the canned cycle assumes the one occupying the largest surface.



4.- It is not required to program inside profiles. Should these be programmed, they must be partially or totally internal with respect to the outside profile. Some examples are given below.



5.- An internal profile totally contained within another internal profile cannot be programmed. In this case, only the most external profile will be considered.



The canned cycle will verify all these geometry rules before beginning to make the pocket adapting the profile of the pocket to them and displaying the error message when necessary.

11.1.5 INTERSECTION OF PROFILES

In order to facilitate the programming of profiles, the canned cycle allows the profiles to intersect one another and the external profile.

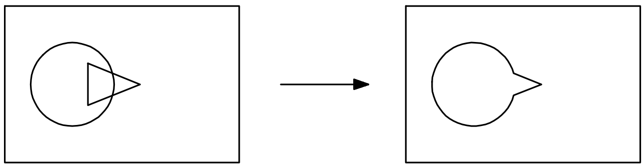
The two available types of intersection can be selected by parameter "**K**"

11.1.5.1 BASIC PROFILE INTERSECTION (K=0)

When selecting this type, the following profile intersecting rules are to be followed:

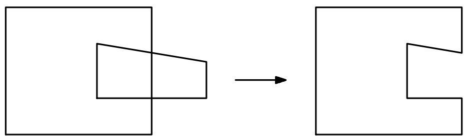
1.- The intersection of islands generates a new inside profile which is their boolean union. Example:

MP1116



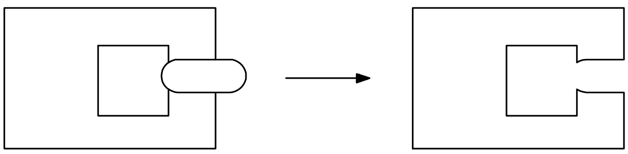
2.- The intersection between an internal and an external profile generates a new external profile as a result of the difference between the external and the internal profiles. Example:

MP1117



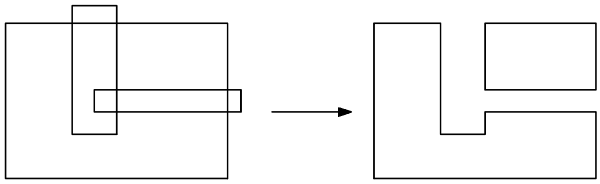
3.- If there is an inside profile which has an intersection with another inside profile and with the external profile, the canned cycle first makes the intersection between the inside profiles and then the intersection of these with the external profile.

MP1118



4.- As a result of the intersection of the inside profiles with the outside one, a single pocket will be obtained which corresponds to the outside profile having the largest surface. The rest will be ignored.

MP1119



5.- If the finishing operation has been programmed, the profile of the resulting pocket must comply with all the tool compensation rules, since if a profile is programmed which cannot be machined by the programmed finishing tool, the CNC will show the corresponding error.

11.1.5.2 *ADVANCED PROFILE INTERSECTION (K=1)*

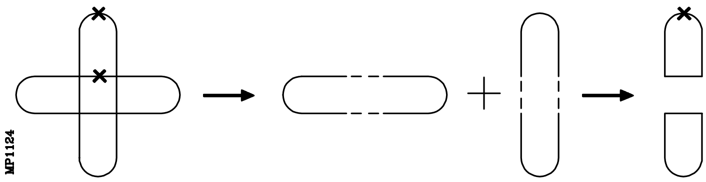
When selecting this type, the following profile intersecting rules are to be followed:
1.- The initial point of each contour determines the section to be selected.

In a profile intersection, each contour is divided into several lines that could be grouped as:

- Lines external to the other contour.
- Lines internal to the other contour.

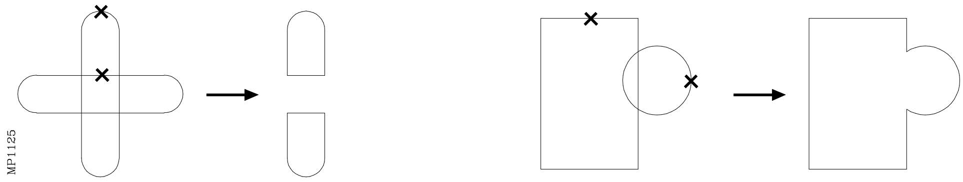
This type of profile intersection selects in each contour the group of lines where the profile defining point is included.

The following example shows the explained selection process. The solid lines indicate the lines external to the other contour and the dashes indicate the internal lines. The initial point of each contour is indicated with an "x".

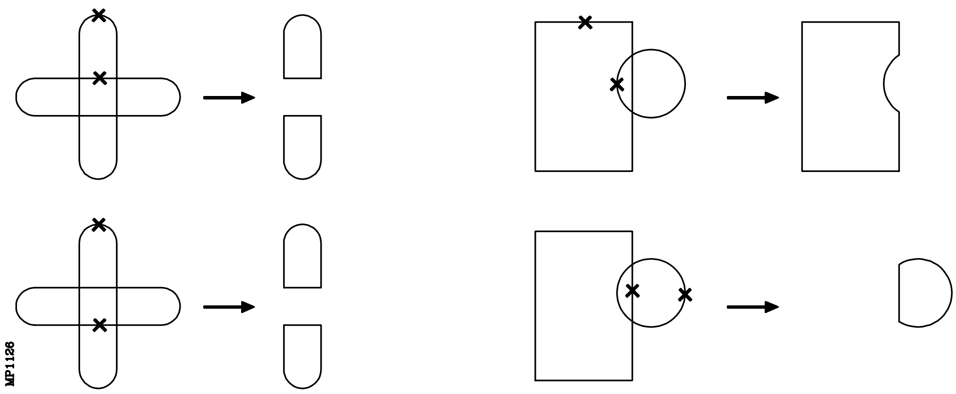


Examples of profile intersections:

Boolean Addition:



Boolean Subtraction:



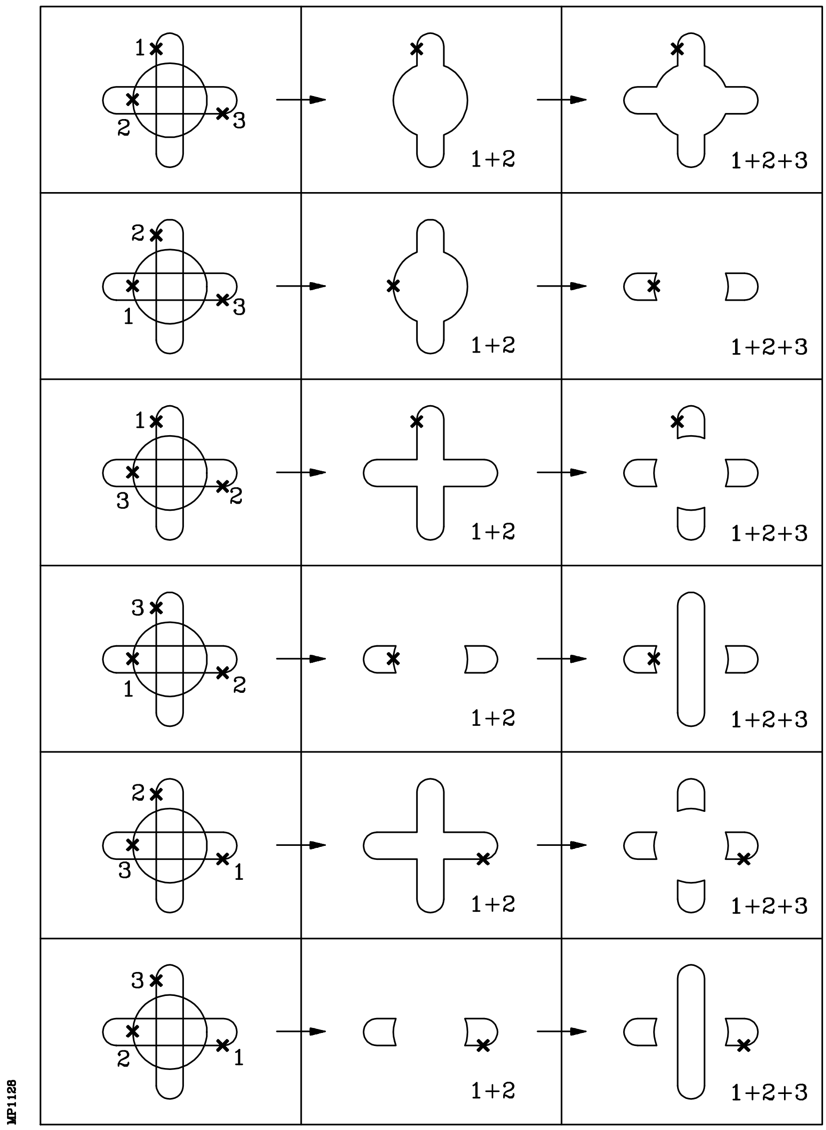
Boolean Intersection:

2.- The programming sequence for the different profiles is determinant when having an intersection of more than 2 profiles.

The profile intersection process is performed according to the order in which the profiles have been programmed. This way, the result of the intersection between the first two will be intersected with the third one and so forth.

The initial point of the resulting profiles always coincides with the initial point which defined the first profile.

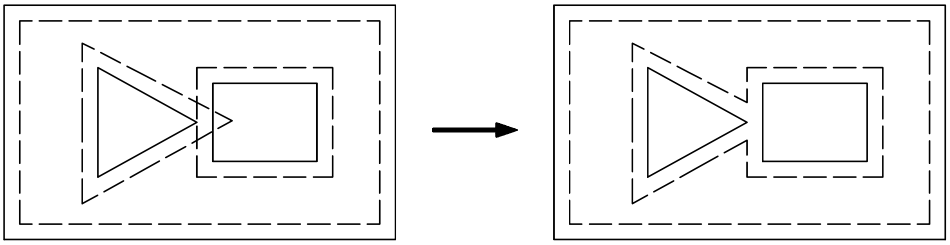
Examples:



11.1.5.3 RESULTING PROFILE

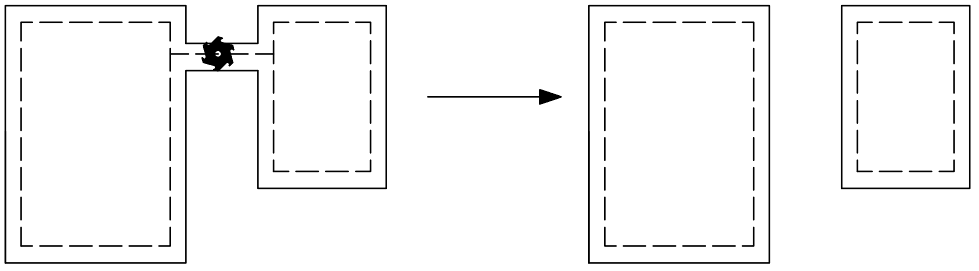
Once the profiles of the pocket and islands have been obtained, the canned cycle calculates the remaining profiles according to the radius of the roughing tool and the programmed finishing stock.

It may occur that in this process intersections are obtained which do not appear among the programmed profiles. Example:



MP1120

If there is an area in which the roughing tool cannot pass, when the intersection is made between the offset of the profiles, several pockets will be obtained as a result, all of which will be machined. Example:



MP1121

11.1.6 PROFILE PROGRAMMING SYNTAX

The outside profile and the inside profiles or islands which are programmed must be defined by simple geometrical elements such as straight lines or arcs.

The first definition block (where the external profile starts) and the last (where the last profile defined ends) must be provided with the block label number. These label numbers will be those which indicate to the canned cycle the beginning and end of the geometric description of the profiles which make up the pocket.

Example:	G66 D100 R200 F300 **S400 E500**	;Definition of irregular pocket
	N400 G0 G90 X300 Y50 Z3	;Beginning of geometric description
	------ ----- ---- ---	
	N500 G2 G6 X300 Y50 I150 J0	;End of geometric description

The profile programming syntax must comply with the following rules:

1.- The external profile must begin in the first definition block of the geometric description of the part profiles. This block will be assigned a label number in order to indicate canned cycle G66 the beginning of the geometric description.

2.- The part surface coordinate will be programmed in this block.

3.- All the internal profiles which are required may be programmed, one after the other. Each of these must commence with a block containing the **G00** function (indicating the beginning of the profile).

<u>Warning:</u>

Care must be taken to program G01, G02 or G03 in the block following the definition of the beginning, as G00 is modal, thus preventing the CNC from interpreting the following blocks as the beginnings of a new profile.

4.- Once the definition of the profiles has been completed, a label number must be assigned to the last block programmed, in order to indicate the canned cycle G66 the end of the geometric description.
Example

G0 G17 G90 X-350 Y0 Z50
G66 D100 R200 F300 **S400 E500** .... ;Description of cycle
G0 G90 X0 Y0 Z50
M30

N400 G0 G90 X-260 Y-190 Z4.5 .... ;Beginning of first profile
....
....
G0 X230 Y170.................................... ;Beginning of another profile
G1........
....
G0 X-120 Y90 .... ;Beginning of another profile
G2........
....
N500 G1 X-120 Y90................................. ;End of geometric description

5.- Profiles are described as programmed paths, it being possible to include corner rounding, chamfers, etc., following the syntax rules defined for this purpose.

6.- Mirror images, scaling factor changes, rotation of coordinate system, zero offsets, etc., cannot be programmed in the description of profiles.

7.- Nor is it possible to program blocks in high level language, such as jumps, subroutine calls or parametric programming.

8.- Other canned cycles cannot be programmed.

In addition to the G00 function, which has a special meaning, the irregular pocket canned cycle allows the use of the following functions for the definition of profiles.

G01	Linear interpolation
G02	Clockwise circular interpolation
G03	Counter-clockwise circular interpolation
G06	Arc center in absolute coordinates
G08	Arc tangent to previous path.
G09	Arc defined by three points
G36	Controlled corner rounding
G39	Chamfer
G53	Programming with respect to machine reference zero (home)
G70	Programming in inches
G71	Programming in millimeters
G90	Absolute programming
G91	Incremental programming
G93	Polar origin preset

11.1.7 ERRORS

The CNC will issue the following errors:

ERROR 1023 : G67. Tool radius too large.
When selecting a wrong roughing tool.

ERROR 1024 : G68. Tool radius too large.
When selecting a wrong finishing tool.

ERROR 1025 : A tool of no radius has been programmed.
When using a tool with "0" radius while machining a pocket.

ERROR 1026 : A step greater than the tool diameter has been programmed.
When parameter "C" of the roughing operation is greater than the diameter of the roughing tool.

ERROR 1041 : A mandatory parameter not programmed in the canned cycle.
It comes up in the following instances:

- When parameters "I" and "R" have not been programmed in the roughing operation.
- When not using a roughing operation and not programming the "I" and "R" parameters for the finishing operation.

ERROR 1042 : Wrong canned cycle parameter value.
It comes up in the following instances:

- When parameter "Q" of the finishing operation has the wrong value.
- When parameter "B" of the finishing operation has a "0" value.
- When parameter "J" of the finishing operation has been programmed with a value greater than the finishing tool radius.

ERROR 1044 : The plane profile intersects itself in an irregular pocket with islands.
It comes up when any of the plane profiles of the programmed contours intersects itself.

ERROR 1046 : Wrong tool position prior to the canned cycle.
It comes up when calling the G66 cycle if the tool is positioned between the reference plane and the depth coordinate (bottom) of any of the operations.

ERROR 1047 : Open plane profile in an irregular pocket with islands.
It comes up when any of the programmed contours does not begin and end at the same point. It may be because G1 has not been programmed after the beginning, with G0, on any of the profiles.

ERROR 1048 : The part surface coordinate (top) has not been programmed in an irregular pocket with islands.
It comes up when the first point of the geometry does not include the pocket top coordinate.

ERROR 1049 : Wrong reference plane coordinate for the canned cycle.
It comes up when the coordinate of the reference plane is located between the part's "top" and "bottom" in any of the operations.

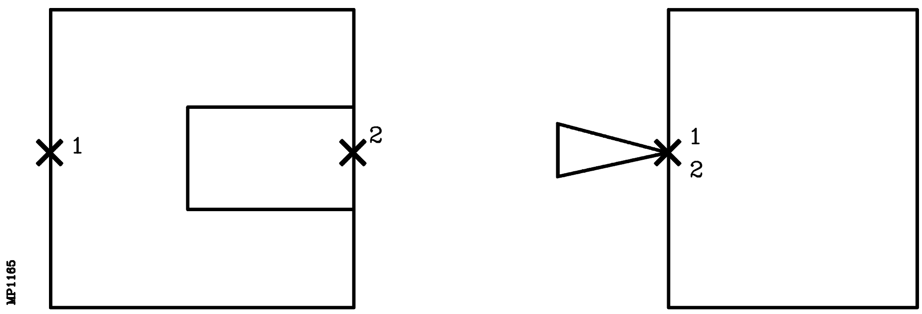
ERROR 1084 : Wrong circular path.

It comes up when any of the paths programmed in the geometry definition of the pocket is wrong.

ERROR 1227 : Wrong profile intersection in an irregular pocket with islands.

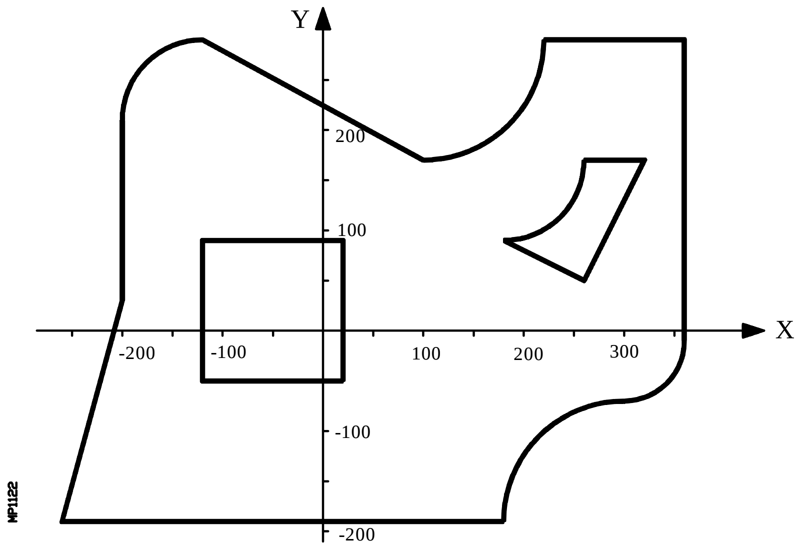
It comes up in the following instances:

- When two plane profiles have a common section (drawing on the left).
- When the initial points of two profiles in the main plane coincide (drawing on the right).



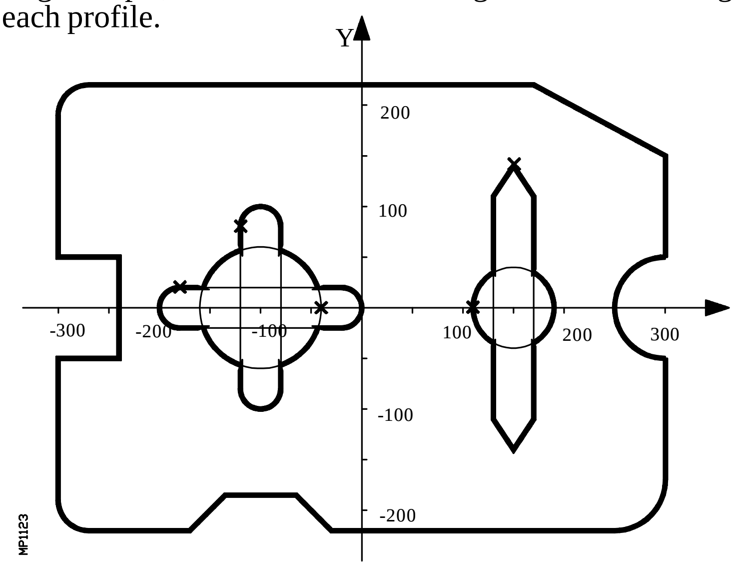
11.1.8 PROGRAMMING EXAMPLES

Programming example, without automatic tool changer




```
      (TOR1=5, TOI1=0, TOL1=25,TOK1=0)          ;Tool 1 dimensions
      (TOR2=3, TOI2=0, TOL2=20,TOK2=0)          ;Tool 2 dimensions
      (TOR3=5, TOI3=0, TOL3=25,TOK3=0)          ;Tool 3 dimensions
      G0 G17 G43 G90 X0 Y0 Z25 S800             ;Initial positioning
      G66 D100 R200 F300 S400 E500              ;Irregular pocket description
      M30                                       ;End of program

N100  G81 Z5 I-40 T3 D3 M6                      ;Definition of drilling operation
N200  G67 B20 C8 I-40 R5 K0 V100 F500 T1 D1 M6  ;Definition of roughing operation
N300  G68 B0 L0.5 Q0 V100 F300 T2 D2 M6         ;Definition of finishing operation

N400  G0 G90 X-260 Y-190 Z0                     ;Definition of pocket profiles
      G1 X-200 Y30                              ;(External profile)
      X-200 Y210
      G2 G6 X-120 Y290 I-120 J210
      G1 X100 Y170
      G3 G6 X220 Y290 I100 J290
      G1 X360 Y290
      X360 Y-10
      G2 G6 X300 Y-70 I300 J-10
      G3 G6 X180 Y-190 I300 J-190
      G1 X-260 Y-190

      G0 X230 Y170                              ;First island profile definition
      G1 X290 Y170
      X230 Y50
      X150 Y90
      G3 G6 X230 Y170 I150 J170

      G0 X-120 Y90                              ;Second island profile definition
      G1 X20 Y90
      X20 Y-50
      X-120 Y-50
N500  X-120 Y90                                 ;End of contour definition
```

Programming example, with automatic tool changer. The "x" of the figure indicate the initial points of each profile.




```
(TOR1=9, TOI1=0, TOL1=25,TOK1=0)          ;Tool 1 dimensions
(TOR2=3.6, TOI2=0, TOL2=20,TOK2=0)        ;Tool 2 dimensions
(TOR3=9, TOI3=0, TOL3=25,TOK3=0)          ;Tool 3 dimensions
G0 G17 G43 G90 X0 Y0 Z25 S800             ;Initial positioning
G66 D100 R200 F300 S400 E500              ;Irregular pocket description
M30                                       ;End of program

N100  G81 Z5 I-40 T3 D3 M6                     ;Definition of drilling operation
N200  G67 B10 C5 I-40 R5 K1 V100 F500 T1 D1 M6 ;Definition of roughing operation
N300  G68 B0 L0.5 Q1 V100 F300 T2 D2 M6        ;Definition of finishing operation

N400  G0 G90 X-300 Y50 Z3                 ;Definition of pocket profiles
      G1 Y190                             ;(External profile)
      G2 G6 X-270 Y220 I-270 J190
      G1 X170
      X300 Y150
      Y50
      G3 G6 X300 Y-50 I300 J0
      G1 G36 R50 Y-220
      X-30
      G39 R50 X-100 Y-150
      X-170 Y-220
      X-270
      G2 G6 X-300 Y-190 I-270 J-190
      G1 Y-50
      X-240
      Y50
      X-300

      G0 X-120 Y80                        ;First island contour definition
      G2 G6 X-80 Y80 I-100 J80            ;(Contour a)
      G1 Y-80
      G2 G6 X-120 Y-80 I-100 J-80
      G1 Y80
```

```
      G0 X-40 Y0                          ;(Contour b)
      G2 G6 X-40 Y0 I-100 J0
      G0 X-180 Y20                        ;(Contour c)
      G1 X-20
      G2 G6 X-20 Y-20 I-20 J0
      G1 X-180
      G2 G6 X-180 Y20 I-180 J0

      G0 X150 Y140                        ;Second island profile definition
      G1 X170 Y110                        ;(Contour d)
      Y-110
      X150 Y-140
      X130 Y-110
      Y110
      X150 Y140
      G0 X110 Y0                          ;(Contour e)
N500  G2 G6 X110 Y0 I150 J0               ;End of contour definition
```

11.2 3D POCKETS

The cycle calling function G66 is not modal; therefore, it must be programmed every time a 3D pocket is to be executed.

A block containing function G66 may not contain any other function. Its format is:

G66 R I C J F K S E

R (0-9999) & I (0-9999) Label number of the first block (R) and last block (I) defining the roughing operation.
When not setting "I" only block "R" is executed.
When not setting "R" there is no roughing operation.

C (0-9999) & J (0-9999) Label number of the first block (C) and last block (J) defining the semi-finishing operation.
When not setting "J" only block "c" is executed.
When not setting "C" there is no semi-finishing operation.

F (0-9999) & K (0-9999) Label number of the first block (F) and last block (K) defining the finishing operation.
When not setting "K" only block "F" is executed.
When not setting "F" there is no finishing operation.

S (0-9999) & E (0-9999) Label number of the first block (S) and last block (E) defining the geometry of the profiles forming the pocket.
Both parameters must be set.

Programming example:

```
      G00 G90 X100 Y200 Z50 F5000 T1 D2   ;Initial positioning
      M06
      G66 R100 C200 J210 F300 S400 E500   ;Definition of irregular pocket canned cycle
      M30                                 ;End of program

N100  G67 ............                    ;Defines the roughing operation
N200  ...................                 ;Starts the semi-finishing operation
      G67 ............
N210  ...................                 ;End the semi-finishing operation
N300  G68 ............                    ;Defines the finishing operation
N400  G0 G90 X300 Y50 Z3                  ;Starts the geometry description
      ...................
      ...................
N500  G2 G6 X300 Y50 I150 J0              ;End of geometry description
```

Basic operation:

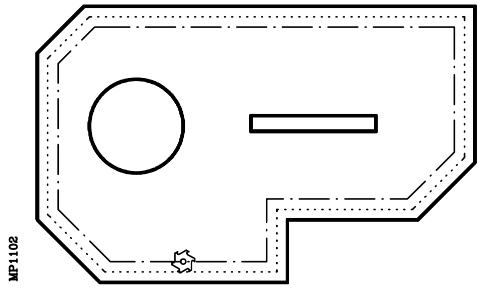
1.- Roughing operation. Only if it has been programmed.

It consists of several surface milling passes, until the total depth programmed has been reached. On each surface milling pass, the steps below will be followed depending on the type of machining that has been programmed:

Case A: When the machining paths are linear and maintain a certain angle with the abscissa axis.

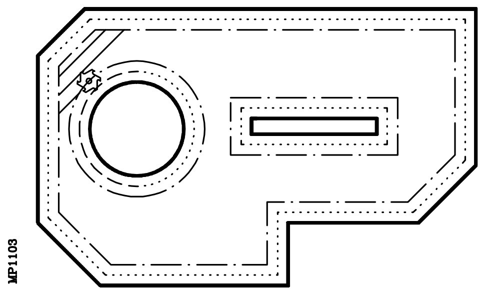
* It first contours the external profile of the part.

If the finishing operation has been selected on the cycle call, this contouring is performed leaving the finishing stock programmed for the finishing pass.

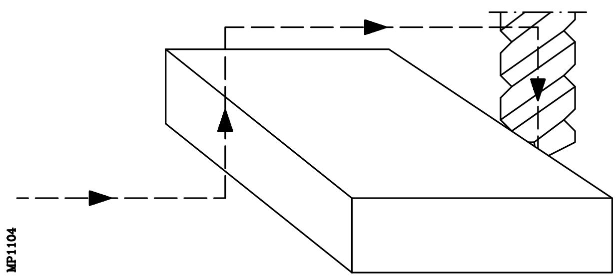


* Next the milling operation, with the programmed feed and steps.

If, while milling, an island is run into for the first time, it will be contoured.

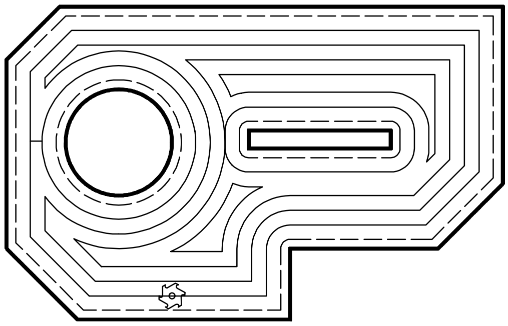


After the contouring and the remaining times, the tool will pass over the island, withdrawing along the longitudinal axis, to the reference plane, and will continue machining once the island has been cleared.



Case B: When using concentric machining paths

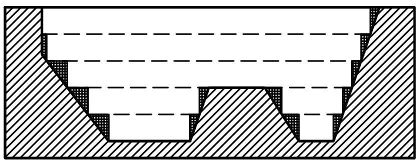
* The roughing operation is carried out following paths concentric to the profile. It will done as fast as possible without going over the islands if possible.



MP1129

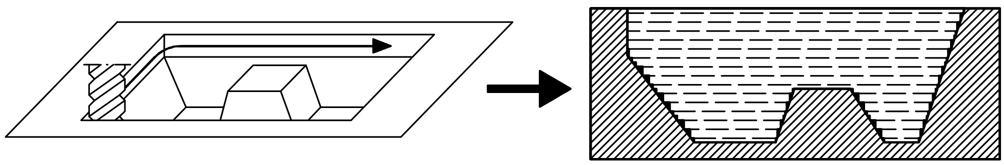
2.- Semi-finishing operation. Only if it has been programmed.

After the roughing, some ridges appear on the external profile as well as on the islands themselves as shown in the illustration below:



MP1133

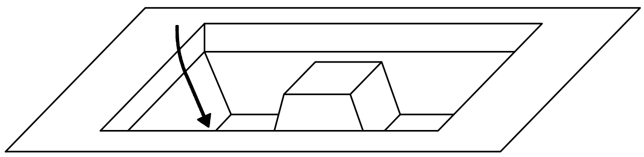
With the semi-finishing operation, it is possible to minimize these ridges by running several contouring passes at different depths.



MP1134

3.- Finishing operation. Only if it has been programmed.

It runs consecutive finishing passes in 3D. Either inward or outward machining direction may be selected or both may be alternated.



MP1135

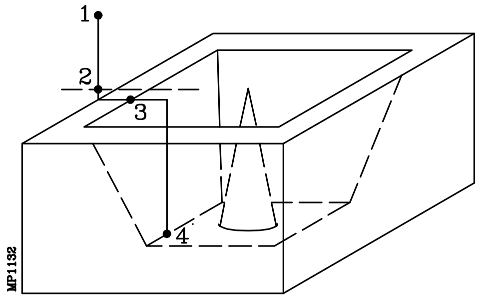
The CNC will machine both the outside profile and the islands by performing tangential entries and exits to them at constant surface speed.

After cycle conditions

Once the canned cycle has ended, the active feedrate will be the last one programmed. The one corresponding to the roughing or finishing operation. On the other hand, the CNC will assume functions G00, G40 and G90.

Reference coordinates

The irregular pocket canned cycle has four coordinates along the longitudinal axis, usually perpendicular to the plane (selected with G15), which, due to their importance, are described next:



1.- Starting plane coordinate. Given by the tool position at the beginning of the cycle.

2.- Reference plane coordinate. It must be programmed in absolute values and it represents a part approaching coordinate.

3.- Part surface coordinate (top). It is programmed in absolute values and in the first profile defining block.

4.- Machining depth coordinate (bottom. It must be programmed in absolute values.

11.2.1 ROUGHING OPERATION

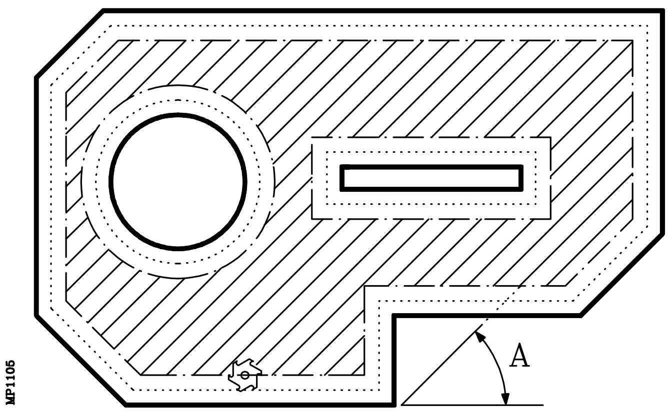
This is the main operation in the machining of an irregular pocket, and its programming is optional.

It will be programmed in a block which will need to bear a label number in order to indicate to the canned cycle the block where the roughing operation is defined.

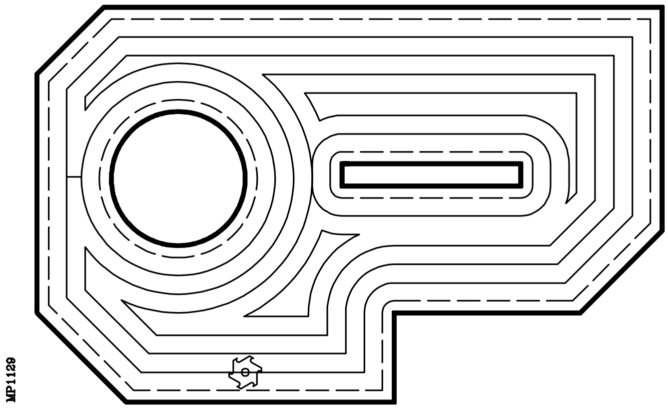
Example:	G66 **R100** C200 F300 S400 E500	; Definition of the irregular pocket cycle.
	N100 G67	; Definition of the roughing operation.

The function for the roughing operation is G67 and it cannot be executed independently from the G66.
Its programming format: **G67 A B C I R V F S T D M**

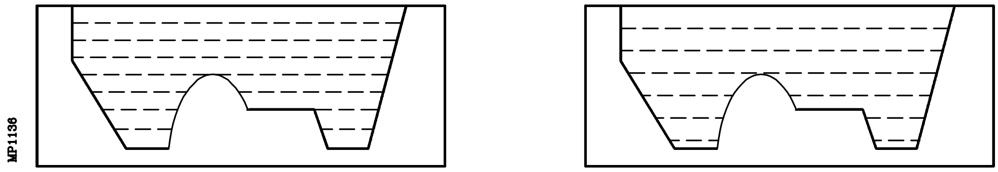
A(+/-5.5) Defines the angle which forms the roughing path with the abscissa axis.



If parameter "A" is not programmed, the roughing operation is carried out following concentric paths. It will be machined as fast as possible since it does not have to go over the islands.



B(+/-5.5) Defines the machining pass along the longitudinal axis (depth of the roughing pass). It <u>must be</u> defined and it must have a value other than 0; otherwise, the roughing operation will be cancelled.

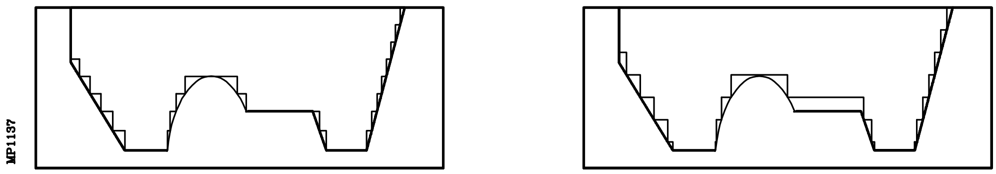


B(+) B(−)

- If programmed with a positive sign, all the roughing will be performed with the same machining pass, and the canned cycle calculates a pass equal to or smaller than the programmed pass.

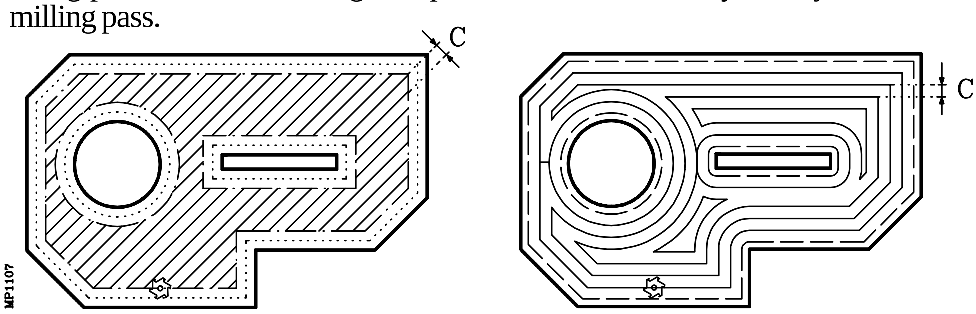
- If programmed with a negative sign, all the roughing will be performed with the programmed pass, and the canned cycle will adjust the last pass to obtain the total programmed depth.

- If done with B(+), the ridges will appear only on the pocket walls; but, if done with B(-), they could also show up above the islands.



B(+) B(−)

C(+/-5.5) Defines the milling pass in roughing along the main plane, the entire pocket being performed with the given pass, and the canned cycle adjusts the last milling pass.

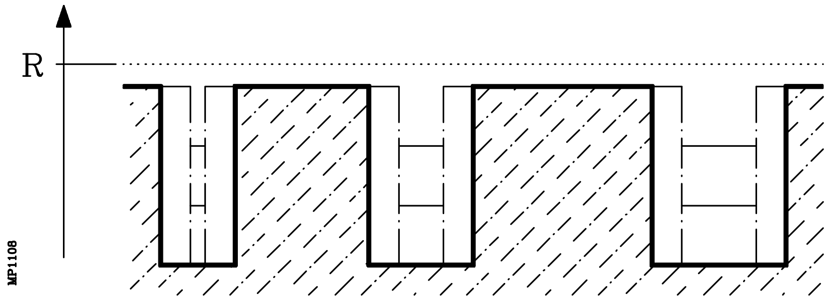


If it is not programmed or is programmed with either a value of 0, it will assume a value of 3/4 the diameter of the selected tool.

If programmed with a value greater than the tool diameter, the CNC will issue the corresponding error.

I(+/-5.5) Defines the total depth of the pocket and is programmed in absolute coordinates. It <u>must be</u> programmed.

R(+/-5.5) Defines the reference plane coordinate and is programmed in absolute coordinates. It <u>must</u> be programmed.



V (5.5) Defines the tool penetrating feedrate.

If not programmed or programmed with a value of "0", the CNC will assume 50% of the feedrate in the plane (F).

F (5.5) Optional. Defines the machining feedrate in the plane.

S (5.5) Optional. Defines the spindle speed.

T (4) Defines the tool used for the roughing operation. It <u>must be</u> programmed.

D (4) Optional. Defines the tool offset number.

M Optional. Up to 7 miscellaneous M functions can be programmed.

This operation allows M06 with an associated subroutine to be defined, and the tool change is performed before beginning the roughing operation.

11.2.2 SEMI-FINISHING OPERATION

This operation is optional.

It will be programmed in a block which will need to bear a label number in order to indicate to the canned cycle the block where the roughing operation is defined.

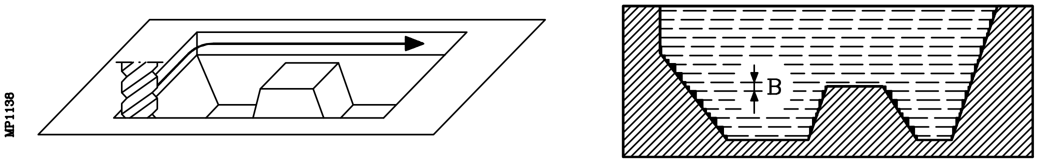
Example:	G66 R100 **C200** F300 S400 E500	; Definition of the irregular pocket cycle.
	N200 G67	; Definition of the semi-finish operation.

The function for the semi-finishing operation is G67 and it cannot be executed independently from the G66.

Both the roughing and the semi-finishing operations are defined with G67; but, in different blocks. It is function G66 who indicates which is which by means of parameters "R" and "C".

Its programming format is: **G67 B I R V F S T D M**

B (±5.5) Defines the machining step along the longitudinal axis (semi-finishing pass). It must be programmed and with a value other than "0". Otherwise, the semi-finishing operation will be canceled.



- If programmed with a positive sign, the whole semi-finish operation will be carried out with the same machining pass and the canned cycle will calculate a pass equal or smaller than the one programmed.

- If programmed with a negative sign, the whole semi-finish operation will be run with the programmed pass. The canned cycle will adjust the last pass to obtain the total programmed depth.

I (±5.5) Defines the total pocket depth and it is programmed in absolute coordinates.

If there is a roughing operation and it is not programmed, the CNC takes the value defined for the roughing operation.

If there is no roughing operation, it <u>must be</u> programmed.

R (±5.5) Defines the coordinate of the reference plane and it is programmed in absolute values.

If there is a roughing operation and it is not programmed, the CNC takes the value defined for the roughing operation.

If there is no roughing operation, it <u>must be</u> programmed.

V (5.5) Defines the tool penetrating feedrate.

If not programmed or programmed with a value of "0", the CNC will assume 50% of the feedrate in the plane (F).

F (5.5) Optional. Defines the machining feedrate in the plane.

S (5.5) Optional. Defines the spindle speed.

T (4) Defines the tool used for the semi-finishing operation. It must be programmed.

D (4) Optional. Defines the tool offset number.

M Optional. Up to 7 miscellaneous M functions can be programmed.

This operation allows M06 with an associated subroutine to be defined, and the tool change is performed before beginning the semi-finishing operation.

11.2.3 FINISHING OPERATION

This operation is optional.

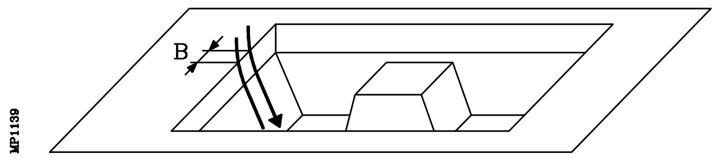
It will be programmed in a block which will need to bear a label number in order to indicate to the canned cycle the block where the roughing operation is defined.

Example: G66 R100 C200 **F300** S400 E500 ; Definition of the irregular pocket cycle.
N300 G67 .... ; Definition of the finishing operation.

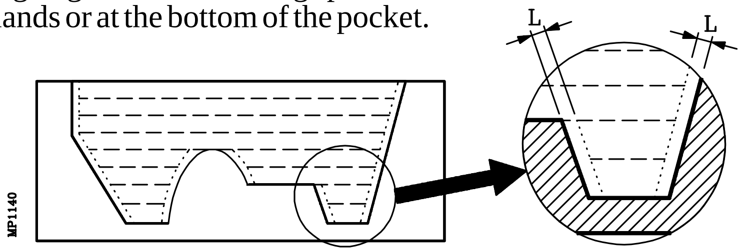
The function for the finishing operation is G68 and it cannot be executed independently from the G66.

Its programming format is: **G68 B L Q J I R V F S T D M**

B (5.5) Defines the pass in the plane between two 3D paths of the finishing operation. It must be defined and with a value other than "0".



L (±5.5) Defines the value of the finishing stock on the side walls of the pocket left by the roughing and semi-finishing operations. No residual stock is left on the surface of the islands or at the bottom of the pocket.



If programmed with a positive value, the finishing pass will be carried out in G7 (square corner). If programmed with a negative value, the finishing pass will be carried out in G5 (round corner). If not programmed, the cycle assumes "L0".

Q Indicates the direction of the finishing pass.

Q = 1 All the passes will be inward from the top of the pocket to its bottom

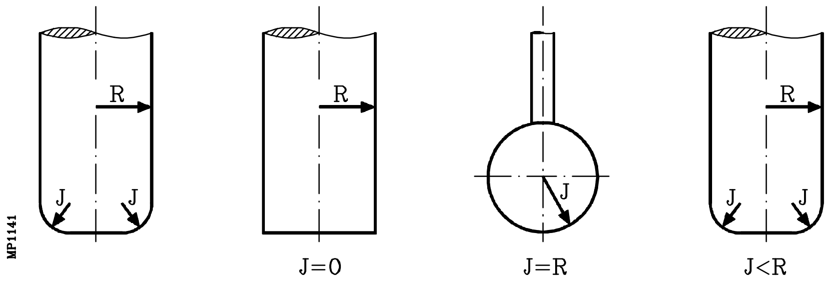
Q = 2 All the passes will be outward from the bottom of the pocket to the top.

Q = 0 Alternating direction for every 2 consecutive paths.

Any other value will generate the corresponding error. If parameter "Q" is not programmed, the cycle assumes "Q0".

J (5.5) Indicates the tool tip radius and, therefore, the type of finishing tool being used.

Depending on the radius assigned to the tool in the tool offset table (of the CNC variables: "TOR" + "TOI") and the value of assigned to this parameter, three tool types may be defined.



FLAT .... If J is not programmed or J = 0
BALL-END .... If J = R
TORIC (Corner rounding) ..... If J <> 0 (other than 0) and J < R

I (±5.5) Defines the total pocket depth and it is given in absolute coordinates.

- If defined, the cycle will take it into account during the finishing operation.
- If not defined and the pocket has a roughing operation, the cycle will assume the value defined for the roughing operation.
- If not defined and the pocket has no roughing operation, but it has a semi-finishing operation, the cycle will assume the one define in the semi-finishing operation.
- If the pocket has neither roughing nor semi-finishing operation, this parameter must be defined.

R (±5.5) Defines the coordinate of the reference plane and it must be given in absolute values.

- If defined, the cycle will take it into account during the finishing operation.
- If not defined and the pocket has a roughing operation, the cycle will assume the value defined for the roughing operation.
- If not defined and the pocket has no roughing operation, but it has a semi-finishing operation, the cycle will assume the one define in the semi-finishing operation.
- If the pocket has neither roughing nor semi-finishing operation, this parameter must be defined.

V (5.5) Defines the tool penetrating feedrate.

If not programmed or programmed with a value of "0", the CNC will assume 50% of the feedrate in the plane (F).

F (5.5) Optional. Defines the machining feedrate in the plane.

S (5.5) Optional. Defines the spindle speed.

T (4) Defines the tool used for the finishing operation. It must be programmed.

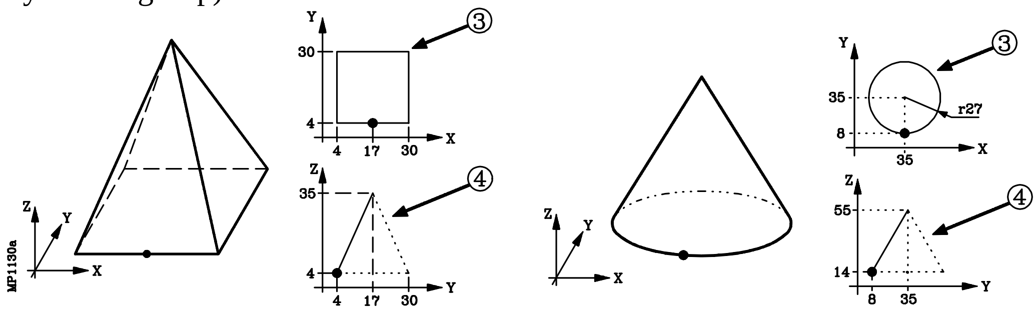
D (4) Optional. Defines the tool offset number.

M Optional. Up to 7 miscellaneous M functions can be programmed.

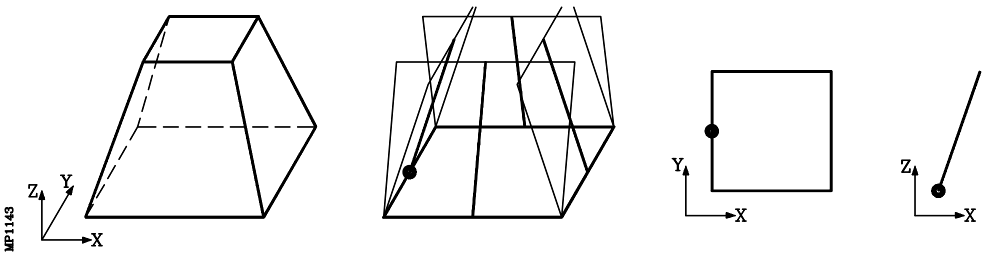
This operation allows M06 with an associated subroutine to be defined, and the tool change is performed before beginning the finishing operation.

11.2.4 PROFILE OR CONTOUR GEOMETRY

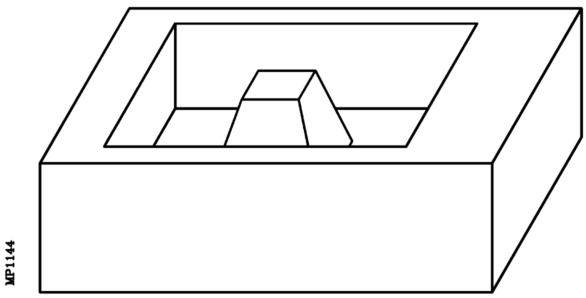
To define the contours or profiles of a 3D pocket, one must specify the plane profile or horizontal cross section (3) and the depth profile or vertical cross section (4) of all contours (even when they are straight up).



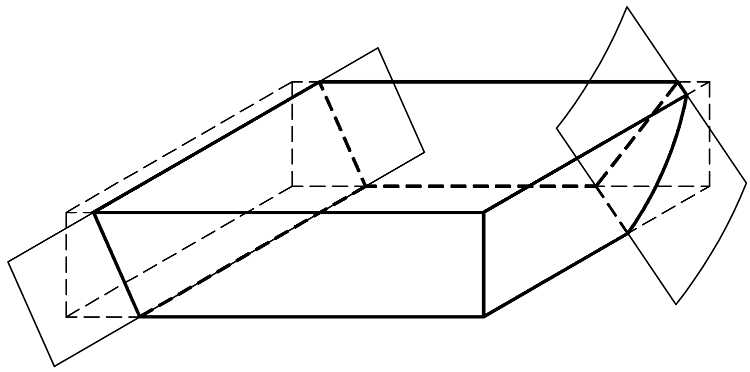
Since the canned cycle applies the same depth profile to the whole contour, the same start point must be used to define the plane profile as for the depth profile.



Example of a 3D pocket:



3D contours with more than one depth profile are also possible. These contours are called "composite 3D profiles" and will be described later on.



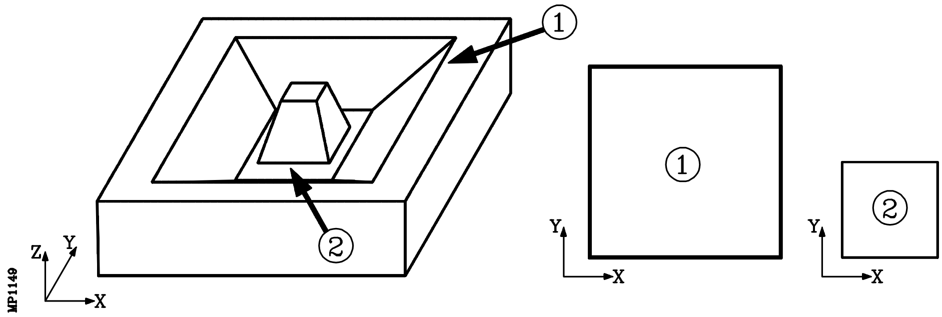
11.2.5 PROFILE PROGRAMMING RULES

When programming inside or outside contours of an irregular 3D pocket (with islands) , the following rules must be complied with:

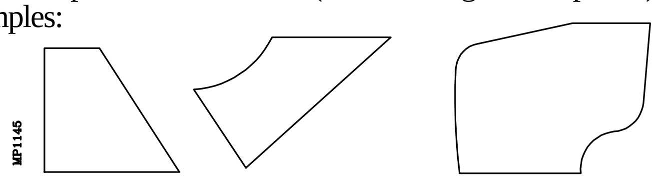
1.- The profile in the main plane indicates the shape of the contour.

Since a 3D contour has an infinite number of different profiles (1 per each depth coordinate), the following must be programmed:

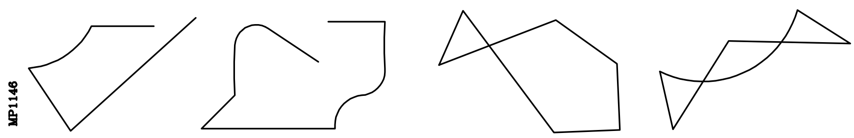
* For the outside contour of the pocket: the one corresponding to the surface coordinate or top of the part (1).
* For the inside contours: the one corresponding to the base or bottom (2).



2.- The profile in the plane must be closed (same starting and end points) and it <u>must not</u> intersect itself. Examples:



The following examples cause a geometry error:



3.- The depth profile (vertical cross section) must be programmed with any of the axes of the active plane. If the active plane is the XY and the perpendicular axis is the Z axis, one must program: G16XZ or G16YZ.

All profiles, plane and depth, must start with the definition of the plane containing it.

Example:
G16 XY .... Beginning of the outside profile definition
----- *plane profile definition* -----
G16 XZ
------ *depth profile definition* ----

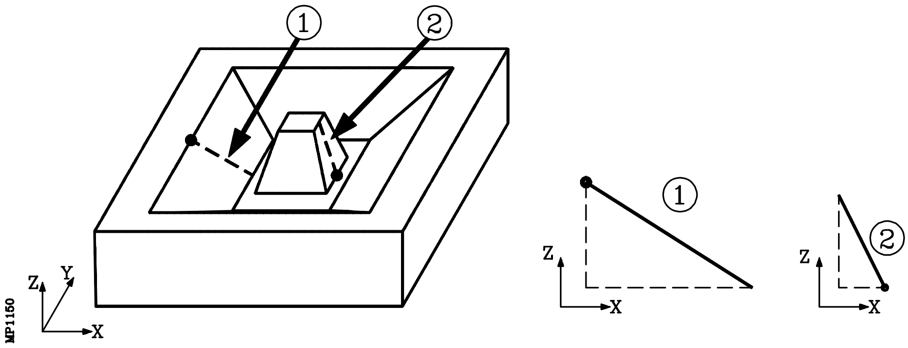
G16 XY .... Beginning of the island definition
----- *plane profile definition* -----
G16 XZ
------ *depth profile definition* ----

4.- The depth profile must be defined after having defined the plane profile.

The beginning points of the plane profile and depth profile must be the same one.

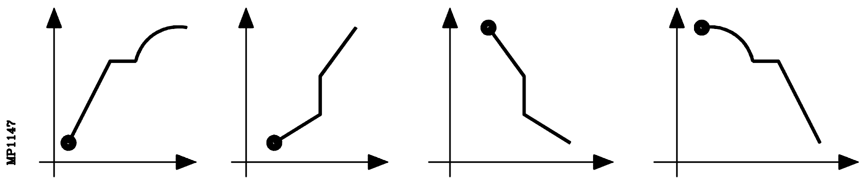
Nevertheless, the depth profile must be programmed:

* For the outside contour of the pocket starting from the top or surface coordinate (1).
* For the inside contours, islands, starting from the bottom or base coordinate (2).

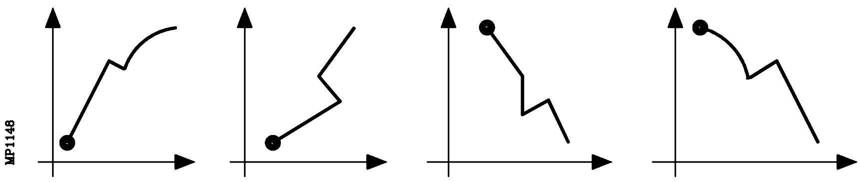


5.- The depth profile must be open and without direction changes along its path. In other words, it cannot zig-zag.

Examples:

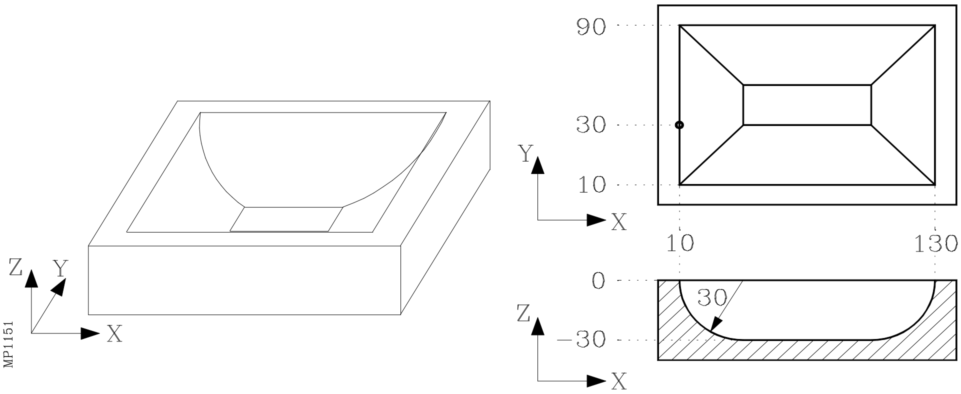


The following examples cause geometry errors.



11.2.5.1 PROGRAMMING EXAMPLES

<u>Example of a pocket without islands:</u>

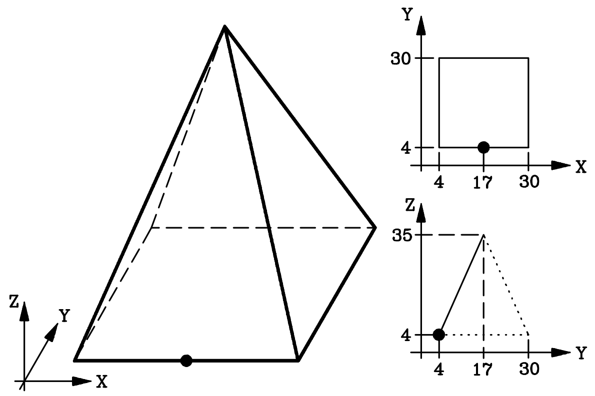



```
        (TOR1=2.5,TOL1=20,TOI1=0,TOK1=0)
        G17 G0 G43 G90 Z50 S1000 M4
        G5
        G66 R200 C250 F300 S400 E500 ............................ ;3D pocket definition
        M30

N200    G67 B5 C4 I-30 R5 V100 F400 T1D1 M6 ............................;Roughing operation
N250    G67 B2 I-30 R5 V100 F550 T2D1 M6 .................................;Semi-finishing operation
N300    G68 B1.5 L0.75 Q0 I-30 R5 V80 F275 T3D1 M6 .................;Finishing operation

N400    G17 ........................................................................ ;Beginning of the pocket geometry definition
        G90  G0 X10 Y30 Z0 ............................................. ;Plane profile (horizontal cross section)
        G1   Y90
             X130
             Y10
             X10
             Y30
        G16  XZ ................................................................. ;Depth profile (vertical cross section)
        G0   X10 Z0
N500    G3   X40 Z-30 I30 K0 ........................................... ;End of the pocket geometry definition
```

Profile definition examples:

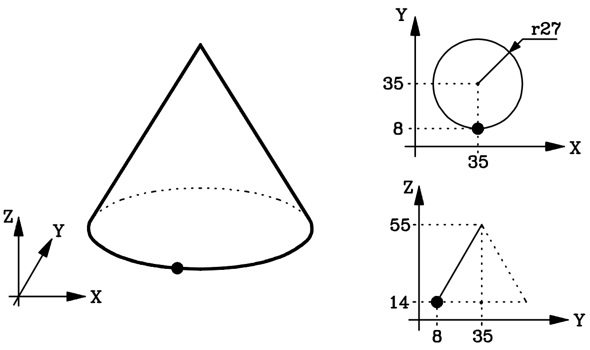
Pyramid Island

Plane profile
G17
G0 G90 X17 Y4
G1 X30
G1 Y30
G1 X4
G1 Y4
G1 X17
Depth profile
G16 YZ
G0 G90 Y4 Z4
G1 Y17 Z35



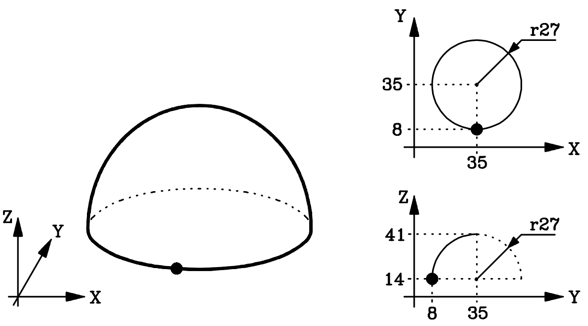
Conic Island

Plane profile
G17
G0 G90 X35 Y8
G2 X35 Y8 I0 J27
Depth profile
G16 YZ
G0 G90 Y8 Z14
G1 Y35 Z55

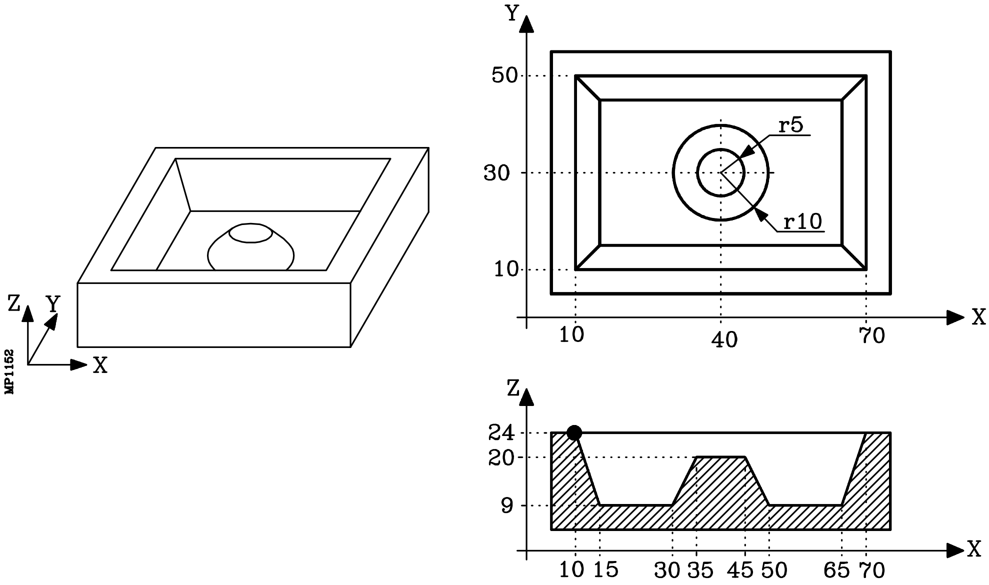


Semi-spherical Island

Plane profile
G17
G0 G90 X35 Y8
G2 X35 Y8 I0 J27
Depth profile
G16 YZ
G0 G90 Y8 Z14
G2 Y35 Z41 R27



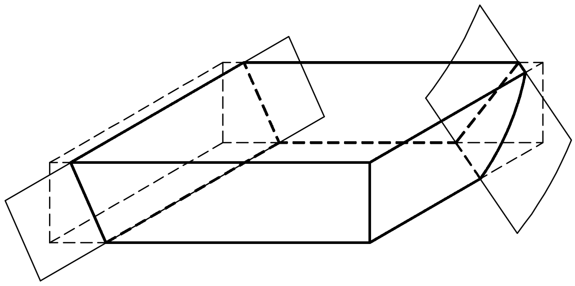
Example of a 3D pocket with islands:




```
        (TOR1=2.5,TOL1=20,TOI1=0,TOK1=0)
        G17 G0 G43 G90 Z50 S1000 M4
        G5
        G66 R200 C250 F300 S400 E500 ............................ ;3D pocket definition
        M30

N200    G67 B5 C4 I9 R25 V100 F400 T1D1 M6 .............................;Roughing operation
N250    G67 B2 I9 R25 V100 F550 T2D1 M6 ..................................;Semi-finishing operation
N300    G68 B1.5 L0.75 Q0 I9 R25 V50 F275 T3D1 M6 ...................;Finishing operation

N400    G17 ....................................................................... ;Beginning of the pocket geometry definition
        G90  G0 X10 Y30 Z24 ........................................... ;Outside contour (plane profile)
        G1   Y50
             X70
             Y10
             X10
             Y30
        G16  XZ ................................................................ ;Depth profile
        G0   X10 Z24
        G1   X15 Z9

        G17  ..................................................................... ;Island definition
        G90  G0 X30 Y30 ................................................... ;Plane profile
        G2   X30 Y30 I10 K0
        G16  XZ ................................................................ ;Depth profile
        G90  G0 X30 Z9
N500    G1   X35 Z20 ........................................................ ;End of the pocket geometry definition
```

11.2.6 *COMPOSITE 3D PROFILES*

A composite 3D profile is a 3D contour with more than one depth profile.

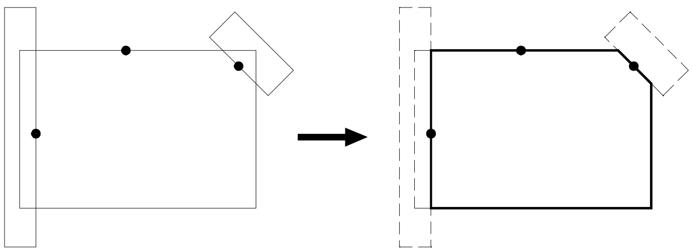


It is defined by means of the intersection of several contours with different depth profiles.

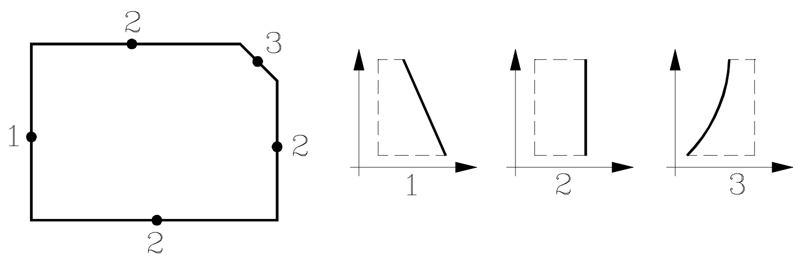
Each contour is defined by a profile in the plane and a depth profile. All the contours must meet the following conditions:

- ? The plane profile must contain the corresponding sides completely.
- ? Only a depth profile per contour must be defined.
- ? The plane profile and the depth profile of the contour gathering several sides must start at the same point.

The resulting plane profile will be formed by the intersection of the plane profiles of each element or contour.



Each wall of the resulting profile will assume the corresponding depth profile.



11.2.6.1 PROFILE INTESECTING RULES

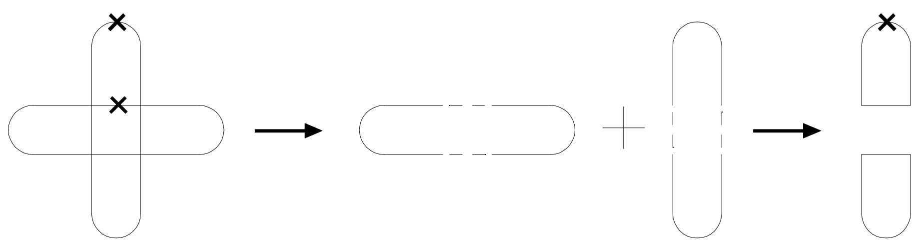
The plane profile intersecting rules are:

1.- At a profile intersection, each contour is divided into several lines which could be grouped as:

- Lines external to the other contour.
- Lines internal to the other contour.

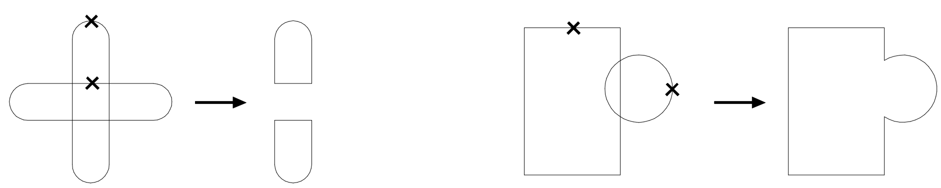
The starting point of each contour (x) determines the group of lines to be selected.

The following example shows the selection process using a solid line for the lines external to the other contour and a dotted line the internal ones.

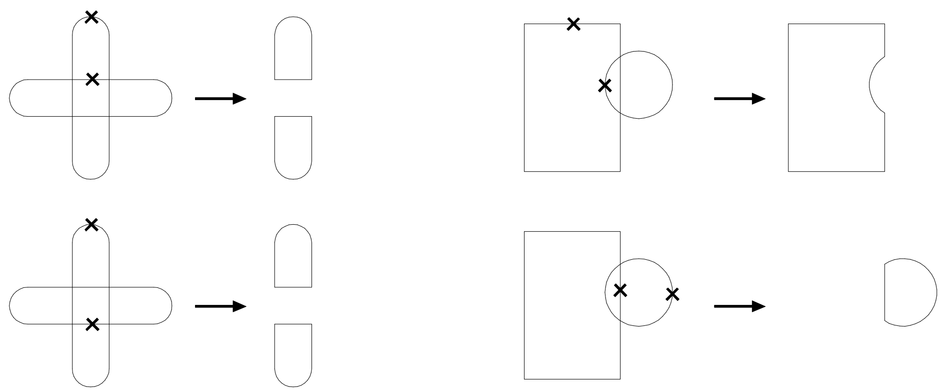


Profile intersection examples:

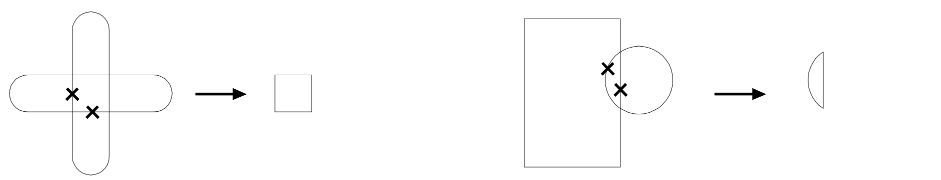
Boolean addition



Boolean subtraction



Boolean intersection

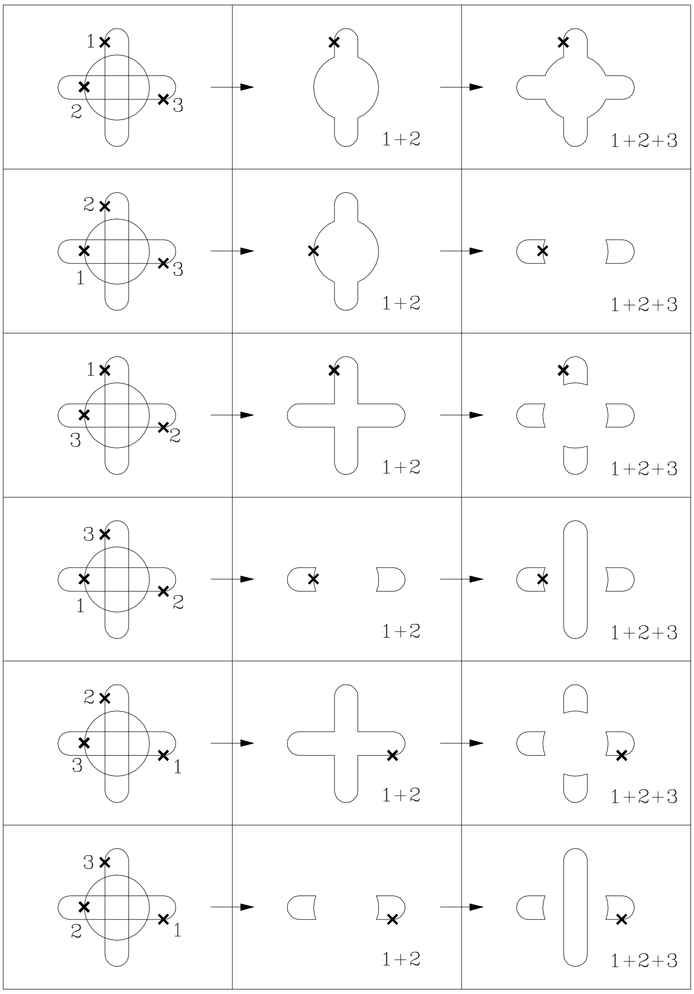


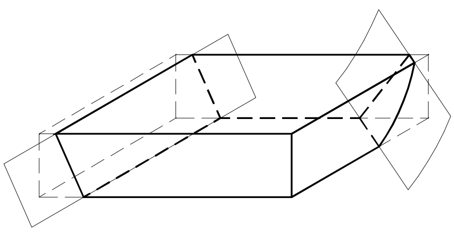
2.- The programming order of the various profiles is a determining factor when caring out an intersection of 3 or more profiles.

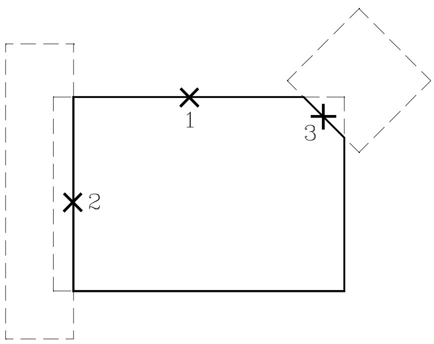
The profile intersecting process is done according the order (sequence) followed when programming the profiles. This way, after doing the intersection of the two profiles programmed first, the resulting profile will be intersected with the third one and so on.

The starting point of the resulting profiles always coincides with the starting point used to define the first profile.

Examples:



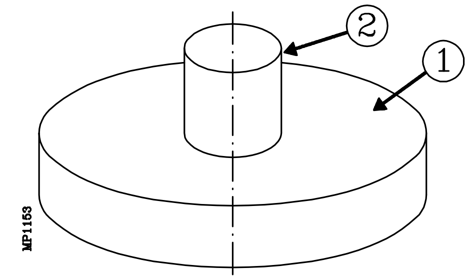




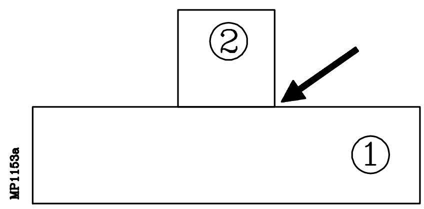
11.2.7 STACKED PROFILES

When 2 or more profiles stack on top of each other) the following considerations must be taken into account.

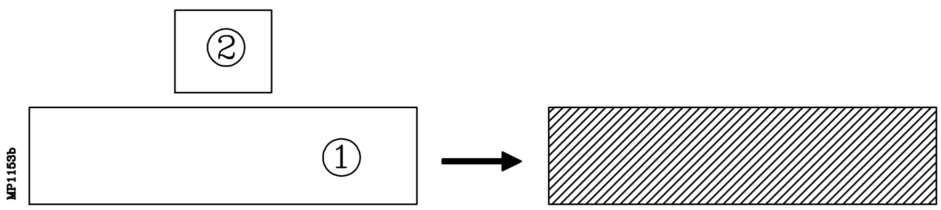
For clarity sakes, refer to the drawing on the right which consists of 2 stacked profiles: 1 and 2.



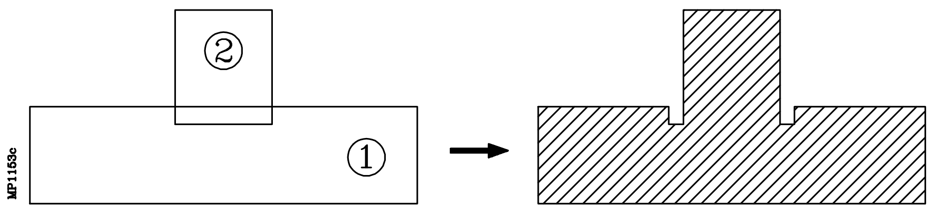
The base coordinate of the top profile (2) must coincide with the surface coordinate of the bottom profile (1).



If there is a gap between them, the cycle will consider that they are 2 different profiles and it will eliminate the top profile when executing the bottom one.



If the profiles mix, the canned cycle will make a groove around the top profile when running the finishing pass.



11.2.8 *PROFILE PROGRAMMING SYNTAX*

The outside profile and the inside profiles or islands which are programmed must be defined by simple geometrical elements such as straight lines or arcs.

The first definition block (where the external profile starts) and the last (where the last profile defined ends) must be provided with the block label number. These label numbers will be those which indicate to the canned cycle the beginning and end of the geometry description of the profiles which make up the pocket.

Example:

```
G66 R100 C200 F300 S400 E500     ;Irregular pocket canned cycle definition
N400 G17                         ;Beginning of geometry description
------ ----- ---- ---
N500 G2 Y50 Z-15 I10 K0          ;End of geometry description
```

The profile programming syntax must comply with the following rules:

1.- The first profile defining block must have a label number to indicate to the G66 canned cycle the beginning of the geometry description.

2.- First, the outside pocket contour must be defined and, then, the contour of each island.

3.- When a contour has more than one depth profile, the contours must be defined one by one indicating, on each one, the plane profile and, then, its depth profile.

4.- The first profile defining block of the plane profile as well as that of the depth profile must contain function G00 (indicative of the beginning of the profile).

Care must be taken to program G01, G02 or G03 in the block following the definition of the beginning, as G00 is modal, thus preventing the CNC from interpreting the following blocks as the beginnings of a new profile.

5.- The last profile defining block must have a label number to indicate to the G66 canned cycle the end of the geometry description.

Example:

```
     G66 R200 C250 F300 S400 E500 ......... ;3D pocket definition

N400 G17 .............................................. ;Beginning of the pocket geometry description
     G0  G90 X5 Y-26 Z0 ............................ ;Outside contour (plane profile)
     --- ---- ---- ----
     --- ---- ---- ----
     G16 XZ ................................................ ;Depth profile
     G0
     --- ---- ---- ----
     --- ---- ---- ----
     G17 ;Island
     G0 X30 Y-6 .......................................... ;Plane profile
     --- ---- ---- ----
     --- ---- ---- ----
     G16 XZ ................................................ ;Depth profile
     G0
     --- ---- ---- ----
     --- ---- ---- ----
N500 G3 Y-21 Z0 J-5 K0 ......................... ;End of the pocket geometry description
```

6.- Profiles are described as programmed paths, it being possible to include corner rounding, chamfers, etc., following the syntax rules defined for this purpose.

7.- Mirror images, scaling factor changes, rotation of coordinate system, zero offsets, etc., cannot be programmed in the description of profiles.

8.- Nor is it possible to program blocks in high level language, such as jumps, subroutine calls or parametric programming.

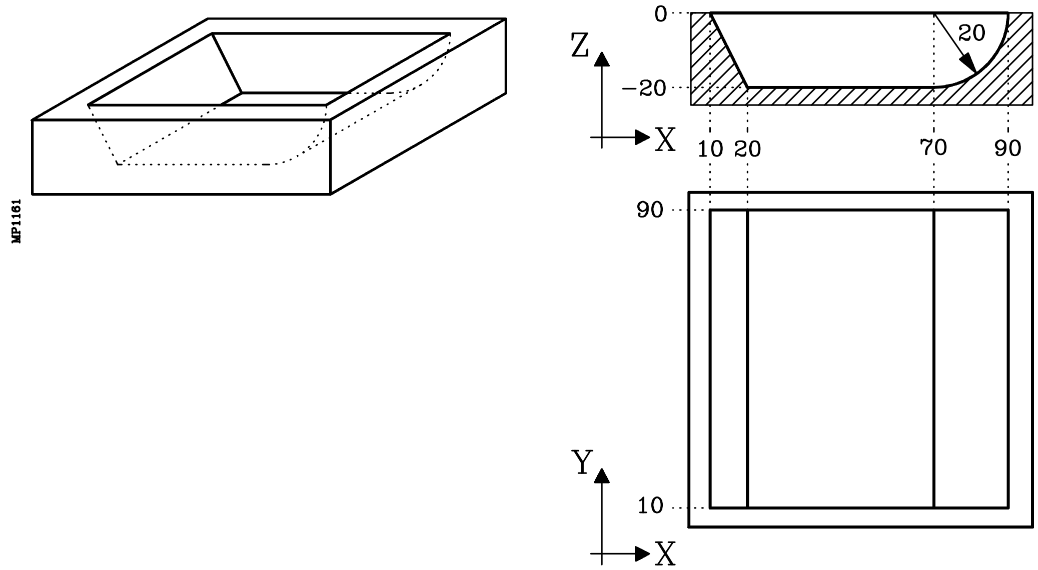
9.- Other canned cycles cannot be programmed.

In addition to the G00 function, which has a special meaning, the irregular pocket canned cycle allows the use of the following functions for the definition of profiles.

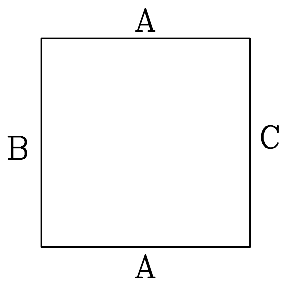
G01 Linear interpolation
G02 Clockwise circular interpolation
G03 Counter-clockwise circular interpolation
G06 Arc center in absolute coordinates
G08 Arc tangent to previous path.
G09 Arc defined by three points
G16 Main plane section by two directions
G17 Main plane X-Y and longitudinal Z (perpendicular)
G18 Main plane Z-X and longitudinal Y (perpendicular)
G19 Main plane Y-Z and longitudinal X (perpendicular)
G36 Automatic radius blend (controlled corner rounding)
G39 Chamfer
G53 Programming with respect to machine reference zero (home)
G70 Programming in inches
G71 Programming in millimeters
G90 Absolute programming
G91 Incremental programming
G93 Polar origin preset

11.2.9 EXAMPLES

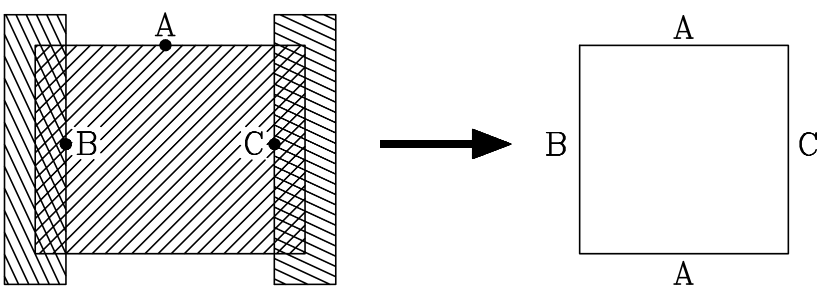
Example 1, Pocket without islands:



In this example, the island has 3 types of depth profiles: A, B and C.

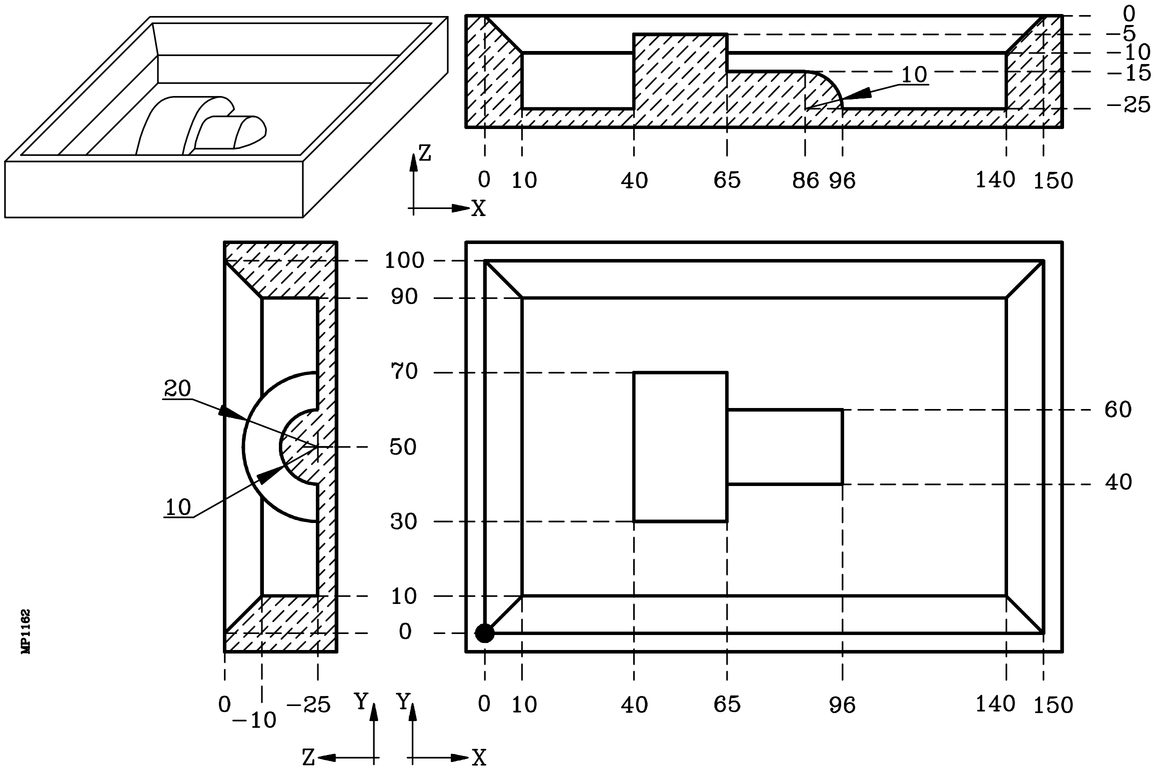


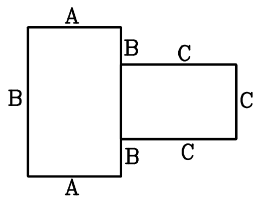
3 contours are used to define the island: A-type contour, B-type contour and C-type contour.



```
       (TOR1=2.5,TOL1=20,TOI1=0,TOK1=0)
       G17 G0 G43 G90 Z50 S1000 M4
       G5
       G66 R200 C250 F300 S400 E500 ............................ ;3D pocket definition
       M30

N200   G67 B5 C4 I-20 R5 V100 F400 T1D1 M6 ........................... ;Roughing operation
N250   G67 B2 I-20 R5 V100 F550 T2D1 M6 ................................ ;Semi-finishing operation
N300   G68 B1.5 L0.75 Q0 I-20 R5 V80 F275 T3D1 M6 ................. ;Finishing operation


N400   G17 ..................................... ;Beginning of pocket geometry definition
       G0 G90 X50 Y90 Z0 ............ ;A-type contour (Plane profile)
       G1 X0
          Y10
          X100
          Y90
          X50
       G16 YZ ............................... ;Depth profile
       G0 G90 Y90 Z0
       G1 Z-20


       G17 ..................................... ;B-type contour
       G0 G90 X10 Y50 ................. ;Plane profile
       G1 Y100
          X-10
          Y0
          X10
          Y50
       G16 XZ ............................... ;Depth profile
       G0 G90 X10 Z0
       G1 X20 Z-20


       G17 ..................................... ;C-type contour
       G0 G90 X90 Y50 ................. ;Plane profile
       G1 Y100
          X110
          Y0
          X90
          Y50
       G16 XZ ............................... ;Depth profile
       G0 G90 X90 Z0
N500   G2 X70 Z-20 I-20 K0 ........... ;End of pocket geometry definition
```

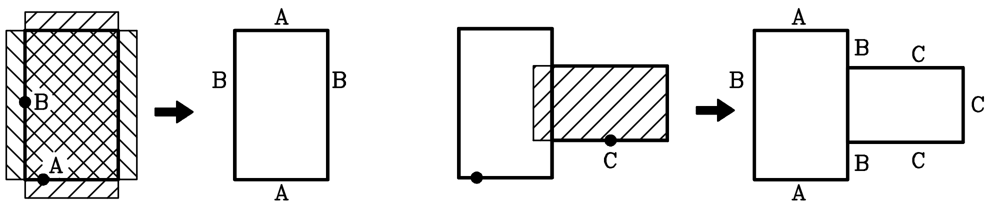
Example 2:





In this example, the island has 3 types of depth profiles: A, B and C.

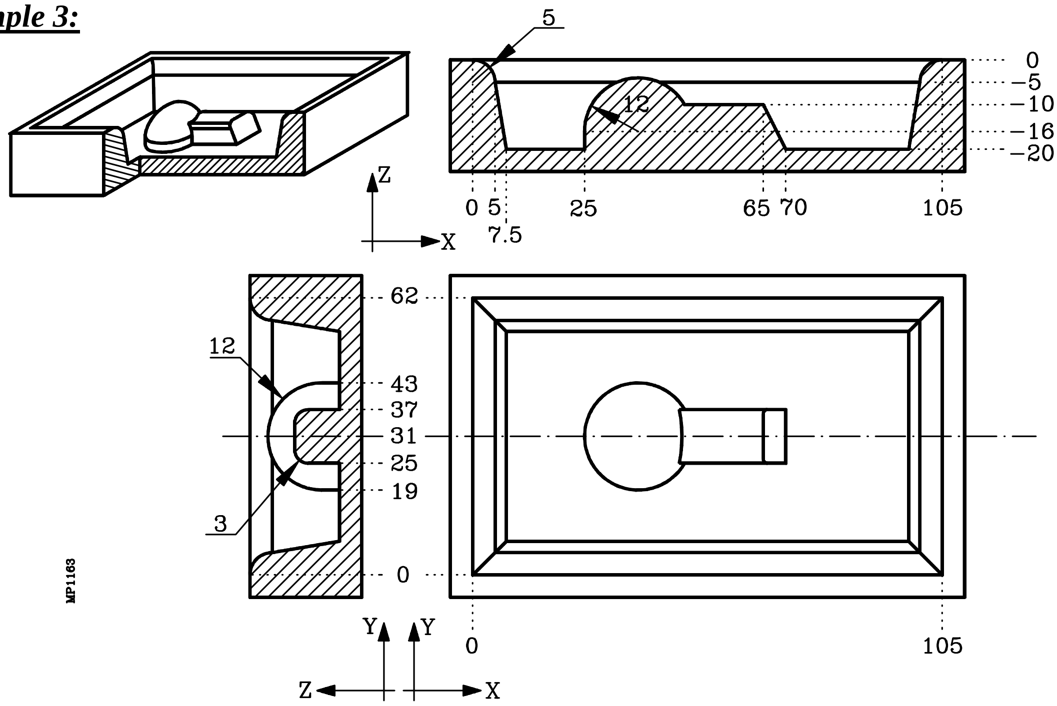
3 contours are used to define the island: A-type contour, B-type contour and C-type contour.



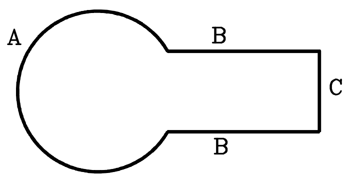

```
(TOR1=7.5,TOI1=0,TOR2=5,TOI2=0,TOR3=2.5,TOI3=0)
G17 G0 G43 G90 Z50 S1000 M4
G5
G66 R200 C250 F300 S400 E500 ............................ ;3D pocket definition
M30

N200   G67 B7 C14 I-25 R3 V100 F500 T1D1 M6 .................... ;Roughing operation
N250   G67 B3 I-25 R3 V100 F625 T2D2 M6 ........................... ;Semi-finishing operation
N300   G68 B1 L1 Q0 J0 I-25 R3 V100 F350 T3D3 M6 ............. ;Finishing operation
```

```
N400  G17 ..................................... ;Beginning of pocket geometry definition
      G0 G90 X0 Y0 Z0 ................ ;Outside contour (plane profile)
      G1 X150
         Y100
         X0
         Y0
      G16 XZ ............................... ;Depth profile
      G0 G90 X0 Z0
      G1 X10 Z-10
         Z-25


      G17 ..................................... ;A-type profile
      G0 G90 X50 Y30 ................. ;Plane profile
      G1 X70
         Y70
         X35
         Y30
         X50
      G16 YZ ............................... ;Depth profile
      G0 G90 Y30 Z-25
      G2 Y50 Z-5 J20 K0


      G17 ..................................... ;B-type profile
      G0 G90 X40 Y50 ................. ;Plane profile
      G1 Y25
         X65
         Y75
         X40
         Y50
      G16 XZ ............................... ;Depth profile
      G0 G90 X40 Z-25
      G1 Z-5


      G17 ..................................... ;C-type profile
      G0 G90 X80 Y40 ................. ;Plane profile
      G1 X96
         Y60
         X60
         Y40
         X80
      G16 YZ ............................... ;Depth profile
      G0 G90 Y40 Z-25
N500  G2 Y50 Z-15 J10 K0............. ;End of pocket geometry definition
```

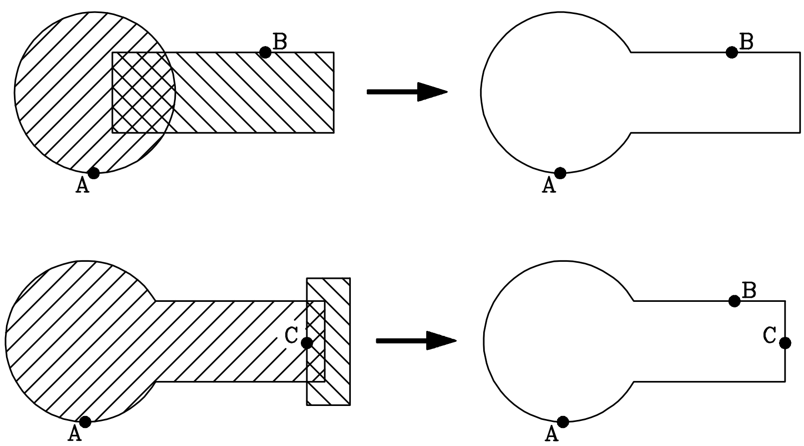
Example 3:



In this example, the island has 3 types of depth profiles: A, B and C.

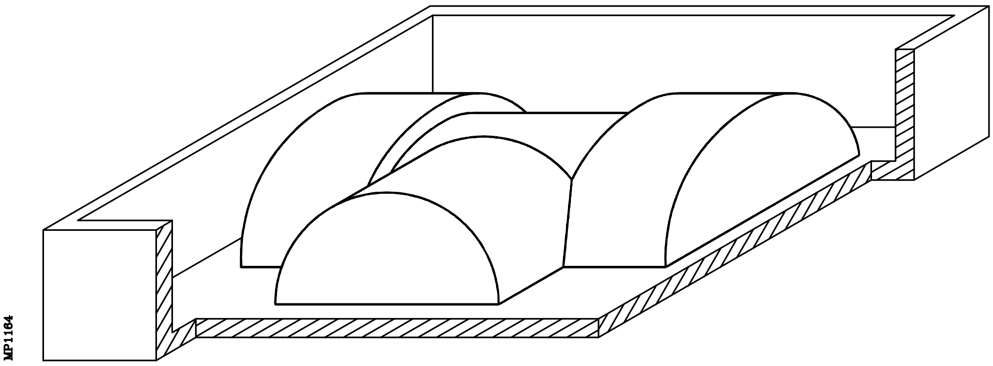


3 contours are used to define the island: A-type contour, B-type contour and C-type contour.

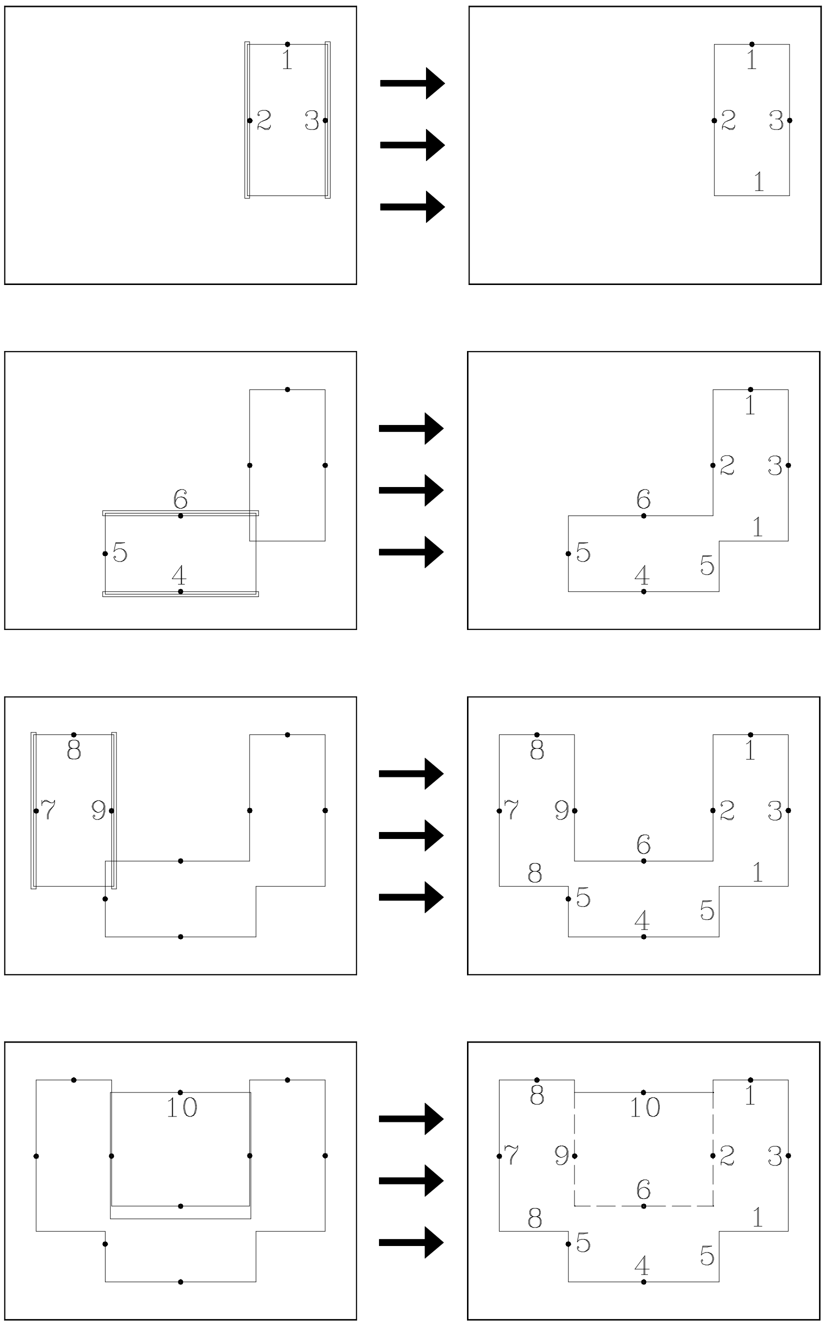


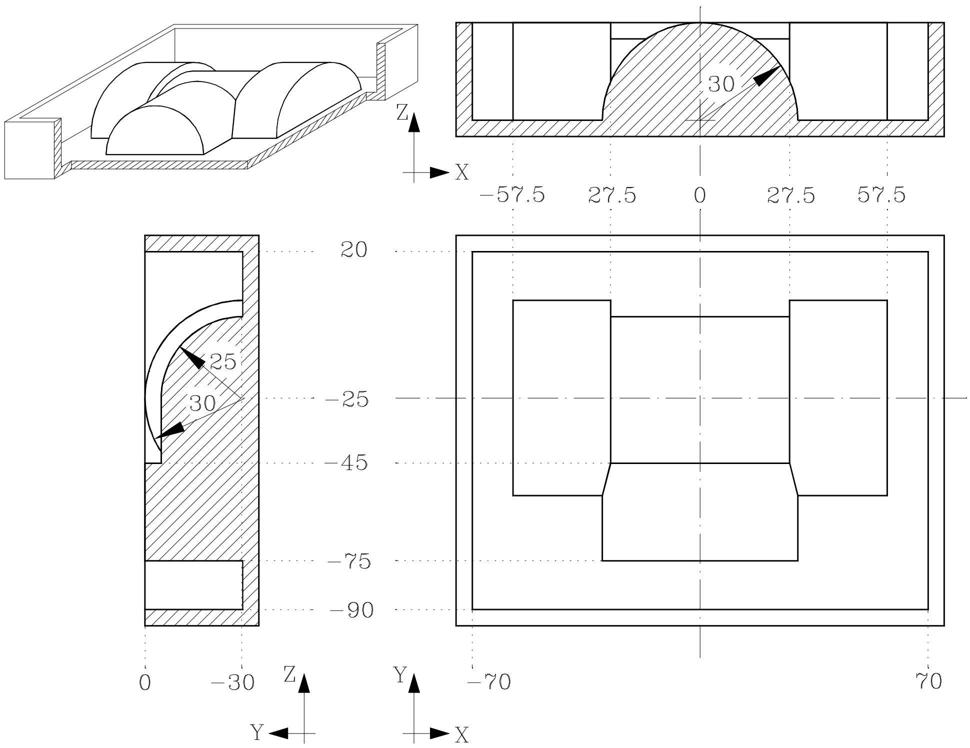
```
        (TOR1=4,TOI1=0,TOR2=2.5,TOI2=0)
        G17 G0 G43 G90 Z25 S1000 M3
        G66 R200 C250 F300 S400 E500 ............................ ;3D pocket definition
        M30

N200    G67 B5 C4 I-20 R5 V100 F700 T1 D1 M6 ........................... ;Roughing operation
N250    G67 B2 I-20 R5 V100 F850 T1 D1 M6 ................................ ;Semi-finishing operation
N300    G68 B1.5 L0.25 Q0 I-20 R5 V100 F500 T2 D2 M6 ............... ;Finishing operation
;

N400    G17 ..................................... ;Beginning of pocket geometry definition
        G0 G90 X0 Y0 Z0 ................ ;Outside contour (plane profile)
        G1  X105
            Y62
            X0
            Y0
        G16 XZ ............................... ;Depth profile
        G0  X0 Z0
        G2 X5 Z-5 I0 K-5
        G1  X7.5 Z-20


        G17 ..................................... ;A-type contour
        G90 G0 X37 Y19 .................. ;Plane profile
        G2  I0 J12
        G16 YZ ............................... ;Depth profile
        G0  Y19 Z-20
        G1  Z-16
        G2  Y31 Z-4 R12 .................. ;End of pocket geometry definition


        G17 ..................................... ;B-type contour
        G90 G0 X60 Y37 .................. ;Plane profile
        G1  X75
            Y25
            X40
            Y37
            X60
        G16 YZ ............................... ;Depth profile
        G0  Y37 Z-20
        G1  Z-13
        G3  Y34 Z-10 J-3 K0


        G17 ..................................... ;C-type contour
        G0  X70 Y31 ........................ ;Plane profile
        G1  Y40
            X80
            Y20
            X70
            Y31
        G16 XZ ............................... ;Depth profile
        G0  X70 Z-20
N500    G1  X65 Z-10 ....................... ;End of pocket geometry definition
```

Example 4:



To define the island 10 contours are used as shown here:

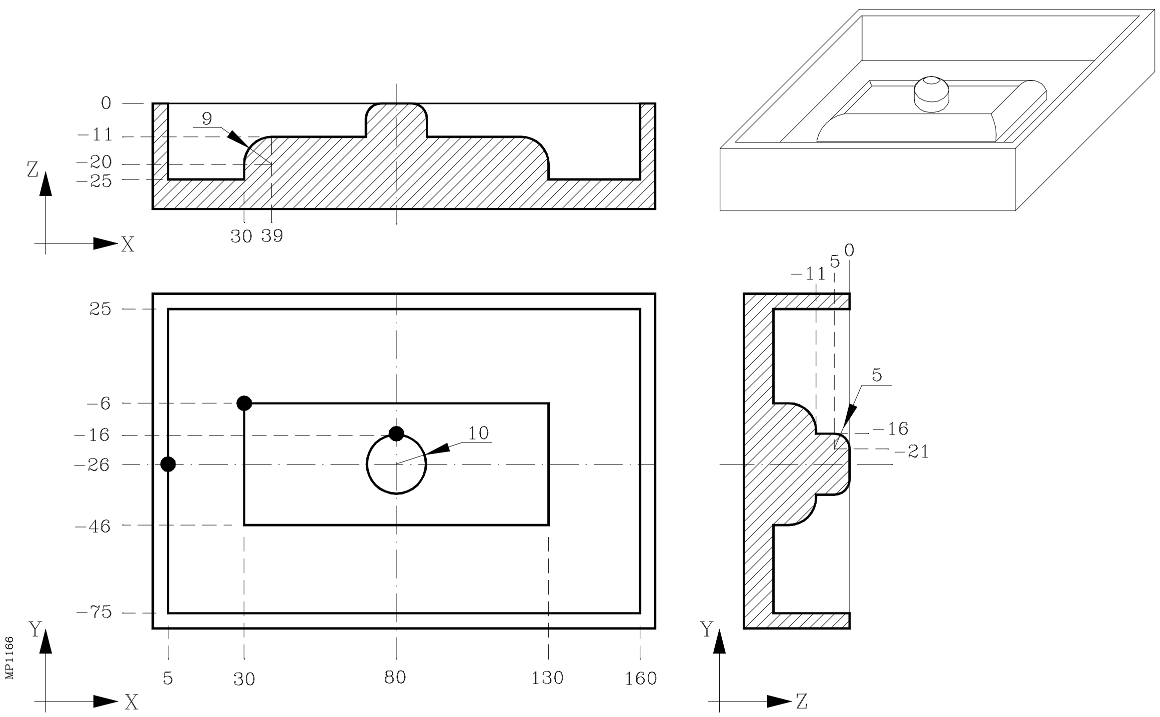




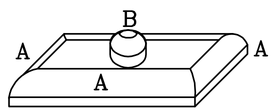

```
      (TOR1=4,TOI1=0,TOR2=2.5,TOI2=0)
      G17 G0 G43 G90 Z25 S1000 M3
      G66 R200 C250 F300 S400 E500 ................................... ;Definition of the 3D pocket
      M30

N200 G67 B5 C0 I-30 R5 V100 F700 T1D1 M6 ......................... ;Roughing Operation
N250 G67 B1.15 I-29 R5 V100 F850 T1D1 M6 ......................... ;Semi-finishing Operation
N300 G68 B1.5 L0.25 Q0 I-30 R5 V100 F500 T2D2 M6 ............ ;Finishing Operation

N400 G17 ............................................................................... ;Beginning of the pocket geometry definition
      G90 G0 X-70 Y20 Z0 .................................................... ;Outside contour (plane profile)
      G1    X70
            Y-90
            X-70
            Y20

      G17   ............................................................................. ;Contour number 1
      G90   G0 X42.5 Y5 ........................................................ ;Plane profile
      G1 G91 X-16
            Y-60
            X32
            Y60
            X-16
      G16YZ ........................................................................... ;Depth profile
      G0 G90 Y5 Z-30
      G3 Y-25 Z0 J-30 K0

      G17   ............................................................................. ;Contour number 2
      G0 X27.5 Y-25
      G1 G91 Y31
      G1    X-2
            Y-62
            X2
            Y31
      G16XZ ........................................................................... ;Depth profile
      G0 G90 X27.5 Z-30
      G1 Z0
```

```
G17 .................. ;Contour number 3
G0 X57.5 Y-25
G1 G91 Y-31
       X2
       Y62
       X-2
       Y-31
G16XZ ............... ;Depth profile
G0 G90 X57.5 Z-30
G1 Z0

G17 .................. ;Contour number 4
G0 X0 Y-75
G1 G91 X-31
       Y-2
       X62
       Y2
       X-31
G16YZ ............... ;Depth profile
G0 G90 Y-75 Z-30
G1 Z0

G17 .................. ;Contour number 5
G0 X-30 Y-60
G1 G91 Y-16
       X60
       Y32
       X-60
       Y-16
G16XZ ............... ;Depth profile
G0 G90 X-30 Z-30
G2 X0 Z0 I30 K0

G17 .................. ;Contour number 6
G0 X0 Y-45
G1 G91 X31
       Y2
       X-62
       Y-2
       X31
G16YZ ............... ;Depth profile
G0 G90 Y-45 Z-30
G1 Z0

G17 ...................;Contour number 7
G0 X-57.5 Y-25
G1 G91 Y31
       X-2
       Y-62
       X2
       Y31
G16XZ ................;Depth profile
G0 G90 X-57.5 Z-30
G1 Z0

G17 ...................;Contour number 8
G0 X-42.5 Y5
G1 G91 X-16
       Y-60
       X32
       Y60
       X-16
G16YZ
G0 G90 Y5 Z-30
G3 Y-25 Z0 J-30 K0

G17 ...................;Contour number 9
G0 X-27.5 Y-25
G1 G91 Y-31
       X2
       Y62
       X-2
       Y-31
G16XZ ................;Depth profile
G0 G90 X27.5 Z-30
G1 Z0

G17 ...................;Contour number 10
G0 X0 Y0
G1     X-28
       Y-50
       X28
       Y0
       X0
G16YZ ................;Depth profile
G0 Y0 Z-30
N500 G3 Y-25 Z-5 J-25 K0
```

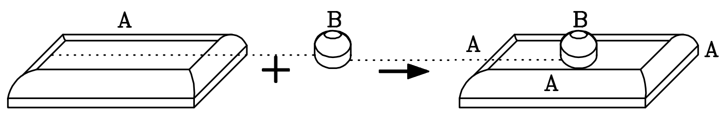
Example 5:



In this example, the island has 2 types of depth profiles: A and B.



2 contours are used to define the island: the low contour (A-type) and the high contour (B-type).




```
       (TOR1=2.5,TOL1=20,TOI1=0,TOK1=0)
       G17 G0 G43 G90 Z50 S1000 M4
       G5
       G66 R200 C250 F300 S400 E500 ............................. ;3D pocket definition
       M30

N200   G67 B5 C4 I-25 R5 V100 F400 T1D1 M6 ........................... ;Roughing operation
N250   G67 B2 I-25 R5 V100 F550 T2D1 M6 ................................ ;Semi-finishing operation
N300   G68 B1.5 L0.75 Q0 I-25 R5 V100 F275 T3D1 M6 ............... ;Finishing operation
```

```
N400  G17 ............................................................ ;Beginning of pocket geometry definition
      G90 G0 X5 Y-26 Z0 ................................... ;Outside contour (plane profile)
      G1    Y25
            X160
            Y-75
            X5
            Y-26

      G17   ........................................................ ;Low contour (A type)
      G90 G0 X30 Y-6 ........................................ ;Plane profile
      G1    Y-46
            X130
            Y-6
            X30
      G16 XZ ...................................................... ;Depth profile
      G0    X30 Z-25
      G1    Z-20
      G2    X39 Z-11 I9 K0

      G17   ........................................................ ;High contour (B-type)
      G90 G0 X80 Y-16 ....................................... ;Plane profile
      G2    I0 J-10
      G16   YZ ..................................................... ;Depth profile
      G0    Y-16 Z-11
      G1    Y-16 Z-5
N500  G3    Y-21 Z0 J-5 K0 .................................. ;End of pocket geometry definition
```

11.2.10 ERRORS

The CNC will issue the following errors:

ERROR 1025 : A tool of no radius has been programmed.
When using a tool with "0" radius while machining a pocket.

ERROR 1026 : A step greater than the tool diameter has been programmed.
When parameter "C" of the roughing operation is greater than the diameter of the roughing tool.

ERROR 1041 : A mandatory parameter not programmed in the canned cycle.
It comes up in the following instances:
- When parameters "I" and "R" have not been programmed in the roughing operation.
- When not using a roughing operation and not programming the "I" and "R" parameters for the semi-finishing operation.
- When not using a semi-finishing operation and not programming the "I" and "R" parameters for the finishing operation.
- When parameter "B" has not been programmed in the finishing operation.

ERROR 1042 : Wrong canned cycle parameter value.
It comes up in the following instances:
- When parameter "Q" of the finishing operation has the wrong value.
- When parameter "B" of the finishing operation has a "0" value.
- When parameter "J" of the finishing operation has been programmed with a value greater than the finishing tool radius.

ERROR 1043 : Wrong depth profile in an irregular pocket with islands
It comes up in the following instances:
- When the depth profiles of 2 sections of the same contour (simple or composite) cross each other
- When the finishing operation cannot be performed with the programmed tool.
  A typical case is a spherical mold with a non-spherical tool (parameter "J" not equal to the radius).

ERROR 1044 : The plane profile intersects itself in an irregular pocket with islands.
It comes up when any of the plane profiles of the programmed contours intersects itself.

ERROR 1046 : Wrong tool position prior to the canned cycle.
It comes up when calling the G66 cycle if the tool is positioned between the reference plane and the depth coordinate (bottom) of any of the operations.

ERROR 1047 : Open plane profile in an irregular pocket with islands.
It comes up when any of the programmed contours does not begin and end at the same point. It may be because G1 has not been programmed after the beginning, with G0, on any of the profiles.

ERROR 1048 : The part surface coordinate (top) has not been programmed in an irregular pocket with islands.
It comes up when the first point of the geometry does not include the pocket top coordinate.

ERROR 1049 : Wrong reference plane coordinate for the canned cycle.

It comes up when the coordinate of the reference plane is located between the part's "top" and "bottom" in any of the operations.

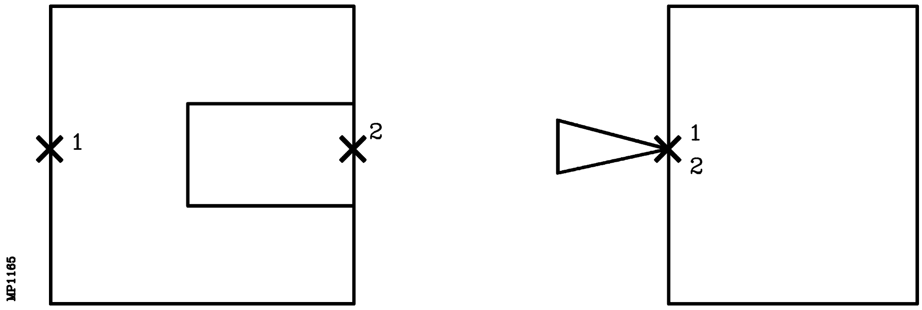
ERROR 1084 : Wrong circular path.

It comes up when any of the paths programmed in the geometry definition of the pocket is wrong.

ERROR 1227 : Wrong profile intersection in an irregular pocket with islands.

It comes up in the following instances:

- When two plane profiles have a common section (drawing on the left).
- When the initial points of two profiles in the main plane coincide (drawing on the right).



12. *WORKING WITH A PROBE*

The CNC has two probe inputs, one for TTL-type 5V DC signals and another for 24 V DC signals.

The connection of the different types of probes to these inputs are explained in the appendix to the Installation and Start-up manual.

This control allows the following operations to be performed, by using probes:

* Programming probing blocks with functions G75 and G76.

* Several tool calibration and part-measurement cycles by means of high-level language programming.

12.1 PROBING (G75,G76)

The **G75** function allows movements to be programmed which will end after the CNC **receives** the signal from the measuring probe used.

The **G76** function allows movements to be programmed which will end after the CNC **no longer receives** the signal from the measuring probe used.

Their definition format is:

G75 X..C # 5.5
G76 X..C # 5.5

After **G75** or **G76**, the required axis or axes will be programmed, as well as the coordinates of these axes which will define the end point of the programmed movement.

The machine will move according to the programmed path until it receives the signal from the probe (G75) or until it no longer receives the probe signal (G76). At this time, the CNC will consider the block finished, taking as the theoretical position of the axes the real position which they have at that time.

If the axes reach the programmed position before receiving (G75) or while receiving (G76) the external signal from the probe, the CNC will stop the movement of the axes.

This type of movement with probing blocks are very useful when it is required to generate measurement or verification programs for tools and parts.

Functions G75 and G76 are not modal and, therefore, must be programmed whenever it is wished to probe.

It is not possible to vary the Feedrate Override while either G75 or G76 is active. It stays set at 100 %.

Functions G75 and G76 are incompatible with each other and with G00, G02, G03, G33, G34, G41 and G42 functions. In addition, once this has been performed, the CNC will assume functions G01 and G40.

12.2 PROBING CANNED CYCLES

The CNC has the following probing canned cycles:

1 Tool length calibration canned cycle.

2 Probe calibration canned cycle

3 Surface measuring canned cycle

4 Outside corner measuring canned cycle

5 Inside corner measuring canned cycle

6 Angle measuring canned cycle

7 Corner and angle measuring canned cycle

8 Hole measuring canned cycle

9 Boss measuring canned cycle

All the movements of these probing canned cycles will be performed in the X, Y, Z axes and the work plane must be formed by 2 of these axes (XY, XZ, YZ, YX, ZX, ZY). The other axis, which must be perpendicular to this plane, must be selected as the longitudinal axis.

Canned cycles will be programmed by means of the high level mnemonic, **PROBE**, which has the following programming format:

(PROBE(expression),(assignment statement),...)

This statement calls the probing cycle indicated by means of a number or any expression which results in a number. Besides, it allows the parameters of this cycle to be initialized with the values required to perform it, by means of assignment statements.

General considerations

Probing canned cycles are not modal, and therefore must be programmed whenever it is required to perform any of them.

The probes used in the performance of these cycles are:

* Probe placed on a fixed position on the machine, used for calibrating tools.

* Probe placed in the spindle, will be treated as a tool and will be used in the different measuring cycles.

The execution of a probing canned cycle does not alter the history of previous “G” functions, except for the radius compensation functions G41 and G42.

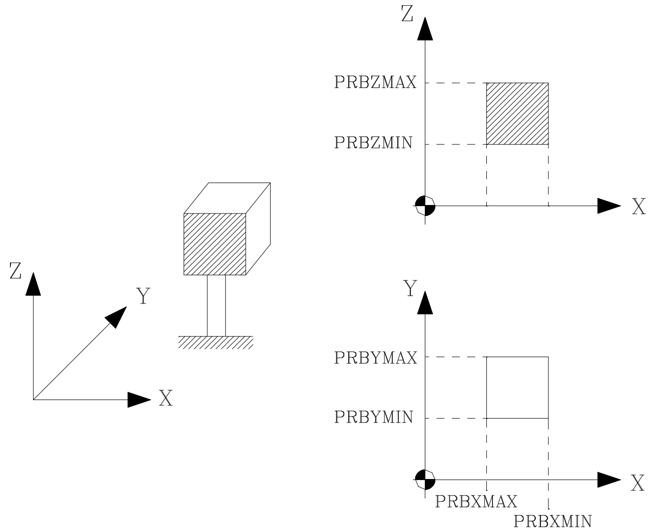
12.3 *TOOL LENGTH CALIBRATION CANNED CYCLE*

This is used to calibrate the length of the selected tool. Once the cycle has ended, the value (L) corresponding to the tool offset which is selected will be updated on the tool offset table.

To perform this cycle it is necessary to have a table-top probe, installed in a fixed position on the machine and with its faces parallel to axes X, Y, Z.

Its position will be indicated in absolute coordinates with respect to machine zero by means of the general machine parameters:

PRBXMIN	Indicates the minimum coordinate occupied by the probe along the X axis.
PRBXMAX	Indicates the maximum coordinate occupied by the probe along the X axis.
PRBYMIN	Indicates the minimum coordinate occupied by the probe along the Y axis.
PRBYMAX	Indicates the maximum coordinate occupied by the probe along the Y axis.
PRBZMIN	Indicates the minimum coordinate occupied by the probe along the Z axis.
PRBZMAX	Indicates the maximum coordinate occupied by the probe along the Z axis.



If it is the first time that the tool length has been calibrated, it is advisable to include an approximate value of its length (L) in the tool offset table.

The programming format for this cycle is as follows:

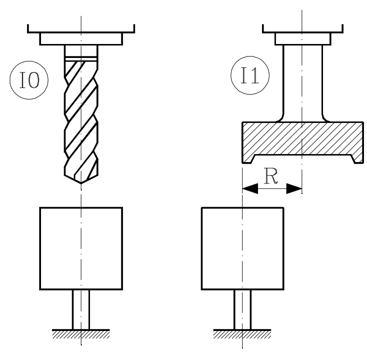
(PROBE 1, B, I, F, X, U, Y, V, Z, W)

B5.5 Defines the safety distance. It must be programmed with a positive value greater than 0.

I Indicates how the calibration canned cycle will be executed.

0= Tool calibration on its center.
1= Tool calibration on its end.

If this is not programmed, the cycle will take the IO value.



F5.5 Defines probing feedrate in mm/min or inch/min.

X, U, Y, V Z, W are optional parameters which usually need not be defined.

On some machines, due to the lack of probe positioning repeatability, the probe must be calibrated before each calibration.

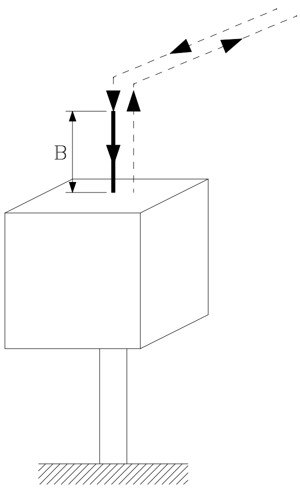
Instead of redefining machine parameters PRBXMIN, PRBXMAX, PRBYMIN, PRBYMAX, PRBZMAX, and PRBZMIN every time the probe is calibrated, these coordinates may be indicated in the X, U, Y, V, Z and W variables respectively.

The CNC does not change the machine parameters but considers the coordinates indicated in X, U, Y, V, Z, W only during this calibration.

If any of the X, U, Y, V, Z, W fields is left out, the CNC assumes the value assigned to the corresponding machine parameter.

Basic operation:

1.- Approach



Movement of the probe in rapid (G00) from the point where the cycle is called to the approach point.

This point is to be found opposite the point where it is wished to measure, at a safety distance (B) from it and along the longitudinal axis.

The approaching movement is made in two stages:

If it is above the safety plane, it first moves in XY and then in Z.

If it is below the safety plane, it first moves in Z to the safety plane and then in XY.

2.- Probing

Movement of the probe along the longitudinal axis at the indicated feedrate (F), until the probe signal is received.

The maximum distance to be travelled in the probing movement is 2B. If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

3.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the point where the cycle was called.

The withdrawal movement is made in two stages:

1st Movement along the longitudinal axis to the coordinate of the point (along this axis) from where the cycle was called.

2nd Movement in the main work plane to the point where the cycle is called.

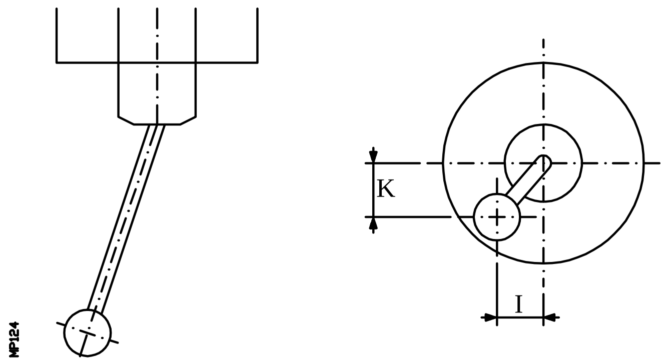
Once the cycle has been completed, the CNC will have updated the tool offset selected at the time on the tool offset table, value (L) and initialized the value of (K) to 0, it also returns the value of the global arithmetic parameter:

P299 Error detected. Difference between the measured tool length and the one assigned to it in the table.

12.4 PROBE CALIBRATING CANNED CYCLE

This is used to calibrate the probe situated in the spindle. This probe which previously must be calibrated in length, will be the one used in probe measuring canned cycles.

The cycle measures the deviation which the probe ball axis has with respect to the tool holder axis, using a previously machined hole with known center and dimensions for its calibration.



The CNC will treat each measuring probe used as just one more tool. The tool offset table fields corresponding to each probe will have the following meaning:

R Radius of the sphere (ball) of the probe. This value will be loaded into the table manually.

L Length of the probe. This value will be indicated by the tool length calibration cycle.

I Deviation of the probe ball with respect to the tool-holder axis, along the abscissa axis. This value will be indicated by the cycle.

K Deviation of the probe ball with respect to the tool holder axis, along the ordinate axis. This value will be indicated by the cycle.

The following steps will be followed for its calibration:

1.- Once the characteristics of the probe have been consulted, the value for the sphere radius (R) will be entered manually in the corresponding tool offset.

2.- After selecting the corresponding tool number and tool offset the Tool Length Calibration Cycle will be performed, the value of (L) will be updated and the value of (K) will be initialized to 0.

3.- Execution of the probe calibration canned cycle, updating the “I” and “K” values.

The programming format for this cycle is:

(PROBE 2,X,Y,Z,B,J,E,H,F)

X+/-5.5 Real coordinate, along the X axis, of the hole center.

Y+/-5.5 Real coordinate, along the Y axis, of the hole center.

Z+/-5.5 Real coordinate, along the Z axis, of the hole center.

B5.5 Defines the safety distance. Must be programmed with a positive value and over 0.

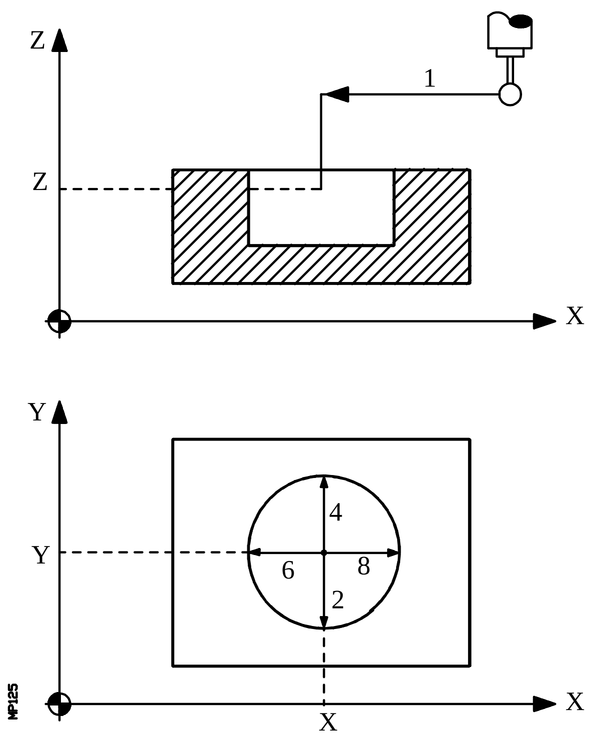
J5.5 Defines the real diameter of the hole. Must be programmed with a positive value and over 0.

E.5.5 Defines the distance which the probe moves back after initial probing. Must be programmed with a positive value and over 0.

H5.5 Defines the feedrate for the initial probing movement. Must be programmed in mm/minute or in inches/minute.

F5.5 Defines the probing feedrate. Must be programmed in mm/minute or in inches/minute.

Basic operation:



1.- Approach

Movement of the probe in rapid (G00) from the point where the cycle is called to the center of the hole.

The approaching movement is made in two stages:

1st Movement in the main work plane.

2nd Movement along the longitudinal axis.

2.- Probing

This movement consists of:

* Movement of the probe along the ordinate axis at the indicated feedrate (H), until the probe signal is received.

  The maximum distance to be travelled in the probing movement is "B+(J/2)". If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

* Return of the probe in rapid (G00) the distance indicated in (E).

* Movement of the probe along the ordinate axis at the indicated feedrate (F), until the probe signal is received.

3.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the real center of the hole.

4.- Second probing movement.

Same as above.

5.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the real center of the hole along the ordinate axis.

6.- Third probing movement.

Same as above.

7.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the real center of the hole.

8.- Fourth probing movement.

Same as above.

9.- Withdrawal

This movement consists of:

* Movement of the probe in rapid (G00) from the point where it probed to the real center of the hole.

* Movement along the longitudinal axis to the coordinate of the point (along this axis) from where the cycle was called.

* Movement in the main work plane to the point where the cycle was called.

Once the cycle has been completed, the CNC will have updated the "I" and "K" values corresponding to the tool offset selected at the time on the tool offset table.

On the other hand, arithmetic parameter P299 returns the best value to be assigned to general machine parameter PRODEL.

12.5 SURFACE MEASURING CANNED CYCLE

A probe placed in the spindle will be used, which must be previously calibrated by means of canned cycles:

Canned cycle for calibrating tool length.
Canned cycle for calibrating probe.

This cycle allows correcting the value of the tool offset of the tool which has been used in the surface machining process. This correction will be used only when the measurement error exceeds a programmed value.

The programming format for this cycle is:

(PROBE 3,X,Y,Z,B,K,F,C,D,L)

X+/-5.5 Theoretical coordinate, along the X axis, of the point over which it is required to measure.

Y+/-5.5 Theoretical coordinate, along the Y axis, of the point over which it is required to measure.

Z+/-5.5 Theoretical coordinate, along the Z axis, of the point over which it is required to measure.

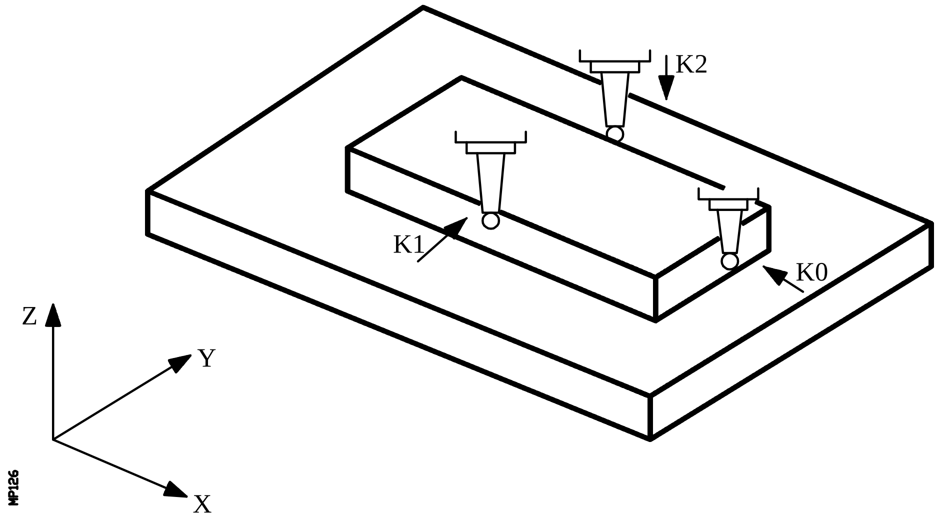
B5.5 Defines the safety distance. Must be programmed with a positive value and over 0.

The probe must be placed, with respect to the point to be measured, at a distance greater than this value when the cycle is called.

K Defines the axis with which it is required to measure the surface and will be defined by means of the following code:

0= With the abscissa axis of the work plane.
1= With the ordinate axis of the work plane.
2= With the longitudinal axis of the work plane.

If this is not programmed, the canned cycle will take the value of K0.



F5.5 Defines the probing feedrate in mm/min. or inches/min.

C Indicates where the probing cycle must finish.

0= Will return to the same point where the call to the cycle was made.
1= The cycle will finish over the measured point returning the longitudinal axis to the cycle calling point.

If this is not programmed, the canned cycle will take the value of C0.

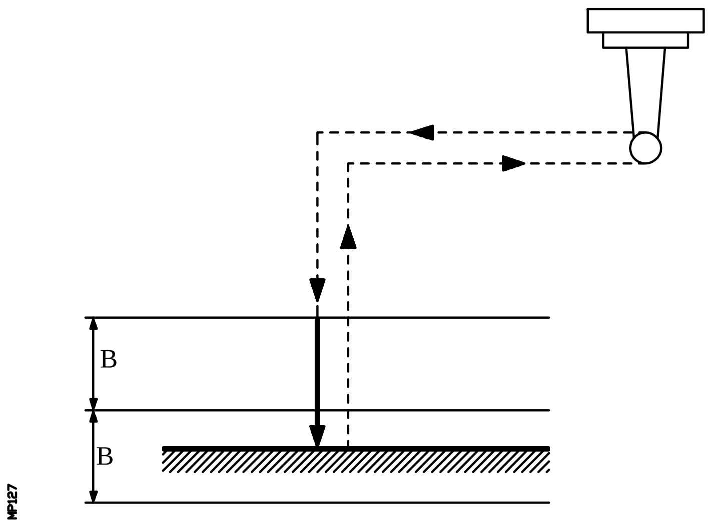
D4 Defines the number of the tool offset to be corrected, once the measurement cycle is completed.

If this is not programmed or is programmed with a value of 0, the CNC will understand that it is not required to make this correction.

L5.5 Defines the tolerance which will be applied to the error measured. It will be programmed with an absolute value and the tool offset will be corrected only when the error exceeds this value.

If this is not programmed, the canned cycle will take the value of 0.

Basic operation:



1.- Approach

Movement of the probe in rapid (G00) from the point where the cycle is called to the approach point.

This point is to be found opposite the point where it is wished to measure, at a safety distance (B) from this and along the probing axis (K).

The approaching movement is made in two stages:

1st Movement in the main work plane.
2nd Movement along the longitudinal axis.

2.- Probing

Movement of the probe along the selected axis (K) at the indicated feedrate (F), until the probe signal is received.

The maximum distance to be travelled in the probing movement is 2B. If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

Once probing has been made, the CNC will assume as their theoretical position the real position of the axes when the probe signal is received .

3. Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the point where the cycle was called.

The withdrawal movement is made in three stages:

1st Movement along the probing axis to the approach point.

2nd Movement along the longitudinal axis to the coordinate of the point (along this axis) from where the cycle was called.

3rd When (C0) is programmed, movement is made in the main work plane to the point where the cycle is called.

Once the cycle has been completed, the CNC will return the real values obtained after measurement, in the following global arithmetic parameters.

P298 **Real surface coordinate.**

P299 **Error detected.** Difference between the real coordinate of the surface and the theoretical programmed coordinate.

If the Tool Offset Number (D) was selected, the CNC will modify the values of this tool offset, whenever the measurement error is equal to or greater than the tolerance (L).

Depending on the axis the measurement is made with (K), the correction will be made on the length or radius value.

* If the measurement is made with the axis longitudinal to the work plane, the length wear (K) of the indicated tool offset (D) will be modified.

* If the measurement is made with one of the axes which make up the work plane, the radius wear (I) of the indicated tool offset (D) will be modified.

12.6 OUTSIDE CORNER MEASURING CANNED CYCLE

A probe placed in the spindle will be used, which must be previously calibrated by means of canned cycles:

Canned cycle for calibrating tool length.
Canned cycle for calibrating probe.

The programming format for this cycle is:

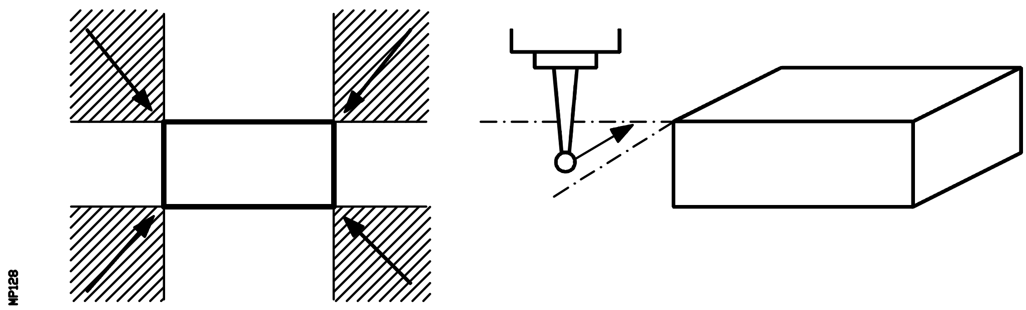
(PROBE 4,X,Y,Z,B,F)

X+/-5.5 Theoretical coordinate, along the X axis, of the corner to be measured.

Y+/-5.5 Theoretical coordinate, along the Y axis, of the corner to be measured.

Z+/-5.5 Theoretical coordinate, along the Z axis, of the corner to be measured.

Depending on the corner of the part it is required to measure, the probe must be placed in the corresponding shaded area (see figure) before calling the cycle.

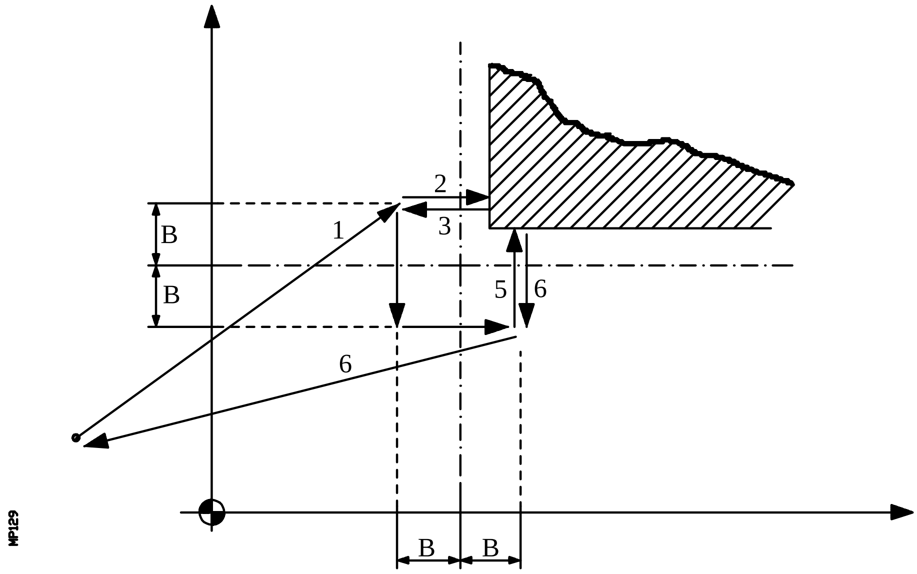


B5.5 Defines the safety distance. Must be programmed with a positive value and over 0.

The probe must be placed, with respect to the point to be measured, at a distance greater than this value when the cycle is called.

F5.5 Defines the probing feedrate in mm/min or inch/min.

Basic operation:



1.- Approach

Movement of the probe in rapid (G00) from the point where the cycle is called to the first approach point, situated at a distance (B) from the first face to be probed.

The approaching movement is made in two stages:

1st Movement in the main work plane.
2nd Movement along the longitudinal axis.

2.- Probing

Movement of the probe along the abscissa axis at the indicated feedrate (F), until the probe signal is received.

The maximum distance to be travelled in the probing movement is 2B. If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

3.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the first approach point

4.- Second approach

Movement of the probe in rapid (G00) from the first approach point to the second.

The approaching movement is made in two stages:

1st Movement along the ordinate plane.
2nd Movement along the abscissa axis.

5.- Second probing

Movement of the probe along the abscissa axis at the indicated feedrate (F), until the probe signal is received.

The maximum distance to be travelled in the probing movement is 2B. If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

6.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed for the second time to the point where the cycle was called.

The withdrawal movement is made in three stages:

1st Movement along the probing axis to the second approach point.

2nd Movement along the longitudinal axis to the coordinate of the point corresponding to this axis where the cycle is called.

3rd Movement in the main work plane to the point where the cycle is called.

Once the cycle has been completed, the CNC will return the real values obtained after measurement, in the following global arithmetic parameters.

P296 **Real coordinate of the corner along the abscissa axis.**

P297 **Real coordinate of the corner along the ordinate axis.**

P298 **Error detected along the abscissa axis.** Difference between the real coordinate of the corner and the theoretical programmed coordinate.

P299 **Error detected along the ordinate axis.** Difference between the real coordinate of the corner and the theoretical programmed coordinate.

12.7 *INSIDE CORNER MEASURING CANNED CYCLE*

A probe placed in the spindle will be used, which must be previously calibrated by means of canned cycles:

Canned cycle for calibrating tool length.
Canned cycle for calibrating probe.

The programming format for this cycle is:

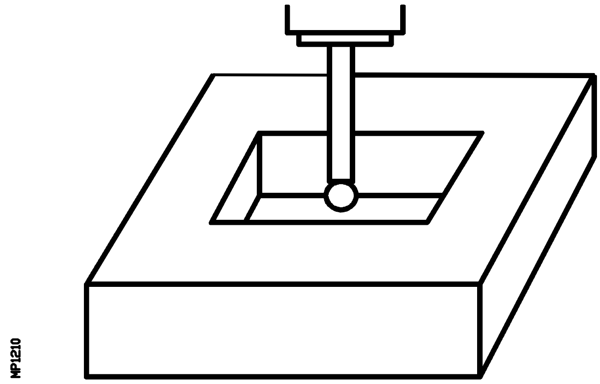
(PROBE 5,X,Y,Z,B,F)

X+/-5.5 Theoretical coordinate, along the X axis, of the corner to be measured.

Y+/-5.5 Theoretical coordinate, along the Y axis, of the corner to be measured.

Z+/-5.5 Theoretical coordinate, along the Z axis, of the corner to be measured.

The probe must be placed within the pocket before calling the cycle.

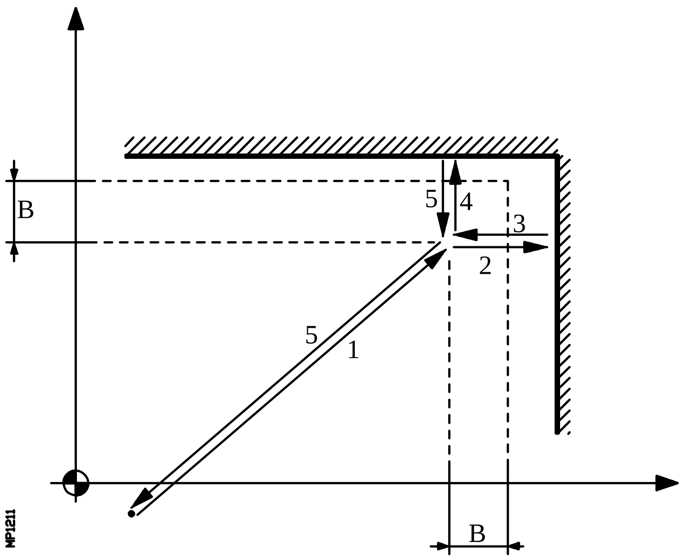


B5.5 Defines the safety distance. Must be programmed with a positive value and over 0.

The probe must be placed, with respect to the point to be measured, at a distance greater than this value when the cycle is called.

F5.5 Defines the probing feedrate in mm/min. or inch/min.

Basic operation:



1.- Approach

Movement of the probe in rapid (G00) from the point where the cycle is called to the first approach point, situated at a distance (B) from both faces to be probed.

The approaching movement is made in two stages:

1st Movement in the main work plane.
2nd Movement along the longitudinal axis.

2.- Probing

Movement of the probe along the abscissa axis at the indicated feedrate (F), until the probe signal is received.

The maximum distance to be travelled in the probing movement is 2B. If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

3.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the approach point

4.- Second probing

Movement of the probe along the abscissa axis at the indicated feedrate (F), until the probe signal is received.

The maximum distance to be travelled in the probing movement is 2B. If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

5.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed for the second time to the point where the cycle was called.

The withdrawal movement is made in three stages:

1st Movement along the probing axis to the approach point.

2nd Movement along the longitudinal axis to the coordinate of the point corresponding to this axis where the cycle is called.

3rd Movement in the main work plane to the point where the cycle is called.

Once the cycle has been completed, the CNC will return the real values obtained after measurement, in the following global arithmetic parameters.

P296 Real coordinate of the corner along the abscissa axis.

P297 Real coordinate of the corner along the ordinate axis.

P298 Error detected along the abscissa axis. Difference between the real coordinate of the corner and the theoretical programmed coordinate.

P299 Error detected along the ordinate axis. Difference between the real coordinate of the corner and the theoretical programmed coordinate.

12.8 ANGLE MEASURING CANNED CYCLE

A probe placed in the spindle will be used, which must be previously calibrated by means of canned cycles:

Canned cycle for calibrating tool length.
Canned cycle for calibrating probe.

The programming format for this cycle is:

(PROBE 6,X,Y,Z,B,F)

X+/-5.5 Theoretical coordinate, along the X axis, of the angle to be measured.

Y+/-5.5 Theoretical coordinate, along the Y axis, of the angle to be measured.

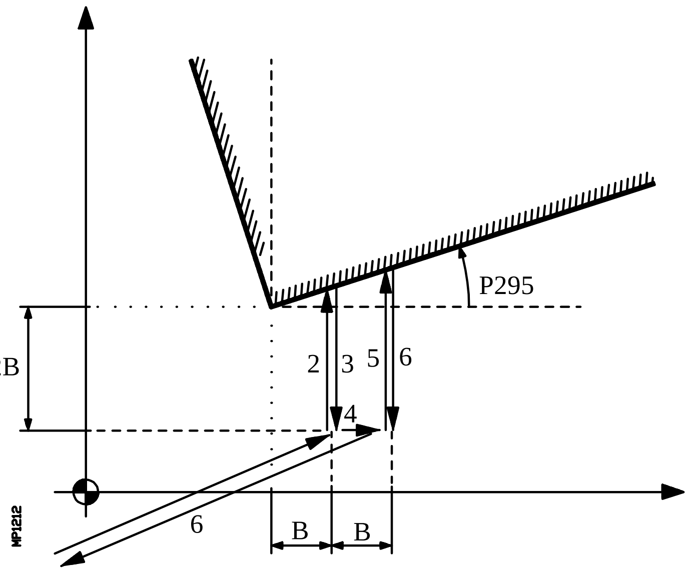
Z+/-5.5 Theoretical coordinate, along the Z axis, of the angle to be measured.

B5.5 Defines the safety distance. Must be programmed with a positive value and over 0.

The probe must be placed, with respect to the point to be measured, at a distance greater than double this value when the cycle is called.

F5.5 Defines the probing feedrate in mm/min. or inch/min.

Basic operation:



1.- Approach

Movement of the probe in rapid (G00) from the point where the cycle is called to the first approach point, situated at a distance (B) from the programmed vertex and at (2B) from the face to be probed.

The approaching movement is made in two stages:

1st Movement in the main work plane.
2nd Movement along the longitudinal axis.

2.- Probing

Movement of the probe along the ordinate axis at the indicated feedrate (F), until the probe signal is received.

The maximum distance to be travelled in the probing movement is 3B. If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

3.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the first approach point

4.- Second approach

Movement of the probe in rapid (G00) from the first approach point to the second. It is at a distance (B) from the first one.

5.- Second probing

Movement of the probe along the abscissa axis at the indicated feedrate (F), until the probe signal is received.

The maximum distance to be travelled in the probing movement is 4B. If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

6.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed for the second time to the point where the cycle was called.

The withdrawal movement is made in three stages:

1st Movement along the probing axis to the second approach point.

2nd Movement along the longitudinal axis to the coordinate of the point (along this axis) from where the cycle was called.

3rd Movement in the main work plane to the point where the cycle is called.

Once the cycle has been completed, the CNC will return the real values obtained after measurement, in the following global arithmetic parameter.

P295 **Inclination angle** which the part has in relation to the abscissa axis.

This cycle allows angles between ±45° to be measured.

If the angle to be measured is $\geq$ 45°, the CNC will display the corresponding error.

If the angle to be measured is $\leq$ -45°, the probe will collide with the part.

12.9 OUTSIDE CORNER AND ANGLE MEASURING CANNED CYCLE

A probe placed in the spindle will be used, which must be previously calibrated by means of canned cycles:

Canned cycle for calibrating tool length.
Canned cycle for calibrating probe.

The programming format for this cycle is:

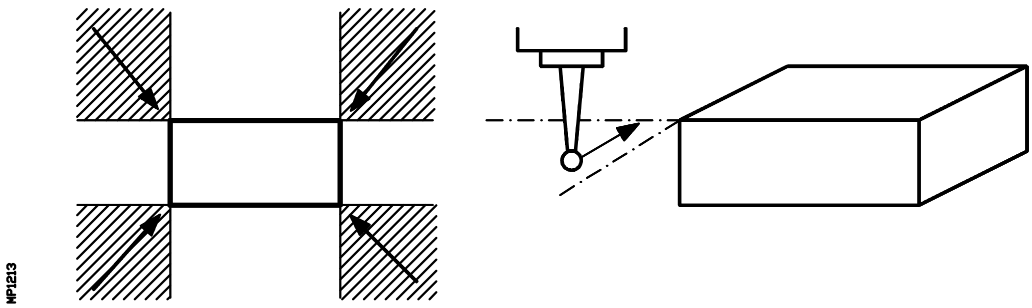
(PROBE 7,X,Y,Z,B,F)

X±5.5 Theoretical coordinate, along the X axis, of the corner to be measured.

Y±5.5 Theoretical coordinate, along the Y axis, of the corner to be measured.

Z±5.5 Theoretical coordinate, along the Z axis, of the corner to be measured.

Depending on the corner of the part it is required to measure, the probe must be placed in the corresponding shaded area (see figure) before calling the cycle.

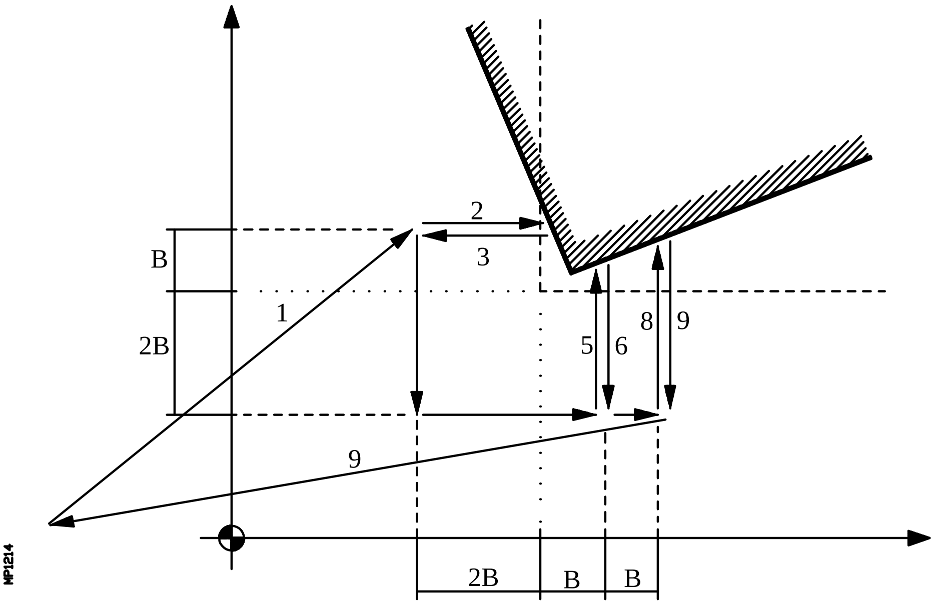


B5.5 Defines the safety distance. Must be programmed with a positive value and over 0.

The probe must be placed, with respect to the point to be measured, at a distance greater than double this value when the cycle is called.

F5.5 Defines the probing feedrate in mm/min. or inch/min.

Basic operation:



1.- Approach

Movement of the probe in rapid (G00) from the point where the cycle is called to the first approach point, situated at a distance (B) from the first face to be probed.

The approaching movement is made in two stages:

1st Movement in the main work plane.
2nd Movement along the longitudinal axis.

2.- Probing

Movement of the probe along the abscissa axis at the indicated feedrate (F), until the probe signal is received.

The maximum distance to be travelled in the probing movement is 3B. If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

3.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the first approach point.

4.- Second approach

Movement of the probe in rapid (G00) from the first approach point to the second, situated at a distance (2B) from the second face to be probed.

The approaching movement is made in two stages:

1st Movement along the ordinate plane.
2nd Movement along the abscissa axis.

5.- Second probing

Movement of the probe along the abscissa axis at the indicated feedrate (F), until the probe signal is received.

The maximum distance to be travelled in the probing movement is 3B. If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

6.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the second point of approach.

7.- Third approach

Movement of the probe in rapid (G00) from the second approach point to the third, situated at a distance (B) from the previous point.

8.- Third probing

Movement of the probe along the ordinate axis at the indicated feedrate (F), until the probe signal is received.

The maximum distance to be travelled in the probing movement is 4B. If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

9.- Withdrawal

Movement of the probe in rapid (G00) from the third probing point to the point where the cycle was called.

The withdrawal movement is made in three stages:

1st Movement along probing axis to the third approach point.

2nd Movement along the longitudinal axis to the coordinate of the point (along this axis) from where the cycle was called.

3rd Movement in the main work plane to the point where the cycle is called.

Once the cycle has been completed, the CNC will return the real values obtained after measurement, in the following global arithmetic parameter.

P295 Inclination angle which the part has in relation to the abscissa axis.

P296 Real coordinate of the corner along the abscissa axis.

P297 Real coordinate of the corner along the ordinate axis.

P298 Error detected along the abscissa axis. Difference between the real coordinate of the corner and the programmed theoretical coordinate.

P299 Error detected along the ordinate axis. Difference between the real coordinate of the corner and the programmed theoretical coordinate.

This cycle allows angles between ±45° to be measured.

If the angle to be measured is ≥ 45° the CNC will display the corresponding error.

If the angle to be measured is ≤ -45°, the probe will collide with the part.

12.10 HOLE MEASURING CANNED CYCLE

A probe placed in the spindle will be used, which must be previously calibrated by means of canned cycles:

Canned cycle for calibrating tool length.
Canned cycle for calibrating probe.

The programming format for this cycle is:

(PROBE 8,X,Y,Z,B,J,E,C,H,F)

X±5.5 Theoretical coordinate, along the X axis, of the center of the hole.

Y±5.5 Theoretical coordinate, along the Y axis, of the center of the hole.

Z±5.5 Theoretical coordinate, along the Z axis, of the center of the hole.

B5.5 Defines the safety distance. Must be programmed with a positive value and over 0.

J5.5 Defines the theoretical diameter of the hole. Must be programmed with a positive value and over 0.

This cycle allows holes to be measured with diameters of no more than (J+B).

E.5.5 Defines the distance which the probe moves back after initial probing. Must be programmed with a positive value and over 0.

C Indicates where the probing cycle must finish.

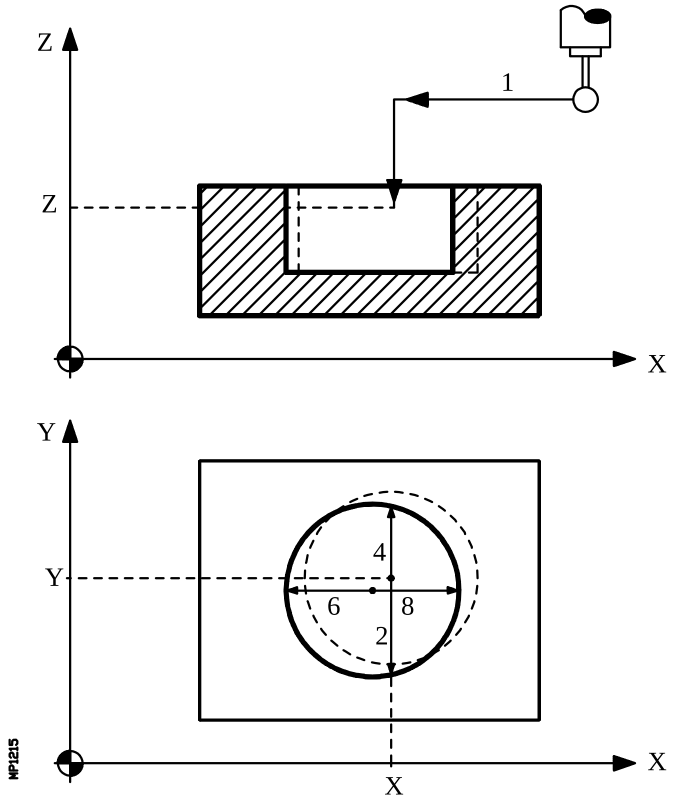
0= Will return to the same point where the call to the cycle was made.
1= The cycle will finish over the measured point returning the longitudinal axis to the cycle calling point.

If this is not programmed, the canned cycle will take the value of C0.

H5.5 Defines the initial probing feedrate in mm/min or in inch/min.

F5.5 Defines the probing feedrate in mm/min or inch/min.

Basic operation:



1.- Approach

Movement of the probe in rapid (G00) from the point where the cycle is called to the center of the hole.

The approaching movement is made in two stages:

1st Movement in the main work plane.
2nd Movement along the longitudinal axis.

2.- Probing

This movement consists of:

* Movement of the probe along the ordinate axis at the indicated feedrate (H), until the probe signal is received.

  The maximum distance to be travelled in the probing movement is "B+(J/2)". If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

* Return of the probe in rapid (G00) the distance indicated in (E).

* Movement of the probe along the ordinate axis at the indicated feedrate (F), until the probe signal is received.

3.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the theoretical center of the hole.

4.- Second probing movement.

Same as above.

5.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the real center (calculated) of the hole along the ordinate axis.

6.- Third probing movement.

Same as above.

7.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the theoretical center of the hole.

8.- Fourth probing movement.

Same as above.

9.- Withdrawal

This movement consists of:

* Movement of the probe in rapid (G00) from the point where it probed to the real center (calculated) of the hole.

* Should (C0) be programmed, the probe will be moved to the point where the cycle was called.

  1st Movement along the longitudinal axis to the coordinate of the point (along this axis) from where the cycle was called.

  2nd Movement on the main work plane to the point where the cycle was called.

Once the cycle has been completed, the CNC will return the real values obtained after measurement, in the following global arithmetic parameter.

P294 **Hole diameter.**

P295 **Hole diameter error.** Difference between the real diameter and programmed diameter.

P296 **Real coordinate of the center along the abscissa axis.**

P297 **Real coordinate of the center along the ordinate axis.**

P298 **Error detected along the abscissa axis.** Difference between the real coordinate of the center and the programmed theoretical coordinate.

P299 **Error detected along the ordinate axis.** Difference between the real coordinate of the center and the programmed theoretical coordinate.

12.11 BOSS MEASURING CANNED CYCLE

A probe placed in the spindle will be used, which must be previously calibrated by means of canned cycles:

Canned cycle for calibrating tool length.
Canned cycle for calibrating probe.

The programming format for this cycle is:

(PROBE 9,X,Y,Z,B,J,E,C,H,F)

X±5.5 Theoretical coordinate, along the X axis, of the center of the boss.

Y±5.5 Theoretical coordinate, along the Y axis, of the center of the boss.

Z±5.5 Theoretical coordinate, along the Z axis, of the center of the boss.

B5.5 Defines the safety distance. Must be programmed with a positive value and over 0.

J5.5 Defines the theoretical diameter of the hole. Must be programmed with a positive value and over 0.

This cycle allows holes to be measured with diameters of no more than (J+B).

E.5.5 Defines the distance which the probe moves back after initial probing. Must be programmed with a positive value and over 0.

C Indicates where the probing cycle must finish.

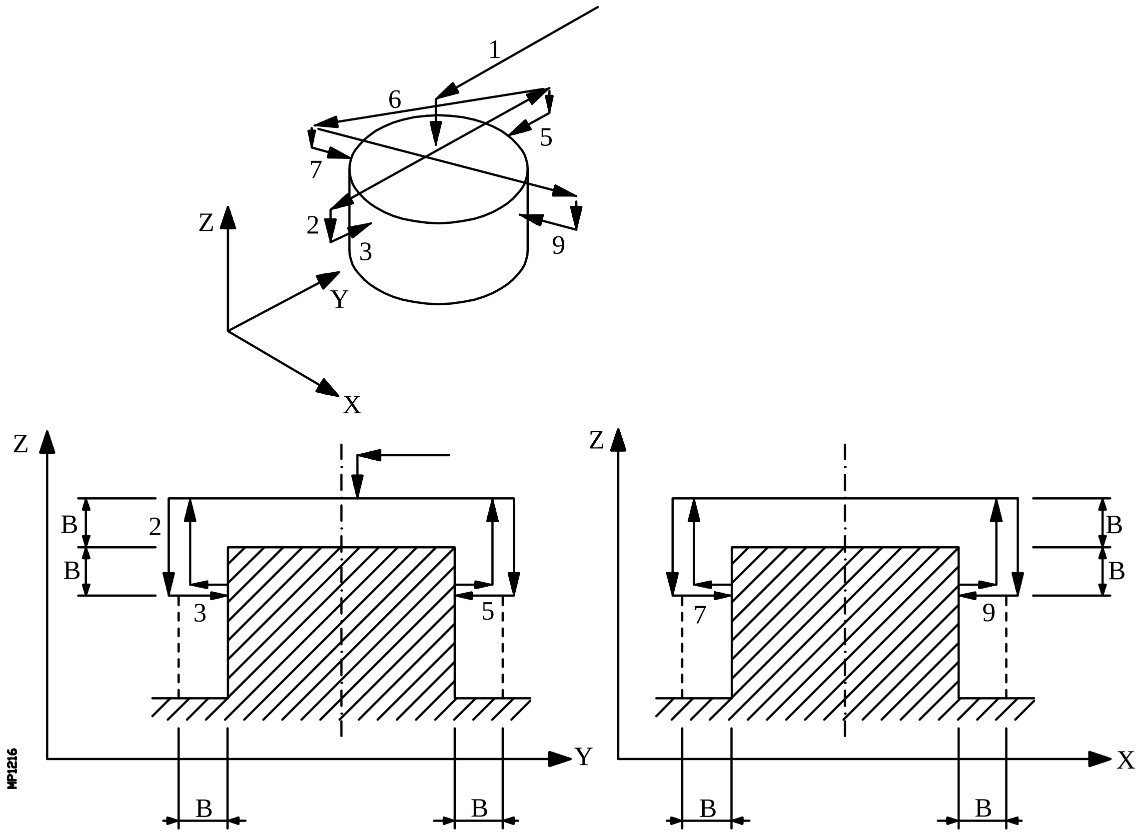
0= Will return to the same point where the call to the cycle was made.
1= The cycle will finish by positioning the probe over the center of the boss, at a distance (B) from the programmed theoretical coordinate.

If this is not programmed, the canned cycle will take the value of C0.

H5.5 Defines the feedrate for the initial probing movement. Must be programmed in mm/minute or in inches/minute.

F5.5 Defines probing feedrate in mm/min. or inch/min.

Basic operation:



1.- Positioning over the center of the boss.

Movement of the probe in rapid (G00) from the point where the cycle is called to the center of the boss.

The approaching movement is made in two stages:

1st Movement in the main work plane.
2nd Movement along the longitudinal axis up to a distance (B) from the programmed surface.

2.- Movement to the first approach point

This movement of the probe which is made in rapid (G00) consists of:

1st Movement along the ordinate axis.
2nd Movement of the longitudinal axis the distance (2B).

3.- Probing

This movement consists of:

* Movement of the probe along the ordinate axis at the indicated feedrate (H), until the probe signal is received.

  The maximum distance to be travelled in the probing movement is "B+(J/2)". If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

* Return of the probe in rapid (G00) the distance indicated in (E).

* Movement of the probe along the ordinate axis at the indicated feedrate (F), until the probe signal is received.

4.- Movement to second approach point

This movement of the probe which is made in rapid (G00) consists of:

* Withdrawal to the first approach point.

* Movement to a distance (B) above the boss, to the second approach point.

5.- Second probing movement.

Same as the first probing.

6.- Third approach movement.

Same as above.

7.- Third probing movement.

Same as above.

8.- Fourth approach movement.

Same as above.

9.- Fourth probing movement.

Same as above.

10.- Withdrawal

This movement consists of:

* Withdrawal to the fourth approach point.

* Movement of the probe in rapid (G00) and at a distance (B) above the boss to the real center (calculated) of the boss.

* Should (C0) be programmed, the probe will be moved to the point where the cycle was called.

  1st Movement along the longitudinal axis to the coordinate of the point (along this axis) from where the cycle was called.

  2nd Movement on the main work plane to the point where the cycle was called.

Once the cycle has been completed, the CNC will return the real values obtained after measurement, in the following global arithmetic parameter.

P294 **Boss diameter.**

P295 **Boss diameter error.** Difference between the real diameter and programmed diameter.

P296 **Real coordinate of the center along the abscissa axis.**

P297 **Real coordinate of the center along the ordinate axis.**

P298 **Error detected along the abscissa axis.** Difference between the real coordinate of the center and the programmed theoretical coordinate.

P299 **Error detected along the ordinate axis.** Difference between the real coordinate of the center and the programmed theoretical coordinate.

13. *PROGRAMMING IN HIGH-LEVEL LANGUAGE*

The CNC has a series of internal variables which can be accessed from the user program, from the PLC program or through DNC.

Access to these variables from the user program is gained with high-level commands.

Each of the system variables which can be accessed will be referred to by means of its mnemonic, and will be separated, according to their use, into read-only variables and read-write variables.

13.1 LEXICAL DESCRIPTION

All the words which form the high-level language of the numerical control must be written in capital letters except for associated texts which may be written in upper and lower case letters.

The following elements are available for high-level programming:

- Reserved words.
- Numerical constants
- Symbols

13.1.1 RESERVED WORDS

The set of words which the CNC uses in high-level programming for naming system variables, operators, control mnemonics, etc. are as follows:

ANAIn	**ANAOn**	**BLKN**	**CALL**	**CALLP**	**CLOCK**
CNCERR	**CNCFRO**	**CNCSSO**	**CYTIME**	**DATE**	**DEFLEX**
DEFLEY	**DEFLEZ**	**DFHOLD**	**DIGIT**	**DIST(X-C)**	**DNCERR**
DNCF	**DNCFPR**	**DNCFRO**	**DNCS**	**DNCSL**	**DNCSSO**
DPOS(X-C)	**DSBLK**	**DSTOP**	**DW**	**EFHOLD**	**ERROR**
ESBLK	**ESTOP**	**EXEC**	**FEED**	**FIRST**	**FLWE(X-C)**
FLWES	**FOZLO(X-C)**	**FOZONE**	**FOZUP(X-C)**	**FPREV**	**FRO**
FZLO(X-C)	**FZONE**	**FZUP(X-C)**	**GGSA**	**GGSB**	**GGSC**
GGSD	**GMS**	**GOTO**	**GSn**	**GTRATY**	**GUP n**
IB	**IF**	**INPUT**	**KEY**	**KEYSRC**	**LONGAX**
LUP (a,b)	**MCALL**	**MDOFF**	**MIRROR**	**MP(X-C)n**	**MPASn**
MPGn	**MPLCn**	**MPSn**	**MPSSn**	**MSG**	**MSn**
NBTOOL	**NXTOD**	**NXTOOL**	**ODW**	**OPEN**	**OPMODA**
OPMODB	**OPMODC**	**OPMODE**	**ORG(X-C)**	**ORG(X-C)n**	**ORGROA**
ORGROB	**ORGROC**	**ORGROI**	**ORGROJ**	**ORGROK**	**ORGROQ**
ORGROR	**ORGROS**	**ORGROT**	**ORGROX**	**ORGROY**	**ORGROZ**
PAGE	**PARTC**	**PCALL**	**PLANE**	**PLCCn**	**PLCERR**
PLCF	**PLCFPR**	**PLCFRO**	**PLCIn**	**PLCMn**	**PLCMSG**
PLCOF(X-C)	**PLCOn**	**PLCRn**	**PLCS**	**PLCSL**	**PLCSSO**
PLCTn	**PORGF**	**PORGS**	**POS(X-C)**	**POSS**	**PPOS(X-C)**
PRBST	**PRGF**	**PRGFIN**	**PRGFPR**	**PRGFRO**	**PRGN**
PRGS	**PRGSL**	**PRGSSO**	**PROBE**	**REPOS**	**RET**
ROTPF	**ROTPS**	**RPOSS**	**RPT**	**RTPOSS**	**SCALE**
SCALE(X-C)	**SCNCSO**	**SDNCS**	**SDNCSL**	**SDNCSO**	**SFLWES**
SK	**SLIMIT**	**SPEED**	**SPLCS**	**SPLCSL**	**SPLCSO**
SPOSS	**SPRGS**	**SPRGSL**	**SPRGSO**	**SREAL**	**SRPOSS**
SRTPOS	**SSLIMI**	**SSO**	**SSPEED**	**SSREAL**	**SSSO**
STPOSS	**SUB**	**SYMBOL**	**SYSTEM**	**SZLO(X-C)**	**SZONE**
SZUP(X-C)	**TIME**	**TIMER**	**TLFDn**	**TLFFn**	**TLFNn**
TLFRn	**TMZPn**	**TMZTn**	**TOD**	**TOIn**	**TOKn**
TOLn	**TOOL**	**TOOROF**	**TOOROS**	**TORn**	**TPOS(X-C)**
TPOSS	**TRACE**	**TZLO(X-C)**	**TZONE**	**TZUP(X-C)**	**WBUF**
WBUF	**WKEY**	**WRITE**			

Words ending in (X-C) indicate a set of 9 elements formed by the corresponding root followed by X,Y,Z,U,V,W,A,B and C.

ORG(X-C)-> ORGX,ORGY,ORGZ,ORGU,ORGV,ORGW,ORGA,ORGB,ORGC

All the letters of the alphabet A-Z are also reserved words, as they can make up a high-level language word when used alone.

13.1.2 NUMERICAL CONSTANTS

The blocks programmed in high-level language allow numbers in decimal format which do not exceed the format ±6.5 and numbers in hexadecimal format, in which case they must be preceded by the **$** sign, with a maximum of 8 digits.

The assignment to a variable of a constant higher than the format ±6,5 will bemade by means of arithmetic parameters, by means of arithmetic expressions or by means of constants expressed in hexadecimal format.

Example: To assign the value 100000000 to the variable "TIMER", It can be done in one of the following ways:

```
(TIMER  = $5F5E100)
(TIMER  = 10000 * 10000)
(P100   = 10000 * 10000)
(TIMER  = P100)
```

When the CNC is working in metric system (mm) resolution is in tenths of a micron, and figures are programmed in the format ±5.4 (positive or negative, with 5 integers and 4 decimals), and if the CNC is operating in inches, resolution is in 0.00001 inches, figures being programmed with the format ±4.5 (positive or negative, with 4 integers and 5 decimals).

For the convenience of the programmer, this control always allows the format ±5.5 (positive or negative, with 5 integers and 5 decimals), adjusting each number appropriately to the working units every time they are used.

13.1.3. SYMBOLS

The symbols used in high-level language are:

( ) " = + - * / ,

13.2 VARIABLES

The internal CNC variables which can be accessed by high-level language are grouped in tables and can be read-only or read-write variables.

There is a group of mnemonics for showing the different fields of the table of variables. In this way, if it is required to access an element from one of these tables, the required field will be indicated by means of the corresponding mnemonic (for example TOR) and then the required element (TOR3).

The variables available at the CNC can be classified in the following way:

- General purpose parameters or variables
- Variables associated with tools.
- Variables associated with zero offsets.
- Variables associated with machine parameters
- Variables associated with work zones
- Variables associated with feedrates
- Variables associated with position coordinates
- Variables associated with the spindle
- Variables associated with the PLC
- Variables associated with local parameters
- Other variables

Variables which access to real values of the CNC interrupt the preparation of blocks and the CNC waits for each command to be performed before restarting block preparation.

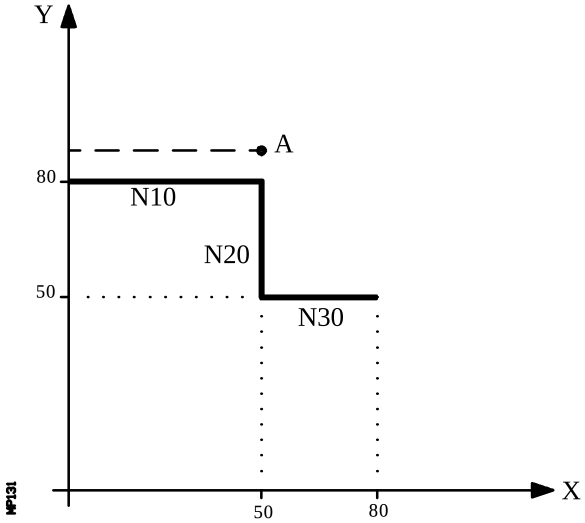
Thus, precaution must be taken when using this type of variable, as should they be placed between machining blocks which are working with compensation, undesired profiles may be obtained.

Example:

The following program blocks are performed in a section with G41 compensation.

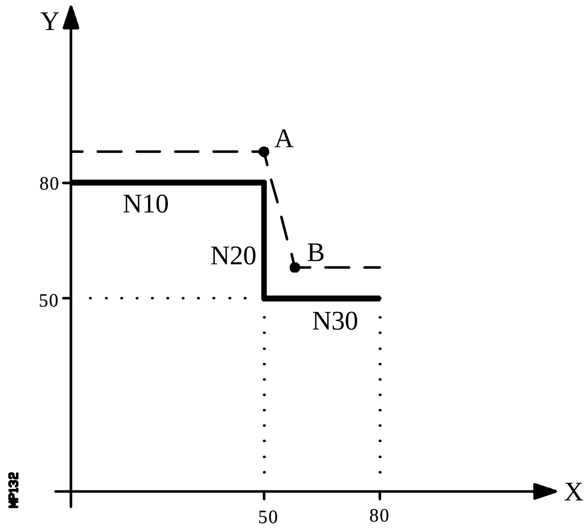
```
.......
.......
N10 X50 Y80
N15 (P100=POSX);Assigns the value of the real coordinate in X to parameter P100
N20 X50 Y590
N30 X80 Y50
.......
.......
```

Block N15 interrupts block preparation and the execution of block N10 will finish at point A.



Once the execution of block N15 has ended, the CNC will continue block preparation from block N20 on.

As the next point corresponding to the compensated path is point “B”, the CNC will move the tool to this point, executing path “A-B”.



As can be observed, the resulting path is not the desired one, and therefore it is recommended to avoid the use of this type of variable in sections requiring tool compensation.

13.2.1 GENERAL PURPOSE PARAMETERS OR VARIABLES

The CNC has two types of general purpose variables: local parameters P0-P25 and global parameters P100-P299.

Programmers may use general purpose variables when editing their own programs. Later and during execution, the CNC will replace these variables with the values assigned to it at that time.

Example:

```
GP0 XP1 Y100                          —> G1 X-12.5 Y100
(IF(P100*P101 EQ P102)GOTO N100)      —> (IF(2*5 EQ 12)GOTO N100)
```

The use of these global purpose variables will depend on the type of block in which they are programmed and the channel of execution.

In block programmed in ISO code parameters can be associated with all fields, G X..C F S T D M. The block label number will be defined with a numerical value.

If parameters are used in blocks programmed in high-level language, these can be programmed within any expression.

Programmes which are executed in the user channel may contain any global parameter, but may not use local parameters.

The CNC will update the parameter table after processing the operations indicated in the block which is in preparation. This operation is always done before executing the block and for this reason, the values shown in the table do not necessarily have to correspond to the block being executed.

If the Execution Mode is abandoned after interrupting the execution of the program, the CNC will update the parameter tables with values corresponding to the block which was being executed.

When accessing the local parameter and global parameter table, the value assigned to each parameter may be expressed in decimal notation (4127.423) or in scientific notation (=23476 E-3).

This CNC has high level statements which allow the definition and use of subroutines which can be called from the main program, or from another subroutine, it also being possible to call a second subroutine, from the second to a third, etc. The CNC limits these calls, allowing up to a maximum of 15 nesting levels.

26 local parameters (P0-P25) can be assigned to a subroutine. These parameters which will be unknown for blocks external to the subroutine may be referenced by the blocks of this subroutine.

The CNC allows local parameters to be assigned to more than one subroutine, 6 nesting levels of local parameters being possible, within the 15 nesting levels of a subroutine.

Local parameters used in high-level language may be defined either using the above format or by using the letter A-Z, except for N, so that A is equal to P0 and Z to P25.

The following example shows these two methods of definition:

```
(IF((P0+P1)   *  P2/P3   EQ  P4)    GOTO N100)
(IF((A+B)     *  C/D     EQ  E)     GOTO N100)
```

When using a parameter name (letter) for assigning a value to it (A instead of P0, for example), if the arithmetic expression is a constant, the statement can be abbreviated as follows:

(P0 = 13.7) —> (A = 13.7) —> (A13.7)

Be careful when using parenthesis since M30 is not the same as (M30). The CNC interprets (M30) as a high level statement meaning (P12 = 30) and not the execution of the miscellaneous M30 function.

The global parameter (P100-P299) can be used throughout the program by any block, irrespective of the nesting level.

Multiple machining (G60, G61, G62, G63, G64, G65) and machining canned cycles (G69, G81 ... G89) use a local parameter nesting level when active.

Machining canned cycles use the global parameter P299 for internal calculations and probing canned cycles use global parameters P294 to P299.

13.2.2 *VARIABLES ASSOCIATED WITH TOOLS*

These variables are associated with the tool offset table, tool table and tool magazine table, so the values which are assigned to or read from these fields will comply with the formats established for these tables.

Tool offset table:

R,L,I,K	They are given in the active units: If G70, in inches. Max.: ±3937.00787 If G71, in millimeters. Max.: ±99999.9999 If rotary axis in degrees. Max.: ±99999.9999

Tool table

Tool offset number	0...NT OFFSET (maximum 255)
Family code	If normal tool $0 \leq n < 200$ If special tool $200 \leq n \leq 255$
Nominal life	0...65535 minutes or operations.
Real life	0.99999.99 minutes or 99999 operations

Tool magazine table

Contents of each magazine position

	Tool number 1 ...NTOOL (maximum 255) 0 Empty -1 Cancelled

Position of tool in magazine

	Position number 1 ..NPOCKET (maximum 255) 0 On spindle -1 Not found -2 In change position

Read-only variables

TOOL: Returns the active tool number
(P100 = TOOL); assigns the number of the active tool to P100

TOD: Returns the active tool offset number

NXTOOL: Returns the next tool number, selected but is awaiting the execution of M06 to be active.

NXTOD: Returns the number of the tool offset corresponding to the next tool, selected but is awaiting the execution of M06 to be active.

TMZPn: Returns the position occupied in the tool magazine by the indicated tool (n).

Read-write variables

TORn: This variable allows the value assigned to the Radius of the indicated tool offset (n) on the tool offset table to be read or modified.

(P110 = TOR3);	Assigns the R value of tool offset 3 to Parameter 3).
(TOR3 = P111);	Assigns the value of parameter P111 to R of tool offset 3)

TOLn: This variable allows the value assigned to the Length of the indicated tool offset (n) to be read or modified on the tool offset table.

TOIn: This variable allows the value assigned to the radius wear (I) of the indicated tool offset (n) to be read or modified on the tool offset table.

TOKn: This variable allows the value assigned to the length wear (K) of the indicated tool offset (n) to be read or modified on the tool offset table.

TLFDn: This variable allows the tool offset number of the indicated tool (n) to be read or modified on the tool table.

TLFFn: This variable allows the family code of the indicated tool (n) to be read or modified on the tool table.

TLFNn: This variable allows the value assigned as the nominal life of the indicated tool (n) to be read or modified on the tool table.

TLFRn: This variable allows the value corresponding to the real life of the indicated tool (n) to be read or modified on the tool table.

TMZTn: This variable allows the contents of the indicated position (n) to be read or modified on the tool magazine table.

13.2.3 *VARIABLES ASSOCIATED WITH ZERO OFFSETS*

These variables are associated with the zero offsets and may correspond to the table values or to those currently preset either by means of function G92 or manually in the JOG mode.

The zero offsets which are possible in addition to the additive offset indicated by the PLC, are G54, G55, G56, G57, G58 and G59.

The values for each axis are given in the active units:

If G70, in inches. Max.: ±3937.00787
If G71, in millimeters. Max.: ±99999.9999
If rotary axis in degrees. Max.: ±99999.9999

Although there are variables which refer to each axis, the CNC only allows those referring to the selected axes in the CNC. Thus, if the CNC controls axes X, Y, Z, U and B, it only allows the variables ORGX, ORGY, ORGZ,. ORGU and ORGB in the case of ORG(X-C).

Read-only variables

ORG(X-C) Returns the value of the active zero offset in the selected axis. The value of the additive offset indicated by the PLC is not included in this value.

(P100 = ORGX); assigns to P100 the X value of the part zero active for the X axis. This value could have been set either by means of function G92 or by the variable "**ORG(X-C)n**".

PORGF: Returns the abscissa value of the polar coordinate origin with respect to the Cartesian origin.

PORGS: Returns the ordinate value of the polar coordinate origin with respect to the cartesian origin.

Read-write variables

ORG(X-C)n: This variable allows the value of the selected axis to be read or modified on the table corresponding to the indicated zero offset (n).

(P110 = ORGX55); Assigns the value of X to parameter P110 on the table corresponding to zero offset G55.

(ORGY 54 = P111); Assigns the value of parameter P111 to the Y axis on the table corresponding to G54 zero offset.

PLCOF(X-C) This variable allows the value of the selected axis to be read or modified on the additive zero offset table indicated by the PLC.

If any of the PLCOF(X-C) variables are accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to begin block preparation again.

13.2.4 *VARIABLES ASSOCIATED WITH MACHINE PARAMETERS*

Variables associated with machine parameters are read-only variables.

In order to become familiar with the values returned it is advisable to consult the installation and start-up manual.

Values 1/0 correspond to the parameters which are defined with YES/NO, +/- and ON/OFF.

The coordinate and feedrate values are given in the active units:

If G70, in inches. Max.: ±3937.00787
If G71, in millimeters. Max.: ±99999.9999
If rotary axis in degrees. Max.: ±99999.9999

Read-only variables

MPGn: Returns the value assigned to the general machine parameter (n).

(P110=MPG 8);assigns the value of the general machine parameter "INCHES" to parameter P110, if millimeters P110=0 and if inches P110=1.

MP(X-C)n Returns the value which was assigned to the machine parameter (n) of the indicated axes.

(P110=MPY 1); assigns the value of the machine parameter P1 to arithmetic parameter P110 of the Y axis "DFORMAT", which indicates the format used in its display.

MPSn: Returns the value which was assigned to the main spindle machine parameter (n).

MPSSn: Returns the value which was assigned to the secondary spindle machine parameter (n).

MPASn: Returns the value of the machine parameter (n) for the auxiliary spindle.

MPLCn: Returns the value which was assigned to the PLC machine parameter (n)

13.2.5 *VARIABLES ASSOCIATED WITH WORK ZONES*

Variables associated with work zones are read-only variables.

The values of the limits are given in the active units:

If G70, in inches. Max.: ±3937.00787
If G71, in millimeters. Max.: ±99999.9999
If rotary axis in degrees. Max.: ±99999.9999

The status of the work zones is determined according to the following code:

0 = Disabled.
1 = Enabled as no-entry zone.
2 = Enabled as no-exit zone.

Read-only variables

FZONE: Returns the status of work zone 1.
(P100=FZONE); assigns to parameter P100 the status of work zone 1.

FZLO(X-C) Returns the value of the lower limit of Zone 1 according to the selected axis (X-C).

FZUP(X-C) Returns the value of the upper limit of Zone 1 according to the selected axis (X-C).

SZONE: Status of work zone 2.
SZLO(X-C) Lower limit of Zone 2 according to the selected axis (X-C).
SZUP(X-C) Upper limit of Zone 2 according to the selected axis (X-C).

TZONE: Sstatus of work zone 3.
TZLO(X-C) Lower limit of Zone 3 according to the selected axis (X-C).
TZUP(X-C) Upper limit of Zone 3 according to the selected axis (X-C).

FOZONE: Status of work zone 4.
FOZLO(X-C) Lower limit of Zone 4 according to the selected axis (X-C).
FOZUP(X-C) Upper limit of Zone 4 according to the selected axis (X-C).

FIZONE: Status of work zone 5.
FIZLO(X-C) Lower limit of Zone 5 according to the selected axis (X-C).
FIZUP(X-C) Upper limit of Zone 5 according to the selected axis (X-C).

13.2.6 VARIABLES ASSOCIATED WITH FEEDRATES

Read-only variables associated with the actual feedrate

FREAL: Returns the real feedrate of the CNC in mm/min. or inches/min.

(P100 = FREAL); Assigns the real feedrate value of the CNC to parameter P100

Read-only variables associated with function G49

FEED: Returns the feedrate selected in the CNC by means of the G94 function. This will be in mm/minute or inches/minute.

This feedrate can be indicated by program, by the PLC or DNC, and the CNC selects one of these, the one with the highest priority being that indicated by DNC and the one with the lowest priority that indicated by program.

DNCF: Returns the feedrate, in mm/minute or inches/minute, selected by DNC. If this has a value of 0 it means that it is not selected.

PLCF: Returns the feedrate, in mm/minute or inches/minute, selected by PLC. If this has a value of 0 it means that it is not selected.

PRGF: Returns the feedrate, in mm/minute or inches/minute, selected by program.

Read-only variables associated with function G95

FPREV: Returns the feedrate selected in the CNC by means of the G95 function. This will be in mm/rev. or inches/rev.

This advance can be indicated by program, by the PLC or DNC, and the CNC selects one of these, the one with the highest priority being that indicated by DNC and the one with the lowest priority that indicated by program.

DNCFPR: Returns the feedrate, in mm/rev. or inches/rev., selected by DNC. If this has a value of 0 it means that it is not selected.

PLCFPR: Returns the feedrate, in mm/rev. or inches/rev., selected by PLC. If this has a value of 0 it means that it is not selected.

Read-only variables associated with function G32

PRGFIN: Returns the feedrate, in 1/min selected by program.

Also, the CNC variable FEED associated with G94 will show the resulting feedrate in mm/min or inches/min.

Read-only variables associated with Feedrate Override

PRGFPR: Returns the feedrate, in mm/rev. or inches/rev., selected by program.

FRO: Returns the Feedrate Override (%) selected at the CNC. This will be given by an integer between 0 and “MAXFOVR” (maximum 255).

This feedrate percentage may be indicated by the PLC, by DNC or from the front panel, and the CNC will select one of them, the order of priority (from highest to lowest) being: by program, by DNC, by PLC and from the switch.

DNCFRO: Returns the Feedrate Override % selected by DNC. If this has a value of 0 it means that it is not selected.

PLCFRO: Returns the Feedrate Override % selected by PLC. If this has a value of 0 it means that it is not selected.

CNCFR0: Returns the Feedrate Override % selected from the switch at the CNC Operator Panel.

PLCCFR: Returns the Feedrate Override % selected for the PLC execution channel.

Read-write variables

PRGFRO: This variable allows the feedrate percentage selected by program to be read or modified. This will be given by an integer between 0 and “MAXFOVR” (maximum 255). If it has a value of 0 this means that it is not selected.

(P110 = PRGFRO); assigns to P110 the % of feedrate override selected by program

(PFRGFRO = P111); sets the feedrate override % selected by program to the value of P111.

13.2.7 VARIABLES ASSOCIATED WITH COORDINATES

The coordinate values for each axis are given in the active units:

If G70, in inches. Max.: ±3937.00787
If G71, in millimeters. Max.: ±99999.9999
If rotary axis in degrees. Max.: ±99999.9999

Read-only variables

PPOS(X-C) Returns the programmed theoretical coordinate of the selected axis.

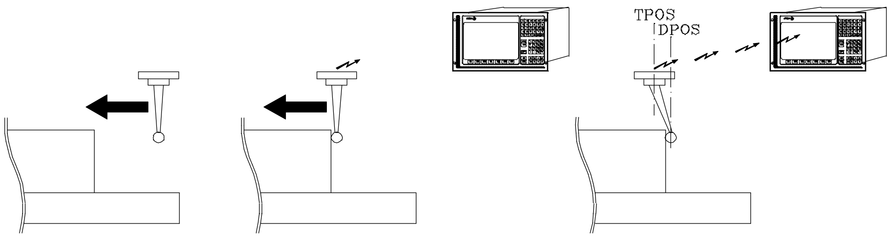
(P100) = PPOSX); assigns to P100 the programmed theoretical position of the X axis.

POS(X-C) Returns the real coordinate of the selected axis referred to machine reference zero (home).

TPOS(X-C) Returns the theoretical coordinate (real + following error) of the selected axis referred to machine reference zero (home).

DPOS(X-C) The CNC updates this variable whenever probing operations are carried out, same as with G75, G76 functions.

When the digital probe and the CNC communicate with each other via infrared beams, there could be a delay of a few milliseconds from when the probe touches the part until the moment the CNC receives the probe signal.



Although the probe keeps moving until the CNC receives the probe signal, the CNC assumes the value assigned to general machine parameter PRODEL and provides the following information (variables associated with coordinates):

TPOS Actual position of the probe when the CNC receives the probe signal.
DPOS Theoretical position of the probe when it touched the part.

FLWE(X-C) Returns the amount of following error of the selected axis.

DEFLEX
DEFLEY
DEFLEZ: They return the current deflection of the Renishaw probe SP2 along each axis, **X, Y, Z.**

When accessing one of these variables (POS(X-C), TPOS(X-C), DPOS(X-C), FLWE(X-C), DEFLEX, DEFLEY or DEFLEZ), **block preparation is interrupted** and the CNC waits for that command to be executed before resuming block preparation.

Read-write variables

DIST(X-C) These variables allow the distance travelled by the selected axis to be read or modified. This value is accumulative and it is very useful when it is required to perform an operation which depends on the distance travelled by the axes, for example: in their lubrication.

(P100= DISTX); assigns to P100 the distance travelled by the X axis

(DISTZ = P111); presets the variable indicating the distance travelled by the Z axis with the value of arithmetic parameter P111.

If any of the DIST(X-C) variables are accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

LIMPL(X-C):
LIMMI(X-C): With these variables, it is possible to set a second travel limit for each axis, LIMPL for the upper limit and LIMMI for the lower limit.

Since the second limits are activated or deactivated from the PLC, through general logic input ACTLIM2 (M5052), besides setting the limits, an auxiliary M code must be executed to let it know.

It is also recommended to execute function G4 after the change so the CNC executes the following blocks with the new limits.

The second travel limit will be taken into consideration when the first one has been set using axis machine parameters LIMIT+ (P5) and LIMIT- (P6).

13.2.8 VARIABLES ASSOCIATED WITH THE ELECTRONIC HANDWHEELS

Read-only variables

HANPF **HANPS**
HANPT **HANPFO**

They return the number of pulses of the first (HANPF), second (HANPS), third (HANPT) or fourth (HANPFO) handwheel received since the CNC was turned on. Regardless of whether the handwheel is connected to the feedback inputs or to the PLC inputs.

HANFCT Returns the multiplying factor set from the PLC for each handwheel.

It must be used when having several electronic handwheels or when having a single handwheel but applying different multiplying factors (x1, x10, x100) to each axis.

		C			B			A			W			V			U			Z			Y			X			
		c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	LSB

Once the switch is position at one of the handwheel positions, the CNC checks this variable and depending on the values assigned to the "c b a" bits of each axis it applies the multiplying factor selected for each of them.

c b a	
0 0 0	The one indicated by the switch on the operator panel or keyboard
0 0 1	x1 Factor
0 1 0	x10 Factor
1 0 0	x100 Factor

If there are more than one bit to "1" on an axis, the least significant bit is taken into account.

Thus: c b a
1 1 1 x1 Factor
1 1 0 x10 Factor

Note: The screen always shows the value selected at the switch.

HBEVAR It must be used when having a Fagor HBE handwheel.

It indicates whether the HBE handwheel is enabled or not, the axis to be jogged and the multiplying factor being applied (x1, x10, x100).

					C			B			A			W			V			U			Z			Y			X			
	*	^			c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	LSB

(*) It indicates whether the reading of the HBE handwheel is to be considered or ignored.
0 It is ignored.
1 It is considered.

(^) When the machine has a general handwheel and individual ones (associated with an axis), it indicates which handwheel has priority when they both turn at the same time.
0 The individual handwheel has priority. The relevant axis ignores the pulses from the general handwheel, but not the rest of the axes.
1 The general handwheel has priority. It ignores the pulses from the individual handwheel.

(c b a) indicate the axis to be jogged and the selected multiplying factor.

c	b	a	
0	0	0	It is not to be moved
0	0	1	x1 factor
0	1	0	x10 factor
1	0	0	x100 factor

If there are several axes selected, the CNC attends to the next one in priority: X, Y, Z, U, V, W, A, B, C.

If there are more than one bit to "1" on an axis, the least significant bit is taken into account. Thus:

c	b	a	
1	1	1	x1 factor
1	1	0	x10 factor

The HBE handwheel has priority. In other words, regardless of the mode selected at the CNC switch (continuous or incremental JOG, handwheel) HBEVAR is defined as other than "0", the CNC goes into handwheel mode.

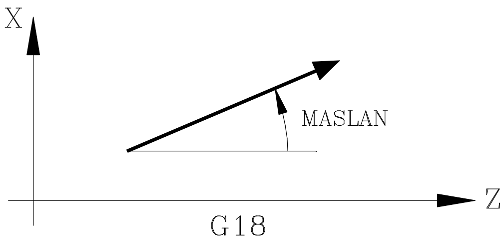
It highlights the selected axis and the multiplying factor selected by PLC. When the HBEVAR variable is set to "0", it displays the mode selected at the switch again.

For further information, refer to chapter 4 "Example of PLC program for Fagor HBE handwheel" in this manual.

Read and write variables

MASLAN must be used when the "Path Handwheel" mode is selected.

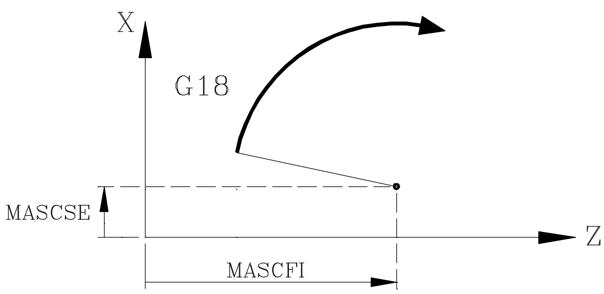
Indicates the angle of the linear path.



MASCFI
MASCSE

Must be used when the "Path Handwheel" mode is selected.

On circular paths, they indicate the coordinates of the arc center.



13.2.9 *VARIABLES ASSOCIATED WITH THE MAIN SPINDLE*

In these variables associated with the spindle, their values are given in revolutions per minute and the main spindle override values are given in integers from 0 to 255.

Certain variables interrupt block preparation (it is indicated in each one) and the CNC waits for that command to be executed before resuming block preparation.

Read-only variables

SREAL: Returns the real main spindle turning speed in revolutions per minute. It interrupts block preparation.

(P100 = SREAL); assigns to P100 the real turning speed of the main spindle.

SPEED: Returns, in revolutions per minute, the main spindle speed selected at the CNC.

This turning speed can be indicated by program, by the PLC or DNC, and the CNC selects one of these, the one with the highest priority being that indicated by DNC and the one with the lowest priority that indicated by program.

DNCS: Returns the turning speed in revolutions per minute, selected by DNC. If this has a value of 0 it means that it is not selected.

PLCS: Returns the turning speed in revolutions per minute selected by PLC. If this has a value of 0 it means that it is not selected.

PRGS: Returns the turning speed in revolutions per minute, selected by program.

SSO: Returns the Override (%) of the main spindle speed selected at the CNC. This will be given by an integer between 0 and “MAXSOVR” (maximum 255).

This spindle speed percentage may be indicated by the PLC, by DNC or from the front panel, and the CNC will select one of them, the order of priority (from highest to lowest) being: by program, by DNC, by PLC and from the front panel.

DNCSSO: Returns the main spindle speed percentage selected by DNC. If this has a value of 0 it means that it is not selected.

PLCSSO: Returns the main spindle speed percentage selected by PLC. If this has a value of 0 it means that it is not selected.

CNCSSO: Returns the main spindle speed percentage selected from the front panel.

SLIMIT: Returns, in revolutions per minute, the value established for the main spindle speed limit selected at the CNC.

This limit can be indicated by program, by the PLC or DNC, and the CNC selects one of these, the one with the highest priority being that indicated by DNC and the one with the lowest priority that indicated by program.

DNCSL: Returns the main spindle speed limit in revolutions per minute, selected by DNC. If this has a value of 0 it means that it is not selected.

PLCSL: Returns the main spindle speed limit in revolutions per minute selected by PLC. If this has a value of 0 it means that it is not selected.

PRGSL: Returns the main spindle speed limit in revolutions per minute, selected by program.

POSS: Returns the main spindle real position value. Its value may be between ±99999.9999. It interrupts block preparation.

RPOSS: Returns the main spindle real position value. Its value may be between 0 and 360º. It interrupts block preparation.

TPOSS: Returns the main spindle theoretical position value. Its value may be between ±99999.9999. It interrupts block preparation.

RTPOSS: Returns the main spindle theoretical position value. Its value may be between 0 and 360º. It interrupts block preparation.

FLWES: Returns the spindle following error. It interrupts block preparation.

SYNCER It returns, in degrees (max ±99999.9999), the following error of the second spindle with respect to the main spindle when they are synchronized in position.

When accessing one of these variables (POSS, RPOSS, TPOSS RTPOSS or FLWES), **block preparation is interrupted** and the CNC waits for that command to be executed before resuming block preparation.

Read-write variables

PRGSSO: This variable allows the percentage of the main spindle speed selected by program to be read or modified. This will be given by an integer between 0 and "MAXSOVR" (maximum 255). If this has a value of 0 it means that it is not selected.

(P110 = PRGSSO); assigns to P110 the % of the main spindle speed selected by program.

(PRGSSO = P111); sets the value indicating the main spindle speed % seleceted by program to the value of arithmetic parameter P111.

13.2.10 *VARIABLES ASSOCIATED WITH THE 2ND SPINDLE*

In these variables associated with the spindle, their values are given in revolutions per minute and the 2nd spindle override values are given in integers from 0 to 255.

Read-only variables

SSREAL: Returns the real 2nd spindle turning speed in revolutions per minute.

(P100 = SRSEAL); assigns to P100 the real turning speed of the 2nd spindle.

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

SSPEED: Returns, in revolutions per minute, the 2nd spindle speed selected at the CNC.

This turning speed can be indicated by program, by the PLC or DNC, and the CNC selects one of these, the one with the highest priority being that indicated by DNC and the one with the lowest priority that indicated by program.

SDNCS: Returns the turning speed in revolutions per minute, selected by DNC. If this has a value of 0 it means that it is not selected.

SPLCS: Returns the turning speed in revolutions per minute selected by PLC. If this has a value of 0 it means that it is not selected.

SPRGS: Returns the turning speed in revolutions per minute, selected by program.

SSSO: Returns the Override (%) of the 2nd spindle speed selected at the CNC. This will be given by an integer between 0 and "MAXSOVR" (maximum 255).

This spindle speed percentage may be indicated by the PLC, by DNC or from the front panel, and the CNC will select one of them, the order of priority (from highest to lowest) being: by program, by DNC, by PLC and from the front panel.

SDNCSO: Returns the 2nd spindle speed percentage selected by DNC. If this has a value of 0 it means that it is not selected.

SPLCSO: Returns the 2nd spindle speed percentage selected by PLC. If this has a value of 0 it means that it is not selected.

SCNCSO: Returns the 2nd spindle speed percentage selected from the front panel.

SSLIMI: Returns, in revolutions per minute, the value established for the 2nd spindle speed limit selected at the CNC.

This limit can be indicated by program, by the PLC or DNC, and the CNC selects one of these, the one with the highest priority being that indicated by DNC and the one with the lowest priority that indicated by program.

SDNCSL: Returns the 2nd spindle speed limit in revolutions per minute, selected by DNC. If this has a value of 0 it means that it is not selected.

SPLCSL: Returns the 2nd spindle speed limit in revolutions per minute selected by PLC. If this has a value of 0 it means that it is not selected.

SPRGSL: Returns the 2nd spindle speed limit in revolutions per minute, selected by program.

SPOSS: Returns the 2nd spindle real position value, when it is in closed loop (M19). Its value will be given in 0.0001 degree units between ±999999999.

SRPOSS: Returns the 2nd spindle real position value. Its value will be given in 0.0001 degree units between 0 and 360º,

TPOSS: Returns the 2nd spindle theoretical position value. Its value will be given in 0.0001 degree units between ±999999999.

SRTPOS: Returns the 2nd spindle theoretical position value. Its value will be given in 0.0001 degree units between 0 and 360º.

SFLWES: Returns the spindle following error when it is operating in closed loop (M19).

When accessing one of these variables (SPOSS, SRPOS, STPOSS, SRTPOS or SFLWES), **block preparation is interrupted** and the CNC waits for that command to be executed before resuming block preparation.

Read-write variables

SPRGSO: This variable allows the percentage of the 2nd spindle speed selected by program to be read or modified. This will be given by an integer between 0 and “MAXSOVR” (maximum 255). If this has a value of 0 it means that it is not selected.

(P110 = SPRGSO); assigns to P110 the % of the 2nd spindle speed selected by program.

(SPRGSO = P111); sets the value indicating the 2nd spindle speed % selecected by program to the value of arithmetic parameter P111.

13.2.11 VARIABLES ASSOCIATED WITH THE LIVE TOOL

Read-only variables

ASPROG It must be used within the subroutine associated with function M45.
It returns the rpm programmed by M45 S
When programming only M45, the variable assumes the value of "0".

The ASPROG variable is updated just before executing M45 so it is updated when executing the associated subroutine.

13.2.12 *VARIABLES ASSOCIATED WITH THE PLC*

It should be borne in mind that the PLC has the following resources:

Inputs	(I1 thru I256)
Outputs	(O1 thru O256).
Marks	M1 thru M5957)
Registers	(R1 thru R499) of 32 bits each.
Timers	(T1 thru T256) with a timer count in 32 bits.
Counters	(C1 thru C256) with a counter count in 32 bits.

If any variable is accessed which allows the status of a PLC variable to be read or modified (I,O,M,R,T,C), **block preparation is interrupted** and the CNC waits for this command to be executed in order to restart block preparation.

Read-only variables

PLCMSG: Returns the number of the active PLC message with the highest priority and will coincide with the number displayed on screen (1...128). If there is none, it returns 0.

(P100 = PLCMSG); assigns to P100 the number of the active PLC message with the highest priority.

Read-write variables

PLCIn: This variable allows 32 PLC inputs to be read or modified starting with the one indicated (n).

The value of the inputs which are used by the electrical cabinet cannot be modified as their values are determined by it. Nevertheless, the status of the remaining inputs can be modified.

PLCOn: This variable allows 32 PLC outputs to be read or modified starting from the one indicated (n).

Bit	31	30	29	28	27	26	25	24	23	22	21	20		6	5	4	3	2	1	0
	0	0	0	0	0	0	0	0	0	0	0	0		0	0	0	1	1	1	1
Output	53	52	51	50	49	48	47	46	45	44	43	42		28	27	26	25	24	23	22

PLCMn: This variable allows 32 PLC marks to be read or modified starting from the one indicated (n).

PLCRn: This variable allows the status of 32 register bits to be read or modified starting from the one indicated (n).

PLCTn: This variable allows the timer count to be read or modified starting from the one indicated (n).

PLCCn: This variable allows the counter count to be read or modified starting from the one indicated (n).

13.2.13 VARIABLES ASSOCIATED WITH LOCAL PARAMETERS

The CNC allows 26 local parameters (P0-P25) to be assigned to a subroutine, by using mnemonics PCALL and MCALL.

In addition to performing the required subroutine these mnemonics allow local parameters to be initialized.

Read-only variables

CALLP: Allows us to know which local parameters have been defined and which have not, in the call to the subroutine by means of the PCALL or MCALL mnemonic.

The information will be given in the 26 least significant bits (bits 0..25), each of these corresponding to the local parameter of the same number, as well as bit 12 corresponding to P12.

Each bit will indicate if the corresponding local parameter has been defined (=1) or not (=0).

31	30	29	28	27	26	25	24	23	22	21	20	....	6	5	4	3	2	1	0
0	0	0	0	0	0	*	*	*	*	*	*	....	*	*	*	*	*	*	*

Example:

```
(PCALL 20, P0=20, P2=3, P3=5)       ;Call to subroutine 20.
....
....
(SUB 20)                            ;Beginning of subroutine 20
P100=CALLP)
....
....
```

In parameter P100 the following will be obtained:

0000	0000	0000	0000	0000	0000	0000	1101

LSB

13.2.14 SERCOS VARIABLES

They are used for data exchange between the CNC and the servo drives via Sercos interface.

Read-only variables

TSVAR(X-C) **identifier**.................... for the axes
TSVARS **identifier**.................... for the main spindle
TSSVAR **identifier**.................... for the second spindle

It returns the third attribute of the sercos variable corresponding to the "identifier". The third attribute is used in particular software applications and its information is coded according to the Sercos standard.

(P110=SVARX 40) assigns to parameter P110 the third attribute of the sercos variable of identifier 40 of the X axis which corresponds to "VelocityFeedback"

Write-only variables

SETGE(X-C) .... for the axes
SETGES .... for the main spindle
SSETGS .... for the second spindle

The drive may have up to 8 work ranges or gears (0 through 7). Sercos identifier 218, GearRatioPreselection.

It may also have up to 8 parameter sets (0 through 7). Sercos indentifier 217, ParameterSetPreselection.

These variables permit changing the work range (gear) or the paramete set for each drive.

The 4 least significant bits of these variables must indicate the work gear and the 4 most significant bits the parameter set to be selected.

Read-Write variables

SVAR(X-C) **identifier** .... for the axes
SVARS **identifier** .... for the main spindle
SSVAR **identifier** .... for the second spindle

They permit reading or modifying the value of the sercos variable corresponding to the axis identifier.

(P110=SVARX 40) assigns to parameter P110 the value of the sercos variable of identifier 40 of the X axis which corresponds to the "VelocityFeedback"

13.2.15 SOFTWARE & HARDWARE CONFIGURATION VARIABLES

Read variables

HARCON Indicates, with bits, the CNC hardware configuration.

The bit will be set to "1" when its corresponding configuration is available.

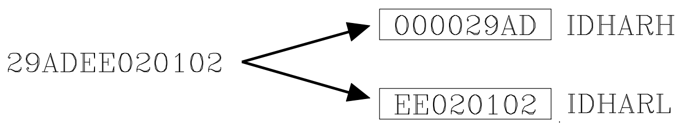
bit	
0	Turbo board
4,3,2,1	0100 8040 model
5	Sercos (digital model)
6	Reserved
9,8,7	000 Expansion board missing 001 "Feedback + I/O" expansion board 010 Feedback-only expansion board 011 I/O-only expansion board
10	Axis board with 12-bit (=0) or 16-bit (=1) Digital/Analog converter.
12, 11	Reserved
14, 13	Reserved
15	It has CAN (digital module)
18,17,16	Keyboard type (technical service department)
20,19	CPU type (technical service department)
23,22,21	000 Memkey Card (4M) 010 Memkey Card (24M) 110 Memkey Card (512M) 111 Memkey Card (2M)

IDHARH
IDHARL

They return, in BCD code, the hardware identification number corresponding to the "Memkey Card". It is the number appearing on the software diagnosis screen.

Since the identification number has 12 digits, the IDHARL variable shows the 8 least significant bits and the IDHARH the 4 most significant bits.

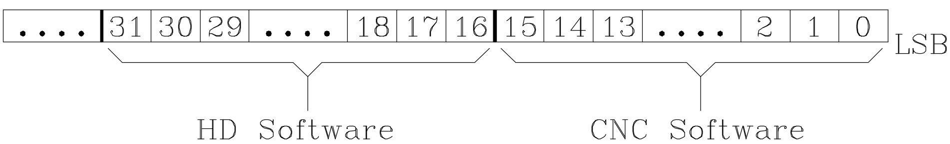
Example:



SOFCON

They return the software version numbers for the CNC and the Hard Disk.

Bits 15-0 return the CNC software version (4 digits)
Bits 31-16 return the software version of the Hard Disk (HD) (4 digits)



For example, SOFCON 01010311 indicates

Hard Disk (HD) software version	0101
CNC software version	0311

13.2.16 VARIABLES ASSOCIATED WITH TELEDIAGNOSIS

Read-only variables

HARSWA
HARSWB They return, in 4 bits, the configuration of the central unit (CPU). Logic address set on each board through micro-switches (see section 1.2 of the installation manual).

HARSWA

bits	31 - 28	27 - 24	23 - 20	19 - 16	15 - 12	11 - 8	7 - 4	3 - 0	
board	Large Sercos	I/O 4	I/O 3	I/O 2	I/O 1	Axes	Turbo	CPU	LSB

HARSWB

bits	31 - 28	27 - 24	23 - 20	19 - 16	15 - 12	11 - 8	7 - 4	3 - 0	
board						Small Sercos		HD	LSB

The CPU board must be present in all configurations and set with a value of "0".

In the rest of the cases, if there is no board, it returns a value of "0".

The sercos board may be either large (occupying the whole module) or small installed in the CPU module (1 if inserted in COM1 and 2 if inserted in COM2).

HARTST It returns the result of the Hardware test. The information comes at the lowest bits, with a "1" if wrong and a "0" right or if the relevant board is missing.

bit 13	bit 12	bit 11	bit 10	bit 9	bit 8	bit7	bit 6	bit 5	bit 4	bit 3	bit 2	bit 1	bit 0	
Inside temperature	Board voltage					Supply voltages								
	I/O 3	I/O 2	I/O 1		Axes	+3.3 V	GND	GNDA	-15 V	+15 V	Battery	-5 V	+5 V	LSB

MEMTST It returns the result of the Memory test. Each data uses 4 bits that will be at "1" if the test is OK and will have a value other than "1" if there is an error.

bits	30	....	19 - 16	15 - 12	11 - 8	7 - 4	3 - 0	
Test	Test status	....	Cache	Sdram	HD	Flash	Ram	LSB

During the test, the most significant bit (bit 30) stays at "1".

NODE It returns the number of the node of the CNC within the Sercos ring.

VCHECK It returns the checksum for the software version currrently installed. It is the value appearing in the code test.

13.2.17 VARIABLES ASSOCIATED WITH THE OPERATING MODE

Read-only variables related to standard mode

OPMODE: Returns the code corresponding to the selected operating Mode.

0 = Main menu.

10 = Automatic execution.
11 = Single block execution.
12 = MDI in EXECUTION
13 = Tool inspection

20 = Theoretical path movement simulation
21 = G functions simulation
22 = G, M, S and T functions simulation
23 = Simulation with movement on main plane
24 = Simulation with rapid movement

30 = Normal editing
31 = User editing
32 = TEACH-IN editing
33 = Interactive editor
34 = Profile editor

40 = Movement in continuous JOG
41 = Movement in incremental JOG
42 = Movement with electronic handwheel
43 = HOME search in JOG
44 = Position preset in JOG
45 = Tool calibration
46 = MDI in JOG
47 = User JOG operation

50 = Zero offset table
51 = Tool Offset table
52 = Tool table
53 = Tool magazine table
54 = Global parameter table
55 = Local parameter table

60 = Utilities

70 = DNC status
71 = CNC status

80 = Editing PLC files
81 = Compiling PLC program
82 = PLC monitoring
83 = Active PLC messages
84 = Active PLC pages
85 = Save PLC program
86 = Restore PLC program
87 = “PLC resources in use” mode
88 = PLC statistics

90 = Graphic Editor

100 = General machine parameter table
101 = Axis machine parameter tables
102 = Spindle machine parameter tables
103 = Serial port machine parameter tables
104 = PLC machine parameter table
105 = M function table
106 = Leadscrew and cross compensation table
107 = Machine parameter table for Ethernet

110 = Diagnosis: configuration
111 = Diagnosis: hardware test
112 = Diagnosis: RAM memory test
113 = Diagnosis: FLASH memory test
114 = User diagnosis
115 = Hard Disk diagnosis (HD)
116 = Circle geometry (ballbar) test

Read-only variables related to the Conversational mode (MC, MCO) and Configurable M (SHIFT-ESC).

In this operating modes, it is recommended to use variables: OPMODA, OPMODB and OPMODC. The OPMODE variable is generic and contains values different from those of the standard mode.

OPMODE Returns the code for the selected operating mode.

0 = CNC starting up
10 = In execution mode, it is executing or waiting for the CYCLE START key (cycle start key icon on top)
21 = In graphic simulation mode
30 = Editing a cycle
40 = In jog mode (standard screen).
45 = In tool calibration mode
60 = Managing parts. PPROG mode

OPMODA Indicates the operating mode currently selected when working with the main channel.

Use the OPMODE variable to know at any time the selected operating mode (main channel, user channel, PLC channel).

This information is given at the least significant bits with a "1" when active and with a "0" when not active or when it is not available in the current version.

bit 0 Program in execution.
bit 1 Program in simulation.
bit 2 Block in execution via MDI, JOG
bit 3 Repositioning in progress.
bit 4 Program interrupted, by CYCLE STOP
bit 5 MDI, JOG Block interrupted
bit 6 Repositioning interrupted
bit 7 In tool inspection
bit 8 Block in execution via CNCEX1

bit 9 Block via CNCEX1 interrupted
bit 10 CNC ready to accept JOG movements: jog, handwheel, teach-in, inspection.
bit 11 CNC ready to receive the CYCLE START command: execution, simulation and MDI modes.
bit 12 The CNC is not ready to execute anything involving axis or spindle movement.

OPMODB Indicates the type of simulation currently selected. This information is given at the least significant bits with a "1" indicating the currently selected one.

bit 0 Theoretical path
bit 1 G functions
bit 2 G M S T functions
bit 3 Main plane
bit 4 Rapid
bit 5 Rapid (S=0).

OPMODC Indicates the axes selected by Handwheel. This information is given at the least significant bits indicating with a "1" the one currently selected.

bit 8	bit 7	bit 6	bit 5	bit 4	bit 3	bit	bit 1	bit 0
		Axis 7	Axis 6	Axis 5	Axis 4	Axis 3	Axis 2	Axis 1

The axis number corresponds to the order it is programmed.
Example: If the CNC controls the X, Y, Z, U, B, C axes, Axis 1 will be the X axis, Axis 2= Y, Axis 3=Z, Axis 4= U, Axis 5= B, Axis 6= C.

13.2.18 OTHER VARIABLES

NBTOOL Indicates the tool number being managed.

Example: There is a manual tool changer. Tool T1 is currently selected and the operator requests tool T5.

The subroutine associated with the tools may contain the following instructions:

(P103 = NBTOOL)
(MSG “SELECT T?P103 AND PRESS CYCLE START”)

Instruction (P103 = NBTOOL) assigns the number of the tool currently being managed to parameter P103. Therefore, P103=5

The message displayed by the CNC will be “”SELECT T5 AND PRESS CYCLE START”**.**

PRGN: Returns the program number being executed. Should none be selected, a value of -1 is returned.

BLKN: Returns the label number of the last block executed.

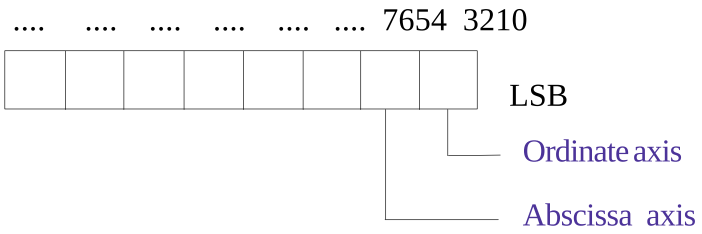
GSn: Returns the status of the G function indicated (n). 1 if it is active and 0 if not.

(P120=GS17); assigns the value 1 to parameter P120 if the G17 function is active and 0 if not.

MSn: Returns the status of the M function indicated (n). 1 if it is active and 0 if not.

This variable provides the status of M00, M01, M02, M03, M04, M05, M06, M08, M09, M19, M30, M41, M42, M43, M44 and M45 functions.

PLANE: Returns data on the abscissa axis (bits 4 to 7) and the ordinate axis (bits 0 to 3) of the active plane in 32 bits and in binary.



The axes are coded in 4 bits and indicate the axis number (from 1 to 6) according to the programming order.

Example: If the CNC controls the X,Y,Z,U,B,C axes and is selected in the ZX plane (G18).

(P122 = PLANE) assigns value $31 to parameter P122.

0000	0000	0000	0000	0000	0000	0011	0001

LSB

LONGAX: Returns the number (1 to 6) according to the programming order corresponding to the longitudinal axis. This will be the one selected with the G15 function and, by default, the axis perpendicular to the active plane, if this is XY, ZX or YZ.

Example: If the CNC controls the X, Y, Z, U, B,C axes and the U axis is selected.

(P122 = LONGAX) assigns the value 4 to parameter 122.

MIRROR Returns in the least significant bits in a group of 32 bits, the status of the mirror image of each axis, 1 in the case of being active and 0 if not.

bit 8	bit 7	bit 6	bit 5	bit 4	bit 3	bit	bit 1	bit 0
		Axis 7	Axis 6	Axis 5	Axis 4	Axis 3	Axis 2	Axis 1

The name of the axis corresponds to the number according to their programming order.

Example: If the CNC controls axes X, Y, Z, U, B, C
Axis 1=X, Axis2=Y, Axis3=Z, Axis4=U, Axis5=B, Axis6=C.

SCALE: Returns the general scaling factor applied.

SCALE (X-C): Returns the specific scaling factor of the axis indicated (X-C).

ORGROT: Returns the turning angle of the coordinate system selected with the G73 function. Its value is given in degrees. Max. ±99999.9999º

ROTPF: Returns the abscissa value of the rotation center with respect to the cartesian coordinate origin. It is given in the active units:

If G70, in inches. Max. ±3937.00787
If G71, in millimeters. Max. ±99999.9999

ROTPS: Returns the ordinate value of the rotation center with respect to the cartesian coordinate origin. It is given in the active units:

If G70, in inches. Max. ±3937.00787
If G71, in millimeters. Max. ±99999.9999

PRBST: Returns the status of the probe.

0 = The probe is **not** touching the part.
1 = The probe is touching the part.

CLOCK: Returns in seconds the time indicated by the system clock. Possible values 0...4294967295

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

TIME: Returns the time in hours-minutes-seconds format.

(P150=TIME); assigns hh-mm-ss to P150. For example if the time is 18h 22m 34 sec., P150 will contain 182234.

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

DATE: Returns the date in year-month-day format.

(P151=DATE); assigns year-month-day to P151. For example if the date is April 25th 1992, P151 will contain 920425.

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

CYTIME: Returns in hundredths of a second the time it has taken to make the part. Possible values 0...4294967295

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

FIRST: Indicates whether it is the first time that a program has been run. It returns a value of 1 if it is the first time and 0 for the remainder of times.

A first-time execution is considered as being one made:

After turning on the CNC.
After pressing the "Shift-Reset" keys.
Every time a new program is selected.

ANAIn: Returns in volts and in ±1.4 format (values ±5 Volts), the status of the analog input indicated (n), it being possible to select one among eight (1...8) analog inputs.

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

AXICOM Returns in the 3 least significant bytes the axis pairs toggled with function G28.

		Pair 3		Pair 2		Pair 1	
		Axis 2	Axis 1	Axis 2	Axis 1	Axis 2	Axis 1

The axes are coded in 4 bits and indicate the axis number (1 through 7) according to the order they are programmed.

If the CNC controls the X, Y, Z, B, C axes and G28 BC has been programmed, the AXICOM variable will show the following information:

		Pair 3		Pair 2		Pair 1	
						C	B
0000	0000	0000	0000	0000	0000	0101	0100

TANGAN Variable associated with the tangential control (G45). It indicates the programmed angular position.

Read-write variables

TIMER: This variable allows time, in seconds, indicated by the clock enabled by the PLC to be read or modified. Possible values 0...4294967295

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

PARTC: The CNC has a part counter whose count increases in all modes except simulation every time M30 or M02 is executed and this variable allows it value to be read or modified, which will be given by a number between 0 and 4294967295

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

KEY: Returns the code of the last key accepted.

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

KEYSRC: This variable allows the origin of keys to be read or modified, possible values being:

0 = Keyboard
1 = PLC
2 = DNC

The CNC only allows modification of this variable if this is at 0.

ANAOn: This variable allows the required analog output (n) to be modified. The value assigned will be expressed in volts and in the ±2.4 format (±10 Volts).

The analog outputs which are free among the eight (1..8) available at the CNC may be modified, the corresponding error being displayed if an attempt is made to write in one occupied.

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

13.3 CONSTANTS

Constants are defined as being all those fixed values which cannot be altered by a program. The following are considered as constants:

- Numbers expressed in the decimal system.
- Hexadecimal numbers.
- **PI** (¶) constant.
- Read-only tables and variables as their value cannot be altered with a program.

13.4 OPERATORS

An operator is a symbol which indicates mathematical or logic manipulations which must bemade. The CNC has arithmetic, relational, logic, binary, trigonometric operators and special operators.

Arithmetic operators

+	: add.	P1=3 + 4	—> P1=7
-	: subtraction, also to indicate	P2=5 **-** 2	—> P2=3
	a negative number	P3=-(2*****3)	—> P3=-6
*****	: multiplication	P4=2*****3	—> P4=6
/	: division	P5=9/2	—> P5=4.5
MOD	: module (remainder of a division)	P6=7 **MOD** 4	—> P6=3
EXP	: exponential	P7=2 **EXP** 3	—> P7=8

Relational operators

EQ : equal
NE : different
GT : greater than
GE : greater than or equal to
LT : less than
LE : less than or equal to

Logic or binary operators

NOT, OR, AND, XOR: act as logic operators between conditions and as binary operators between variables and constants.

IF (FIRST **AND** GS1 EQ 1) GOTO N100
P5 = (P1 **AND** (**NOT** P2 **OR** P3))

Trigonometric functions

SIN	: sine	P1=**SIN** 30	—> P1=0.5
COS	: cosine	P2=**COS** 30	—> P2=0.8660
TAN	: tangent	P3=**TAN** 30	—> P3=0.5773
ASIN	: arc sine	P4=**ASIN** 1	—> P4=90
ACOS	: arc cosine	P5=**ACOS** 1	—> P5=0
ATAN	: arc tangent	P6=**ATAN** 1	—> P6=45
ARG	: ARG (x,y) arc tangent y/x	P7=**ARG**(-1,-2)	—> P7=243.4349

There are two functions for calculating the arc tangent: ATAN which returns the result between ±90° and ARG given between 0 and 360°.

Other functions

ABS	: absolute value	P1=**ABS** -8	—> P1=8
LOG	: decimal logarithm	P2=**LOG** 100	—> P2=2
SQRT	: square root	P3=**SQRT** 16	—> P3=4
ROUND	: rounding up a number	P4=**ROUND** 5.83	—> P4=6
FIX	: integer	P5=**FIX** 5.423	—> P5=5
FUP	: if integer takes integer	P6=**FUP** 7	—> P6=7
	if not, takes entire part + 1	P6=**FUP** 5.423	—> P6=6
BCD	: converts given number to BCD	P7=**BCD** 234	—> P7=564

0010	0011	0100

BIN	: converts given number to binary	P8=**BIN** $AB	—> P8=171

1010	1011

Conversions to binary and BCD are made in 32 bits, it being possible to represent the number 156 in the following formats:

Decimal	156
Hexadecimal	9C
Binary	0000 0000 0000 0000 0000 0000 1001 1100
BCD	0000 0000 0000 0000 0000 0001 0101 0110

13.5 EXPRESSIONS

An expression is any valid combination between operators, constants and variables.

All expressions must be placed between brackets, but if the expression is reduced to an integer, the brackets can be removed.

13.5.1 ARITHMETIC EXPRESSIONS

These are formed by combining functions and arithmetic, binary and trigonometric operators with the constants and variables of the language.

The way to operate with these expressions is established by operator priorities and their associativity:

Priority from highest to lowest	**Associativity**
NOT, functions, - (negative)	from right to left
EXP, MOD	from left to right
*,/	from left to right
+,-(add, subtract)	from left to right
relational operators	from left to right
AND, XOR	from left to right
OR	from left to right

It is advisable to use brackets to clarify the order in which the evaluation of the expression is done.

```
(P3 =  P4/P5 - P6*P7 - P8/P9)
(P3 = (P4/P5)-(P6*P7)-(P8/P9))
```

The use of repetitive or additional brackets will not produce errors nor will they slow down execution.

In functions, brackets must be used except when these are applied to a numerical constant, in which case they are optional.

```
(SIN 45)  (SIN (45)) both are valid and equivalent.
(SIN 10+5)                the same as ((SIN 10)+5).
```

Expressions can be used also to reference parameters and tables:

```
(P100  = P9)
(P100  = P(P7))
(P100  = P(P8 + SIN (P8 *20)))
(P100  = ORGX 55)
(P100  = ORGX (12+P9))
(PLCM5008  =  PLCM5008 OR 1); selects Single Block execution (M5008=1)
(PLCM5010  =  PLCM5010 AND $FFFFFFFE); Frees feedrate Override
                                          (M5010=0)
```

13.5.2 RELATIONAL EXPRESSIONS

These are arithmetic expressions joined by relational operators

(IF **(P8 EQ 12.8)**	;Analyzes if the value of P8 is equal to 12.8
(IF **(ABS(SIN(P24)) GT SPEED)** ...	;Analyzes if the sine is greater than the spindle speed.
(IF **(CLOCK LT(P9*10.99))**	;Analyzes if the clock count is less than (P9*10.99)

At the same time these conditions can be joined by means of logic operators.

(IF **((P8EQ12.8) OR (ABS(SIN(P24)) GT SPEED)) AND (CLOCK LT (P9*10.99))** ....

The result of these expressions is either true or false.

14. PROGRAM CONTROL STATEMENTS

The control statements available to high-level programming can be grouped as follows:

* Programming statements consisting of:

  Assignment statements
  Display statements
  Enable-disable statements
  Flow control statements
  Subroutine statements
  Statements for generating programs
  Screen customizing statements

* Screen customizing statements

Only one statement can be programmed in each block, and no other additional information may be programmed in this block.

14.1 ASSIGNMENT STATEMENTS

This is the simplest type of statement and can be defined as:

(target=arithmetic expression)

A local or global parameter or a read-write variable may be selected as target. The arithmetic expression may be as complex as required or a simple numerical constant.

(P102 = FZLOY)
(ORGY 55 = (ORGY 54 + P100))

In the specific case of designating a local parameter using its name (A instead of P0, for example) and the arithmetic expression being a numerical constant, the statement can be abbreviated as follows:

(P0=13.7) ==> (A=13.7) ==> (A13.7)

Within a single block, up to 26 assignments can be made to different targets, a single assignment being interpreted as the set of assignments made to the same target.

(P1=P1+P2, P1=P1+P3,P1=P*P4,P1=P1/p5) is the same as (P1=(P1+P2+P3)*P4/P5).

The different assignments which are made in the same block will be separated by commas ",".

14.2 DISPLAY STATEMENTS

(ERROR integer, "error text"

This statement stops the execution of the program and displays the indicated error, it being possible to select this error in the following ways:

(ERROR integer). This will display the error number indicated and the text associated to this number according to the CNC error code (should there be one).

(ERROR integer "error text"). This will display the number and the error text indicated, it being necessary to write the text between quote marks "".

(ERROR "error text"). This will display the error text only.

The error number may be defined by means of a numerical constant or an arithmetic parameter. When using a local parameter, its numeric format must be used (P0 thru P25 instead of A thru Z).

Programming Examples:

```
(ERROR 5)
(ERROR P100)
(ERROR "Operator error")
(ERROR 3, "Operator error")
(ERROR P120, "Operator error)
```

(MSG "message")

This statement will display the message indicated between quote marks.

The CNC screen is provided with an area for displaying DNC or user program messages, and always displays the last message received irrespective of where it has come from.

Example:

```
(MSG "Check tool")
```

(DGWZ expression 1, expression 2, expression 3, expression 4, expression 5, expression 6)

The DGWZ instruction (Define Graphic Work Zone) defines the graphics area.

Each expression forming the instruction syntax correspond to one of the limits and they must be defined in millimeters or inches.

expression 1	X minimum
expression 2	X maximum
expression 3	Y minimum
expression 4	Y maximum
expression 5	Z minimum
expression 6	Z maximum

14.3 ENABLING-DISABLING STATEMENTS

(ESBLK and DSBLK)

After executing the mnemonic **ESBLK**, the CNC executes all the blocks which come after as if it were dealing with a single block.

This single block treatment is kept active until it is cancelled by executing the mnemonic **DSBLK**.

In this way, should the program be executed in the SINGLE BLOCK operating mode, the group of blocks which are found between the mnemonics ESBLK and DSBLK will be executed in a continuous cycle, i.e., execution will not be stopped at the end of a block but will continue by executing the following one.

Example:

```
G01 X10 Y10 F800 T1 D1
(ESBLK)                        ; Start of single block
G02 X20 Y20 I20 J-10
G01 X40 Y20
G01 X40 Y40 F10000
G01 X20 Y40 F8000
(DSBLK)                        ; Cancellation of single block
G01 X10 Y10
M30
```

(ESTOP and DSTOP)

After executing the mnemonic **DSTOP**, the CNC enables the Stop key, as well as the Stop signal from the PLC.

It will remain disabled until it is enabled once again by means of the mnemonic **ESTOP**.

(EFHOLD and DFHOLD)

After executing the mnemonic **DFHOLD**, the CNC enables the Feed-Hold input from the PLC.

It will remain disabled until it is enabled once again by means of the mnemonic **EFHOLD**.

14.4 FLOW CONTROL STATEMENTS

The **GOTO** and **RPT** instructions cannot be used in programs that are executed from a PC connected through the serial lines.

(GOTO N(expression))

The mnemonic **GOTO** causes a jump within the same program, to the block defined by the label N(expression).

The execution of the program will continue after the jump, from the indicated block.

The jump label can be addressed by means of a number or by any expression which results in a number.

Example:

```
G00  X0   Y0   Z0  T2 D4
     X10
(GOTO N22)                      ; Jump statement
     X15 Y20                    ; Is not executed
     Y22 Z50                    ; Is not executed
N22  G01  X30  Y40  Z40 F10000  ; Continues execution in this block
     G02 X20  Y40  I-5  J-5
     ...........
     ...........
```

(RPT N(expression), N(expression))

The mnemonic **RP**T executes, within the same program, the part of the program which exists between the blocks defined by means of the labels **N(expression**).

Both labels can be indicated by means of a number or by any expression which results in a number.

The part of the program selected by means of the two labels must belong to the same program, by first defining the initial block and then the final block.

The execution of the program will continue in the block following the one in which the mnemonic RPT was programmed, once the selected part of the program has been executed.

Example:

```
N10  G00   X10
     Z20
     G01   X5
     G00   Z0
N20        X0
N30  (RPT  N10, N20) N3
N40  G01   X20
     M30
```

When reaching block N30, the program will execute section N10-N20 three times. Once this has been completed, the program will continue execution in block N40.

(IF condition <action1> ELSE <action2>)

This statement analyzes the given condition which must be a relational expression. If the condition is true (result equal to 1), **<action1**> will be executed, otherwise (result equal to 0) **<action2**> will be executed.

Example:

(IF(P8 EQ 12.8) CALL 3 ELSE PCALL 5, A2, B5, D8)

If P8 = 12.8 executes the mnemonic (CALL3)
If P8 <> 12.8 executes the mnemonic (PCALL 5, A2, B5, D8)

The statement can lack the ELSE part, i.e., it will be enough to program IF condition <action1>.

Example:

(IF(P8 EQ 12.8)CALL 3)

Both <action1> and <action2> can be expressions or statements, except for mnemonics IF and SUB.

Due to the fact that in a high level block local parameters can be named by means of letters, expressions of this type can be obtained:

(IF (E EQ 10)M10)

If the condition of parameter P5 (E) having a value of 10 is met, the miscellaneous function M10 will not be executed, since a high level block cannot have ISO code commands. In this case M10 represents the assignment of value 10 to parameter P12, i.e., one can program either:

(IF(E EQ 10)M10) or (IF(P5 EQ 10) P12=10)

14.5 SUBROUTINE STATEMENTS

A subroutine is a part of a program which, being properly identified, can be called from any position of a program to be executed.

A subroutine can be kept in the memory of the CNC as an independent part of a program and be called one or several times, from different positions of a program or different programs.

Only subroutines stored in the CNC's RAM memory can be executed. Therefore, to execute a subroutine stored in the Memkey Card, HD or in a PC connected through the serial lines, it must be copied first into the CNC's RAM memory.

If the subroutine is too large to be copied into RAM, it must be converted into a program and then the EXEC instruction must be used as described in section 14.6

(SUB integer)

The mnemonic **SUB** defines the set of program blocks which are programmed after this block as a subroutine by identifying this subroutine with an integer, between 0 and 9999, which is specified after it:

There can not be two subroutines with the same identification number in the CNC memory, even when they belong to different programs.

(RET)

The mnemonic **RET** indicates that the subroutine which was defined by the mnemonic SUB, finishes in this block.

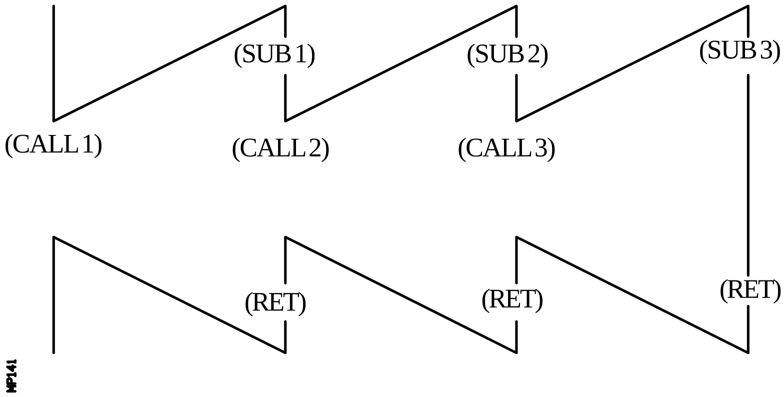
Example:

```
(SUB 12)                    ; Definition of subroutine 12
G91  G01  XP0 F5000
     YP1
     X-P0
     Y-P1
(RET)                       ; End of subroutine
```

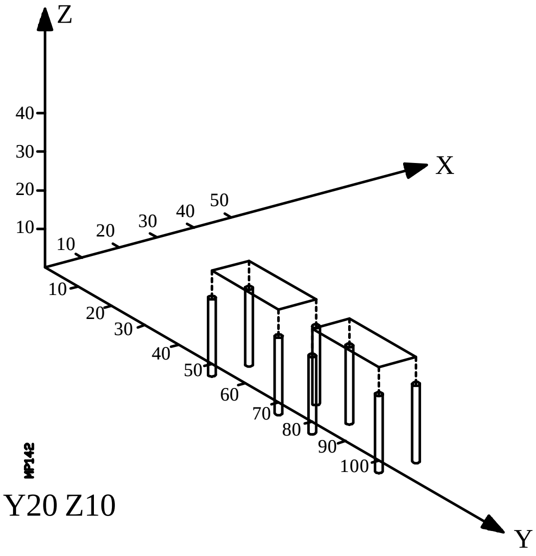
(CALL (expression))

The mnemonic **CALL** makes a call to the subroutine indicated by means of a number or by means of any expression which results in a number.

As a subroutine may be called from a main program, or a subroutine, from this subroutine to a second one, from the second to a third, etc..., the CNC limits these calls to a maximum of 15 nesting levels, it being possible to repeat each of the levels 9999 times.



Example




```
G90 G00 X30 Y20 Z10
(CALL 10)
G90 G00 X60 Y20 Z10
(CALL 10)
M30

(SUB 10)
G91 G01 X20 F5000
(CALL 11)                              ;Drilling and threading
G91 G01 Y10
(CALL 11)                              ;Drilling and threading
G91 G01 X-20
(CALL 11)                              ;Drilling and threading
G91 G01 Y-10
(CALL 11)                              ;Drilling and threading
RET

(SUB 11)
G81 G98 G91 Z-8 I-22 F1000 S5000 T1 D1 ;Drilling canned cycle
G84 Z-8 I-22 K15 F500 S2000 T2 D2      ;Threading canned cycle
G80
(RET)
```

(PCALL (expression), (assignment statement), (assignment statement),...)

The mnemonic **PCALL** calls the subroutine indicated by means of a number or any expression which results in a number. In addition, it allows up to a maximum of 26 local parameters of this subroutine to be initialized.

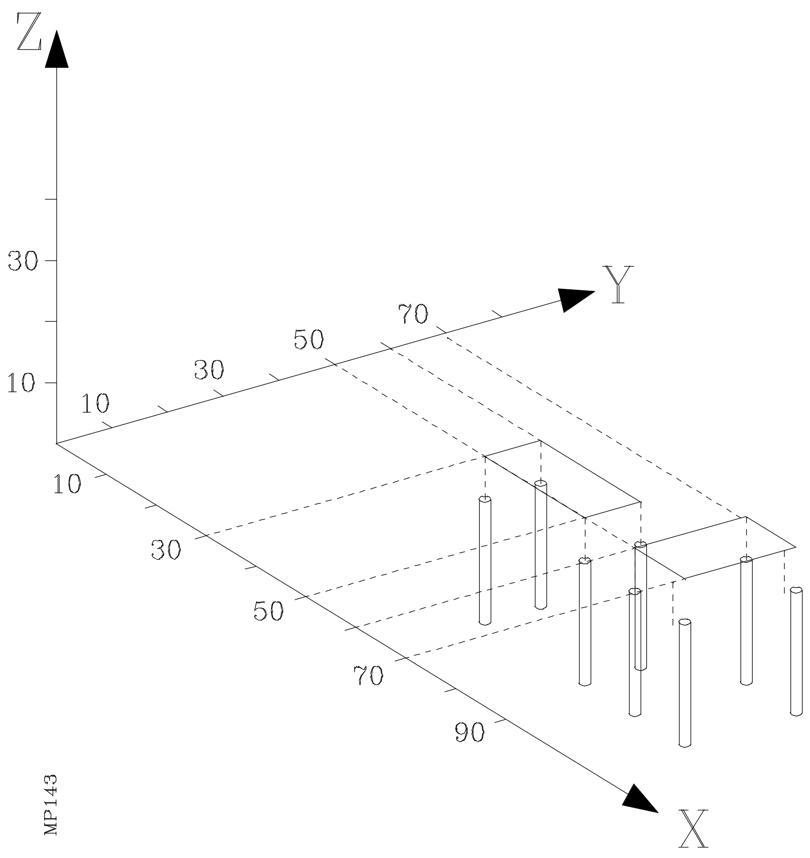
These local parameters are initialized by means of **assignment statements**.

Example: (PCALL 52, A3, B5, C4, P10=20)

In this case, in addition to generating a new subroutine nesting level, a new local parameter nesting level will be generated, there being a maximum of 6 levels of local parameter nesting, within the 15 levels of subroutine nesting.

Both the main program and each subroutine which is found on a parameter nesting level, will have 26 local parameters (P0-P25).

Example:



```
G90 G00 X30 Y50 Z0
(PCALL 10, P0=20, P1=10)                  ; or also (PCALL 10, A20, B10)
G90 G00 X60 Y50 Z0
(PCALL 10, P0=10 P1=20)                   ; or also (PCALL 10, A10 B20)
M30

(SUB 10)
G91 G01 XP0 F5000
(CALL 11)
G91 G01 YP1
(CALL 11)
G91 G01 X-P0
(CALL 11)
G91 G01 Y-P1
(CALL 11)
RET

(SUB 11)
G81 G98 G91 Z-8 I-22 F1000 S5000 T1 D1    ; Drilling canned cycle
G84 Z-8 I-22 K15 F500 S2000 T2 D2         ; Threading canned cycle
G80
(RET)
```

(MCALL (expression), (assignment statement), (assignment statement),...)

By means of the mnemonic **MCALL**, any user-defined subroutine (SUB integer) acquires the category of canned cycle.

The execution of this mnemonic is the same as the mnemonic PCALL, but the call is modal, i.e., if another block with axis movement is programmed at the end of this block, after this movement, the subroutine indicated will be executed and with the same call parameters.

If, when a modal subroutine is selected, a movement block with a number of repetitions is executed, for example X10 N3, the CNC will execute the movement only once (X10) and after the modal subroutine, as many times as the number of repetitions indicates.

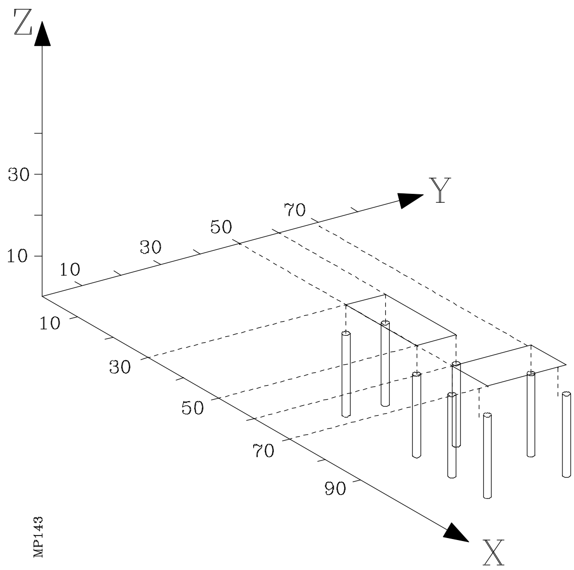
Should block repetitions be chosen, the first execution of the modal subroutine will be made with updated call parameters, but not for the remaining times, which will be executed with the values which these parameters have at that time.

If, when a subroutine is selected as modal, a block containing the MCALL mnemonic is executed, the present subroutine will lose its modal quality and the new subroutine selected will be changed to modal.

(MDOFF)

The mnemonic **MDOFF** indicates that the modal quality acquired by the subroutine with the MCALL mnemonic, finishes in this block.

The use of modal subroutines simplifies programming.



Example:

```
G90 G00 X30 Y50 Z0
(PCALL 10, P0=20, P1=10)
G90 G00 X60 Y50 Z0
(PCALL 10, P0=10 P1=20)
M30

(SUB 10)
G91 G01 XP0 F5000
(MCALL 11)
G91 G01 YP1
G91 G01 X-P0
G91 G01 Y-P1
(MDOFF)
RET)

(SUB 11)
G81 G98 G91 Z-8 I-22 F1000 S5000 T1 D1
G84 Z-8 I-22 K15 F500 S2000 T2 D2
G80
(RET)
```

(PROBE (expression), (assignment statement), (assignment statement),...)

The mnemonic **PROBE** calls the probe cycle indicated by means of a number or any **expression** which results in a number. In addition, it allows the local parameters of this subroutine to be initialized by means of **assignment statements**.

This mnemonic also generates a new level of subroutine nesting.

14.5.1 INTERRUPTION SUBROUTINE STATEMENTS

Whenever one of the general interruption logic input is activated, "INT1" (M5024), "INT2" (M5025), "INT3" (M5026) or "INT4 (M5027), the CNC temporarily interrupts the execution of the program in progress and starts executing the interruption subroutine whose number is indicated by the corresponding general parameter.

With INT1 (M5024) the one indicated by machine parameter INT1SUB (P35)
With INT2 (M5025) the one indicated by machine parameter INT2SUB (P36)
With INT3 (M5026) the one indicated by machine parameter INT3SUB (P37)
With INT4 (M5027) the one indicated by machine parameter INT4SUB (P38)

The interruption subroutines are defined like any other subroutine by using the statements: "(SUB integer)" and "(RET)".

The interruption subroutines do not change the level of the local arithmetic parameters; thus they can only contain global arithmetic parameters.

Within an interruption subroutine, it is possible to use the "(REPOS X, Y, Z, ...)" statement described next.

Once the execution of the subroutine is over, the CNC resumes the execution of the program which was interrupted.

(REPOS X, Y, Z, ...)

The REPOS statement must always be used inside an interruption subroutine and facilitates the repositioning of the machine axes to the point of interruption.

When executing this statement, the CNC moves the axes to the point where the program was interrupted.

* The axes are repositioned one at a time.
* It is not necessary to define all the axes, only those to be repositioned.
* The axes forming the main plane move together; thus, it is not required to program both axes since the CNC moves both of them with the first one. The movement is not repeated when defining the second one, it is ignored.

Example: The main plane is formed by the X and Y axes, the Z axis is the longitudinal (perpendicular) axis and the machine uses the C and W axes as auxiliary axes. It is desired to first move the C axis, then the X and Y axes and finally the Z axis..

This repositioning move may be defined in any of the following ways:

(REPOS C, X, Y, Z) **(REPOS C, X, Z)(REPOS C, Y, Z)**

If the REPOS statement is detected while executing a subroutine not activated by an interruption input, the CNC will issue the corresponding error message.

14.6 PROGRAM STATEMENTS

With this CNC, from a program in execution one can:

Execute another program .... Statement (EXEC P........)
Generate a new program .... Statement (OPEN P........)
Add blocks to an existing program.................................. Statement (WRITE........)

(EXEC P(expression), (directory)

The **EXEC P** statement executes the part-program of the indicated directory

The part-program may be defined by a number or any **expression** resulting in a number.

By default, the CNC assumes that the part-program is in the CNC's RAM memory.
If it is in another device, it must be indicated in (**directory**).

CARDA	in the "Memkey CARD"
HD	on the hard disk
DNC1	at a PC connected through serial line 1
DNC2	at a PC connected through serial line 2

(OPEN P(expression), (destination directory), A/D, "program comment")

This statement starts editing a part-program whose number will be given by any number or **expression** resulting in a number.

By default, the new part-program edited will be stored in the CNC's RAM memory.
To store it another device, it must be indicated in (**destination directory**).

CARDA	in the "Memkey CARD"
HD	on the hard disk
DNC1	at a PC connected through serial line 1
DNC2	at a PC connected through serial line 2

Parameter **A/D** is used when the program to be edited already exists.

A The CNC appends the new blocks after the ones already existing
D The CNC deletes the existing program and starts editing a new one.

A **program comment** may also be associated with it. This comment will later be displayed next to it on the program directory.

The OPEN statement allows generating a program from a program already in execution. That generated program will depend on the values assumed by the program being executed.

To edit blocks, the WRITE statement must be used as described next.

<u>Notes:</u> If the program to be edited already exists and the A/D parameters are not defined, the CNC will display an error message when executing the block.

The program opened with the OPEN statement is closed when executing an M30, or another OPEN statement and after an Emergency or Reset.

From a PC, only programs stored in the CNC'S RAM memory, in the CARD A, or in the Hard Disk module can be opened

(WRITE <block text>)

The mnemonic **WRITE** adds, after the last block of the program which began to be edited by means of the mnemonic OPEN P, the information contained in **<block text>** as a new program block.

When it is an ISO coded parametric block, all the parameters (global and local) are replaced by the numeric value they have at the time.

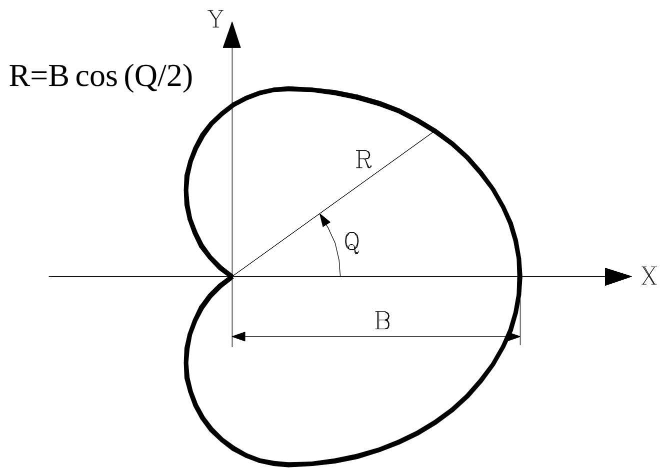
(WRITE G1 XP100 YP101 F100)	=>	G1 X10 Y20 F100

When it is a parametric block edited in high level, use the "?" character to indicate that the parameter is supposed to be replaced by the numeric value it has at the time.

(WRITE (SUB P102))	=>	(SUB P102)
(WRITE (SUB ?P102))	=>	(SUB 55)
(WRITE (ORGX54=P103))	=>	(ORGX54=P103)
(WRITE (ORGX54=?P103))	=>	(ORGX54=222)
(WRITE (PCALL P104))	=>	(PCALL P104)
(WRITE (PCALL ?P104))	=>	(PCALL 25)

If the mnemonic WRITE is programmed without having programmed the mnemonic OPEN previously, the CNC will display the corresponding error, except when editing a user customized program, in which case a new block is added to the program being edited.

Example of the creation of a program which contains several points of a cardioid whose formula is:



Subroutine number 2 is used, its parameters having the following meaning:

A or P0	Value of angle Q.
B or P1	Value of B
C or P2	Angular increment for calculation
D or P3	Axis feedrate

A way to use this example could be:

```
G00 X0 Y0
G93
(PCALL 2, A0, B30, C5, D500)
M30
```

Program generation subroutine:

```
       (SUB 2)
       (OPEN P12345)                  ; Starts editing of program P12345
       (WRITE FP3)                    ; Selects machining feedrate
N100   (P10=P1*(ABS(COS(P0/2)))       ; Calculates R
       (WRITE G01 G05 RP10 QP0)       ; Movement block
       (P0=P0+P2)                     ; New angle
       (IF (P0 LT 365) GOTO N100)     ; If angle less than 365°, calculates new
                                      ; point
       (WRITE M30)                    ; End of program block
       (RET)                          ; End of subroutine
```

14.7 SCREEN CUSTOMIZING STATEMENTS (GRAPHIC EDITOR)

Customizing statements may be used only when customizing programs made by the user.

These customizing programs must be stored in the CNC'S RAM memory and they may utilize the "Programming Statements" and they will be executed in the special channel designed for this use. The program selected in each case will be indicated in the following general machine parameters.

In "USERDPLY" the program to be executed in the Execution Mode will be indicated.

In "USEREDIT" the program to be executed in the Editing Mode will be indicated.

In "USERMAN" the program to be executed in the Manual (JOG) Mode will be indicated.

In "USERDIAG" the program to be executed in the Diagnosis Mode will be indicated.

The customizing programs may have up to five nesting levels besides their current one. Also, the customizing statements do not allow local parameters, nevertheless all global parameters may be used to define them.

(PAGE (expression))

The mnemonic **PAG**E displays the page number indicated by means of a number or by means of any **expressio**n which results in a number.

User-defined pages will be from page 0 to page 255 and will be defined from the CNC keyboard in the Grahic Editor mode and as indicated in the Operating Manual.

System pages will be defined by a number greater than 1000. See the corresponding appendix.

(SYMBOL (expression 1), (expression 2), (expression 3))

The mnemonic **SYMBO**L displays the symbol whose number is indicated by means of the value of **expression 1** once this has been evaluated.

Its position on screen is also defined by **expression 2** (column) and by **expression 3** (row).

Expression 1, expression 2 and expression 3 may contain a number or any expression which results in a number.

The CNC allows to display any user-defined symbol (0-255) defined at the CNC keyboard in the Graphic Editor mode such as is indicated in the Operating Manual.

In order to position it within the display area its pixels must be defined, 0-639 for columns (expression 2) and 0-335 for rows (expression 3).

(IB (expression) = INPUT "text", format))

The CNC has 26 data entry variables (IBO-1B25)

The **I**B mnemonic displays the text indicated in the data input window and stores the data input by the user in the entry variable indicated by means of a number or by means of any **expression which results in a number**.

The wait for data entry will only occur when programming the format of the requested data. This **format** may have a sign, integer part and decimal part.

If it bears the “minus” sign, it will allow positive and negative values, and if it does not have a sign, it will only allow positive values.

The integer part indicates the maximum number of digits (0-6) desired to the left of the decimal point.

The decimal part indicates the maximum number of digits (0-5) desired to the right of the decimal point.

If the numerical format is not programmed; for example: (IB1 =INPUT "text"), the mnemonic will only display the indicated text without waiting for the data to be entered.

(ODW (expression 1), (expression 2), (expression 3))

The mnemonic **ODW** defines and draws a white window on the screen with fixed dimensions (1 row and 14 columns).

Each mnemonic has an associated number which is indicated by the value of **expression 1** once this has been evaluated.

Likewise, its position on screen is defined by **expression 2** (row) and by **expression 3** (column).

Expression 1, expression 2 and expression 3 may contain a number or any expression which results in a number.

The CNC allows 26 windows (0-25) to be defined and their positioning within the display area, providing 21 rows (0-20) and 80 columns (0-79).

(DW(expression 1) = (expression 2), DW (expression 3) = (expression 4),...)

The mnemonic **DW** displays in the window indicated by the value for expression 1, expression 3, .. once they have been evaluated, the numerical data indicated by expression 2, expression 4, ...

Expression 1, expression 2, expression 3, ... may contain a number or any expression which may result in a number.

The following example shows a dynamic variable display:

```
    (ODW 1,6,33)                  ; Defines data window 1
    (ODW 2,14,33)                 ; Defines data window 2
N10 (DW1=DATE,DW2=TIME)           ; Displays the date in window 1 and the time in 2
    (GOTO N10)
```

The CNC allows displaying the data in decimal, hexadecimal and binary format. The following instructions are available:

(DW1 = 100)	Decimal format. Value "100" displayed in window 1.
(DWH2=100)	Hexadecimal format. Value "64" displayed in window 2.
(DWB3=100)	Binary format. Value "01100100" displayed in window 3.

When using the binary format, the display is limited to 8 digits in such a way that a value of "11111111" will be displayed for values greater than 255 and the value of "10000000" for values more negative than -127.

Besides, the CNC allows the number stored in one of the 26 data input variables (IB0-IB25) to be displayed in the requested window.

The following example shows a request and later display of axis feedrate.

```
(ODW3,4,60)                   ; Defines data window 3
(IB1=INPUT"Axis feed:",5.4)   ; Axis feedrate request
(DW3=IB1)                     ; Displays feedrate in window 3
```

(SK(expression 1) = “text1” (expression 2) = “text 2”, ...)

The mnemonic **SK** defines and displays the new softkey menu indicated.

Each of the **expressions** will indicate the softkey number which it is required to modify (1-7, starting from the left) and the **texts** which it is required to write in them.

Expression 1, expression 2, expression 3, ... may contain a number or any expression which may result in a number.

Each text will allow a maximum of 20 characters which will be shown on two lines of 10 characters each. If the text selected has less than 10 characters, the CNC will center it on the top line, but if it has more than 10 characters the programmer will center it.

Examples:

Statement	Softkey 1	Softkey 2
(SK 1=”HELP”, SK 2=”MAXIMUM COORDINATE”)	HELP	MAXIMUM CO ORDINATE
(SK 1=”FEEDRATE”,SK 2=”_MAXIMUM__COORDINATE”)	FEEDRATE	MAXIMUM COORDINATE

Warning:

If while a standard CNC softkey menu is active, one or more softkeys are selected via high level language statement: "**SK**", the CNC will clear all existing softkeys and it will only show the selected ones.

If while a user softkey menu is active, one or more softkeys are selected via high level language statement "**SK**", the CNC will only replace the selected softkeys leaving the others intact.

(WKEY)

The mnemonic **WKEY** stops execution of the program until the key is pressed.

The pressed key will be recorded in the KEY variable.

Example

```
....
....
(WKEY)                          ; Wait for key
8IF KEY EQ $FC00 GOTO N1000     ; If key F1 has been pressed, continue in
                                  N1000
....
....
```

(WBUF “text”, (expression))

The **WBUF** statement can only be used when editing a program in the user channel.

This mnemonic may be programmed in two ways:

(WBUF “text”, (expression))

This statement adds the text and value of the expression once this has been evaluated, to the block which is being edited and within the data input window.

(Expression) may contain a number or any expression which results in a number.

It will be optional to program the expression, but it will be required to define the text. If no text is required, “” must be programmed.

Examples for P100=10 (WBUF "X", P100) => X10
(WBUF "XP100") => XP100

(WBUF)

Enters into memory, adding to the program being edited and after the cursor position, the block being edited by means of (WBUF "text", (expression)). It also clears the editing buffer in order to edit a new block.

This allows the user to edit a complete program without having to quit the user editing mode after each block and press ENTER to "enter" it into memory.

Example:

(WBUF”(PCALL 25,”)	; Adds “(PCALL 25,” to the block being edited
(IB1=INPUT “Parameter A:”,-5.4)	; Request of Parameter A
(WBUF “A=”,IB1)	; Adds “A=(value entered) to the block being edited.
(IB2=INPUT”Parameter B:”,-5.4)	; Request of Parameter B
(WBUF”,B=”,IB2	; Adds “B=(value entered)” to the block being edited
(WBUF”)”)	; Adds “)” to the block being edited
(WBUF)	; Enters the edited block into memory

\------------
\------------

After executing this program the block being edited contains:

(PCALL 25, A=23.5, B=-2.25)

(SYSTEM)

The mnemonic **SYSTEM** stops execution of the user customized program and returns to the corresponding standard menu of the CNC.

Customizing program example

The following customizing program must be selected as user program associated to the Editing Mode.

After selecting the Editing Mode and pressing the USER softkey, this program starts executing and it allows assisted editing of 2 user cycles. This editing process is carried out a cycle at a time and as often as desired.

```
; Displays the initial editing page (screen)
N0   (PAGE 10)

; Sets the softkeys to access the various modes and requests a choice
     (SK 1=”CYCLE 1",SK 2=”CYCLE 2",SK 7=”EXIT”)
N5   (WKEY)                                   ;Request a key
     (IF KEY EQ $FC00 GOTO N10)               ;Cycle 1
     (IF KEY EQ $FC01 GOTO N20)               ;Cycle 2
     (IF KEY EQ $FC06 SYSTEM ELSE GOTO N5)    ;Quit or request a key

; CYCLE 1
; Displays page 11 and defines 2 data entry windows
N10  (PAGE 11)
     (ODW 1,10,60)
     (ODW 2,15,60)

;Editing
     (WBUF “(PCALL 1,”)        ; Adds (PCALL 1,  to the block being edited

     (IB 1=INPUT “X:”,-6.5)    ; Requests the value of X
     (DW 1=IB1)                ; Data window 1 shows the entered value
     (WBUF “X”,IB1)            ; Adds X (entered value) to the block being edited

     (WBUF “,”)                ; Adds ,  to the block being edited

     (IB 2=INPUT “Y:”,-6.5)    ; Requests the value of Y
     (DW 2=IB2)                ; Data window 2 shows the entered value
     (WBUF “Y”,IB2)            ; Adds Y (entered value) to the block being edited

     (WBUF “)”)                ; Adds )  to the block being edited

     (WBUF)                    ; Enters the edited block into memory. For example: (PCALL 1, X2, Y3)

     (GOTO N0)
```

;(This sample program continues on next page)

; __CYCLE 2__

; Displays page 12 and defines 3 data entry windows

```
N20  (PAGE 12)
     (ODW 1,10,60)
     (ODW 2,13,60)
     (ODW 3,16,60)
```

;Editing

(WBUF "(PCALL 2,")	*; Adds* **(PCALL 2,** *to the block being edited*
(IB 1=INPUT "A:",-6.5)	*; Requests the value of A*
(DW 1=IB1)	*; Data window 1 shows the entered value*
(WBUF "A",IB1)	*; Adds* **A (entered value)** *to the block being edited*
(WBUF ",")	*; Adds* **,** *to the block being edited*
(IB 2=INPUT "B:",-6.5)	*; Requests the value of B*
(DW 2=IB2)	*; Data window 2 shows the entered value*
(WBUF "B",IB2)	*; Adds* **B (entered value)** *to the block being edited*
(WBUF ",")	*; Adds* **,** *to the block being edited*
(IB 3=INPUT "C:",-6.5)	*; Requests the value of C*
(DW 3=IB3)	*; Data window 3 shows the entered value*
(WBUF "C",IB3)	*; Adds* **C (entered value)** *to the block being edited*
(WBUF ")")	*; Adds* **)** *to the block being edited*
(WBUF)	;*Enters the edited block into memory*. Example: (PCALL 2, A3, B1, C3)
(GOTO N0)	

APPENDIX

A

ISO CODE PROGRAMMING

Function	M	D	V	Meaning	Section
G00	*	?	*	Rapid travel	6.1
G01	*	?	*	Linear interpolation	6.2
G02	*		*	Clockwise (helical) circular interpolation	6.3
G03	*		*	Counter-clockwise (helical) circular interpolation	6.3
G04				Dwell/block preparation stop	7.1, 7.2
G05	*	?	*	Round corner	7.3.1
G06			*	Absolute arc center coordinates	6.4
G07	*	?		Square corner	7.3.2
G08			*	Arc tangent to previous path	6.5
G09			*	Arc defined by three points	6.6
G10	*	*		Mirror image cancellation	7.5
G11	*		*	Mirror image on X axis	7.5
G12	*		*	Mirror image on Y axis	7.5
G13	*		*	Mirror image on Z axis	7.5
G14	*		*	Mirror image in the programmed directions	7.5
G15	*		*	Longitudinal axis selection	8.2
G16	*		*	Selection of main plane in two directions	3.2
G17	*	?	*	Main plane X-Y and longitudinal Z	3.2
G18	*	?	*	Main plane Z-X and longitudinal Y	3.2
G19	*		*	Main plane Y-Z and longitudinal X	3.2
G20				Definition of lower work zone limits	3.7.1
G21				Definition of upper work zone limits	3.7.1
G22			*	Activate/cancel work zones	3.7.2
G28	*		*	Second spindle selection	5.4
G29	*	*		Main spindle selection	5.4
G28-G29			*	Axes toggle	7.9
G30	*		*	Spindle synchronization	5.5
G32	*		*	Feedrate as an inverted function of time.	6.15
G33	*		*	Threadcutting	6.12
G34				Variable pitch thread	6.13
G36			*	Automatic radius blend	6.10
G37			*	Tangential entry	6.8
G38			*	Tangential exit	6.9
G39			*	Automatic chamfer blend	6.11
G40	*	*		Cancellation of tool radius compensation	8.1
G41	*		*	Right-hand tool radius compensation	8.1
G41N	*		*	Collision detection	8.3
G42	*		*	Left-hand tool radius compensation	8.1
G42N	*		*	Collision detection	8.3
G43	*	?	*	Tool length compensation	8.2
G44	*	?	*	Cancellation of tool length compensation	8.2
G45	*		*	Tangential control (G45)	6.16
G50	*		*	Controlled corner rounding	7.3.3
G51	*		*	Look-Ahead	7.4
G52				Movement until making contact (against hardstop)	6.14
G53				Program coordinates with respect to home	4.3
G54	*		*	Absolute zero offset 1	4.4.2
G55	*		*	Absolute zero offset 2	4.4.2
G56	*		*	Absolute zero offset 3	4.4.2
G57	*		*	Absolute zero offset 4	4.4.2
G58	*		*	Additive zero offset 1	4.4.2
G59	*		*	Additive zero offset 2	4.4.2
G60			*	Straight line canned cycle	10.1
G61			*	Rectangular pattern canned cycle	10.2

Function	M	D	V	Meaning	Section
G62			*	Grid pattern canned cycle	10.3
G63			*	Circular pattern canned cycle	10.4
G64			*	Arc pattern canned cycle	10.5
G65			*	Arc-chord pattern canned cycle	10.6
G66			*	Irregular pocket canned cycle	11.1
G67			*	Irregular pocket roughing	11.1.2
G68			*	Irregular pocket finishing	11.1.4
G69	*		*	Complex deep hole drilling	9.5.1
G70	*	?	*	Programming in inches	3.3
G71	*	?		programming in millimeters	3.3
G72	*		*	General and specific scaling factor	7.6
G73	*		*	Pattern rotation	7.7
G74			*	Machine reference search	4.2
G75			*	Probing until touching	12.1
G76			*	Probing while touching	12.1
G77	*		*	Slaved axis	7.8.1
G77S	*		*	Spindle synchronization	5.5
G78	*	*		Slaved axis cancellation	7.8.2
G78S	*	*		Cancellation of spindle synchronization	5.5
G79				Canned cycle parameter modification	9.2.1
G80	*	*		Canned cycle cancellation	9.3
G81	*		*	Drilling cycle	9.5.2
G82	*		*	Drilling cycle with dwell	9.5.3
G83	*		*	Simple deep hole drilling	9.5.4
G84	*		*	Tapping cycle	9.5.5
G85	*		*	Reaming cycle	9.5.6
G86	*		*	Boring cycle with withdrawal in G00	9.5.7
G87	*		*	Rectangular pocket milling cycle	9.5.8
G88	*		*	Circular pocket milling cycle	9.5.9
G89	*		*	Boring cycle with withdrawal in G01	9.5.10
G90	*	?		Programming in absolute	3.4
G91	*	?	*	Programming in incremental	3.4
G92				Coordinate preset/spindle speed limit	4.4.1
G93				Polar origin preset	4.5
G94	*	?		Feedrate in millimeters(inches) per minute	5.2.1
G95	*	?	*	Feedrate in millimeters(inches) per revolution	5.2.2
G96	*		*	Constant cutting point speed	5.2.3
G97	*	*		Constant tool center speed	5.2.4
G98	*	*		Withdrawal to the starting plane	9.5
G99	*		*	Withdrawal to the reference plane	9.5

M means MODAL, i.e., that once programmed, the G function remains active as long as another incompatible G function is not programmed, M02, M30, EMERGENCY, RESET are not programmed or the CNC is not turned on or off.

Letter **D** means BY DEFAULT, i.e., that these will be assumed by the CNC when turned on, after executing M02, M30 or after EMERGENCY or RESET.

In cases indicated with ? it must be interpreted that the DEFAULT of these G functions depends on the settings of the general CNC machine parameters.

V means that the G function is displayed next to the machining conditions in the execution and simulation modes.

B

INTERNAL CNC VARIABLES

Section (13.2.2)

R indicates that the variable can be read.
W indicates that the variable can be modified.

VARIABLES ASSOCIATED WITH TOOLS

Variable	CNC	PLC	DNC	
TOOL	R	R	R	Number of active tool.
TOD	R	R	R	Number of active tool offset.
NXTOOL	R	R	R	Number of the next requested tool waiting for M06.
NXTOD	R	R	R	Number of the next tool's offset.
TMZPn	R	R	-	(n) tool's position in the tool magazine.
TLFDn	R/W	R/W	-	(n) tool's offset number.
TLFFn	R/W	R/W	-	(n) tool's family code.
TLFNn	R/W	R/W	-	Nominal life assigned to tool (n).
TLFRn	R/W	R/W	-	Real life value of tool (n).
TMZTn	R/W	R/W	-	Contents of tool magazine position (n).
TORn	R/W	R/W	-	Tool radius (R) value of offset (n).
TOLn	R/W	R/W	-	Tool length (L) value of offset (n).
TOIn	R/W	R/W	-	Tool radius wear (I) of offset (n).
TOKn	R/W	R/W	-	Tool length wear (K) of offset (n).

VARIABLES ASSOCIATED WITH ZERO OFFSETS

(Section 13.2.3)

Variable	CNC	PLC	DNC	
ORG(X-C)	R	R	-	Zero offset active on the selected axis without including the additive Zero offset activated via PLC.
PORGF	R	-	R	Abscissa coordinate value of polar origin.
PORGS	R	-	R	Ordinate coordinate value of polar origin.
ORG(X-C)n	R/W	R/W	R	Zero offset (n) value of the selected axis.
PLCOF(X-C)	R/W	R/W	R	Value of the additive Zero Offset activated via PLC.

VARIABLES ASSOCIATED WITH MACHINE PARAMETERS *(Section 13.2.4)*

Variable	CNC	PLC	DNC	
MPGn	R	R	-	Value assigned to general machine parameter (n).
MP(X-C)n	R	R	-	Value assigned to machine parameter (n) of the axis (X-C)
MPSn	R	R	-	Value assigned to machine parameter (n) of the main spindle.
MPSSn	R	R	-	Value assigned to machine parameter (n) of the second spindle.
MPASn	R	R	-	Value assigned to machine parameter (n) of the auxiliary spindle.
MPLCn	R	R	-	Value assigned to machine parameter (n) of the PLC.

VARIABLES ASSOCIATED WITH THE WORK ZONES *(Section 13.2.5)*

Variable	CNC	PLC	DNC	
FZONE	R	R/W	R	Status of work zone 1.
FZLO(X-C)	R	R/W	R	Lower limit of work zone 1 along the selected axis (X/C).
FZUP(X-C)	R	R/W	R	Upper limit of work zone 1 along the selected axis (X/C).
SZONE	R	R/W	R	Status of work zone 2.
SZLO(X-C)	R	R/W	R	Lower limit of work zone 2 along the selected axis (X/C).
SZUP(X-C)	R	R/W	R	Upper limit of work zone 2 along the selected axis (X/C)
TZONE	R	R/W	R	Status of work zone 3.
TZLO(X-C)	R	R/W	R	Lower limit of work zone 3 along the selected axis (X/C).
TZUP(X-C)	R	R/W	R	Upper limit of work zone 3 along the selected axis (X/C).
FOZONE	R	R/W	R	Status of work zone 4.
FOZLO(X-C)	R	R/W	R	Lower limit of work zone 4 along the selected axis (X/C).
FOZUP(X-C)	R	R/W	R	Upper limit of work zone 4 along the selected axis (X/C).
FIZONE	R	R/W	R	Status of work zone 5.
FIOZLO(X-C)	R	R/W	R	Lower limit of work zone 5 along the selected axis (X/C).
FIZUP(X-C)	R	R/W	R	Upper limit of work zone 5 along the selected axis (X/C).

VARIABLES ASSOCIATED WITH FEEDRATES

Section (13.2.6)

Variable	CNC	PLC	DNC	
FREAL	R	R	R	Real feedrate of the CNC in mm/min or inch/min.

Variables associated with function G94

FEED	R	R	R	Active feedrate at the CNC (G94) in mm/min or inch/min.
DNCF	R	R	R/W	Feedrate selected via DNC.
PLCF	R	R/W	R	Feedrate selected via PLC.
PRGF	R	R	R	Feedrate selected by program.

Variables associated with function G95

FPREV	R	R	R	Active feedrate at CNC (G95), in m/rev or inch/rev.
DNCFPR	R	R	R/W	Feedrate selected via DNC.
PLCFPR	R	R/W	R	Feedrate selected via PLC.
PRGFPR	R	R	R	Feedrate selected by program.

Variables associated with function G32

PRGFIN	R	R	R	Feedrate selected by program. In 1/min.

Variables associated with Feedrate Override

FRO	R	R	R	Feedrate Override (%) active at the CNC.
PRGFRO	R/W	R	R	Feedrate Override (%) selected by program.
DNCFRO	R	R	R/W	Feedrate Override (%) selected by DNC.
PLCFRO	R	R/W	R	Feedrate Override (%) selected by PLC.
CNCFRO	R	R	R	Feedrate Override (%) selected from the front panel knob.
PLCCFR	R	R/W	R	Feedrate Override (%) of the PLC execution channel.

VARIABLES ASSOCIATED WITH POSITION VALUES

Section (13.2.7)

Variable	CNC	PLC	DNC	
PPOS(X-C)	R	-	-	Theoretical programmed position value (coordinate).
POS(X-C)	R	R	R	Real position value of the indicated axis.
TPOS(X-C)	R	R	R	Theoretical (real + lag) position value of the indicated axis.
FLWE(X-C)	R	R	R	Following error of the indicated axis.
DEFLEX	R	R	R	Probe deflection along the X axis.
DEFLEY	R	R	R	Probe deflection along the Y axis.
DEFLEZ	R	R	R	Probe deflection along the Z axis.
DIST(X-C)	R/W	R/W	R	Distance travelled by the indicated axis.
LIMPL(X-C)	R/W	R/W	R	Upper second travel limit.
LIMMI(X-C)	R/W	R/W	R	Lower second travel limit.

VARIABLES ASSOCIATED WITH HANDWHEELS

(Section13.2.8)

Variable	CNC	PLC	DNC	
HANPF	R	R	-	1st handwheel pulses received since CNC power-up.
HANPS	R	R	-	2nd handwheel pulses received since CNC power-up.
HANPT	R	R	-	3rd handwheel pulses received since CNC power-up.
HANPFO	R	R	-	4th handwheel pulses received since CNC power-up.
HANFCT	R	R/W	R	Individual multiplying factor for each handwheel.
HBEVAR	R	R/W	R	HBE handwheel: counting enabled, jog axis and (x1, x10, x100) factor
MASLAN	R/W	R/W	R/W	Angle of linear path with "Path handwheel".
MASCFI	R/W	R/W	R/W	Arc center coordinates with "Path handwheel".
MASCSE	R/W	R/W	R/W	Arc center coordinates with "Path handwheel".

VARIABLES ASSOCIATED WITH THE <u>MAIN</u> SPINDLE *(Section 13.2.9)*

Variable	CNC	PLC	DNC	
SREAL	R	R	R	Real spindle speed in r.p.m.
SPEED	R	R	R	Active spindle speed at the CNC.
DNCS	R	R	R/W	Spindle speed selected via DNC.
PLCS	R	R/W	R	Spindle speed selected via PLC.
PRGS	R	R	R	Spindle speed selected by program.
SSO	R	R	R	Spindle Speed Override (%) active at the CNC.
PRGSSO	R/W	R	R	Spindle Speed Override (%) selected by program.
DNCSSO	R	R	R/W	Spindle Speed Override (%) selected via DNC.
PLCSSO	R	R/W	R	Spindle Speed Override (%) selected via PLC.
CNCSSO	R	R	R	Spindle Speed Override (%) selected from front panel.
SLIMIT	R	R	R	Spindle speed limit, in rpm, active at the CNC.
DNCSL	R	R	R/W	Spindle speed limit selected via DNC.
PLCSL	R	R/W	R	Spindle speed limit selected via PLC.
PRGSL	R	R	R	Spindle speed limit selected by program.
POSS	R	R	R	Real Spindle position. Between ±999999999 ten-thousandths º
RPOSS	R	R	R	Real Spindle position. Between 0 and 360º (in ten-thousandths º)
TPOSS	R	R	R	Theoretical Spindle position. Between ±999999999 ten-thousandths of a degree
RTPOSS	R	R	R	Theoretical Spindle position. Between 0 and 360º (in ten-thousandths of a degree.
FLWES	R	R	R	Spindle following error in degrees.
SYNCER	R	R	R	Second spindle following error when synchronized with the main spindle.

VARIABLES ASSOCIATED WITH THE <u>SECOND</u> SPINDLE *(Section 13.2.10*

Variable	CNC	PLC	DNC	
SSREAL	R	R	R	Real spindle speed in r.p.m.
SSPEED	R	R	R	Active spindle speed at the CNC.
SDNCS	R	R	R/W	Spindle speed selected via DNC.
SPLCS	R	R/W	R	Spindle speed selected via PLC.
SPRGS	R	R	R	Spindle speed selected by program.
SSSO	R	R	R	Spindle Speed Override (%) active at the CNC.
SPRGSO	R/W	R	R	Spindle Speed Override (%) selected by program.
SDNCSO	R	R	R/W	Spindle Speed Override (%) selected via DNC.
SPLCSO	R	R/W	R	Spindle Speed Override (%) selected via PLC.
SCNCSO	R	R	R	Spindle Speed Override (%) selected from front panel.
SSLIMI	R	R	R	Spindle speed limit, in rpm, active at the CNC.
SDNCSL	R	R	R/W	Spindle speed limit selected via DNC.
SPLCSL	R	R/W	R	Spindle speed limit selected via PLC.
SPRGSL	R	R	R	Spindle speed limit selected by program.
SPOSS	R	R	R	Real Spindle position. Between ±999999999 ten-thousandths º
SRPOSS	R	R	R	Real Spindle position. Between 0 and 360º (in ten-thousandths º)
STPOSS	R	R	R	Theoretical Spindle position. Between ±999999999 ten-thousandths of a degree.
SRTPOS	R	R	R	Theoretical Spindle position. Between 0 and 360º (in ten-thousandths of a degree.
SFLWES	R	R	R	Spindle following error in degrees.

VARIABLES ASSOCIATED WITH THE LIVE TOOL *Section (13.2.11)*

Variable	CNC	PLC	DNC	
ASPROG	R	R	-	RPM programmed in M45 S (within the associated subroutine)

VARIABLES ASSOCIATED WITH THE PLC *Section (13.2.12)*

Variable	CNC	PLC	DNC	
PLCMSG	R	-	R	Number of the active PLC message with the highest priority.
PLCIn	R/W	-	-	32 PLC inputs starting from (n).
PLCOn	R/W	-	-	32 PLC outputs starting from (n).
PLCMn	R/W	-	-	32 PLC marks starting from (n).
PLCRn	R/W	-	-	Indicated (n) Register.
PLCTn	R/W	-	-	Indicated (n) Timer's count.
PLCCn	R/W	-	-	Indicated (n) Counter's count.

VARIABLES FOR GLOBAL AND LOCAL PARAMETERS *(Section 13.2.13)*

Variable	CNC	PLC	DNC	
GUPn	-	R/W	-	Global parameter (n) (100-P299).
LUP (a,b)	-	R/W	-	Local parameter (b) and its nesting level (a). (P0-P25).
CALLP	R	-	-	Indicates which local parameters have been defined by means of a PCALL or MCALL instruction (calling a subroutine).

VARIABLES SERCOS *(Section 13.2.14)*

Variable	CNC	PLC	DNC	
SETGE(X-C)	W	W	-	Work gear and parameter set for (X-C) axis drive
SETGES	W	W	-	Work gear and parameter set for main spindle drive
SSETGS	W	W	-	Work gear and parameter set for 2nd spindle drive
SVAR(X-C)id	R/W	-	-	Sercos variable for (X-C) axis identifier "id"
SVARS id	R/W	-	-	Sercos variable for main spindle identifier "id"
SSVAR id	R/W	-	-	Sercos variable for 2nd spindle identifier "id"
TSVAR(X-C)id	R	-	-	Third attribute of the sercos variable of (X-C) axis identifier "id"
TSVARS id	R	-	-	Third attribute of the sercos variable of main spindle identifier "id"
TSSVAR id	R	-	-	Third attribute of the sercos variable of 2nd spindle identifier "id"

SOFTWARE & HARDWARE CONFIGURATION VARIABLES *(Section 13.2.15)*

Variable	CNC	PLC	DNC	
HARCON	R	R	R	Its bits indicate the CNC hardware configuration.
IDHARH	R	R	R	Hardware identifier (8 least significant bits)
IDHARL	R	R	R	Hardware identifier (4 most significant bits)
SOFCON	R	R	R	CNC & HD software versions (bits 15-0) and (31-16) respectively.

VARIABLES ASSOCIATED WITH TELEDIAGNOSIS *(Section 13.2.16)*

Variable	CNC	PLC	DNC	
HARSWA	R	R	R	Hardware configuration
HARSWB	R	R	R	Hardware configuration
HARTST	R	R	R	Hardware test
MEMTST	R	R	R	Memory test
NODE	R	R	R	Node number within the Sercos ring
VCHECK	R	R	R	Checksum of the software version

VARIABLES ASSOCIATED WITH THE OPERATING MODE *(Section 13.2.17)*

Variable	CNC	PLC	DNC	
OPMODE	R	R	R	Operating mode.
OPMODA	R	R	R	Operating mode when working in the main channel.
OPMODB	R	R	R	Type of simulation.
OPMODC	R	R	R	Axes selected by handwheel.

OTHER VARIABLES *(Section 13.2.18)*

Variable	CNC	PLC	DNC	
NBTOOL	R	R	R	Number of the tool being managed
PRGN	R	R	R	Number of the program in execution.
BLKN	R	R	R	Label number of the last executed block.
GSn	R	-	-	Status of the indicated G function (n).
GGSA	-	R	R	Status of functions G00 thru G24.
GGSB	-	R	R	Status of functions G25 thru G49.
GGSC	-	R	R	Status of functions G50 thru G74.
GGSD	-	R	R	Status of functions G75 thru G99.
MSn	R	-	-	Status of the indicated M function (n)
GMS	-	-	R	Status of M functions: M (0..6, 8, 9, 19, 30, 41..44)
PLANE	R	R	R	Axes which form the active main plane.
LONGAX	R	R	R	Axis affected by the tool length compensation (G15).
MIRROR	R	R	R	Active mirror images.
SCALE	R	R	R	Active general Scaling factor.
SCALE(X-C)	R	R	R	Scaling Factor applied only to the indicated axis.
ORGROT	R	R	R	Rotation angle (G73) of the coordinate system in degrees.
ROTPF	R	-	-	Abscissa of rotation center.
ROTPS	R	-	-	Ordinate of rotation center.
PRBST	R	R	R	Returns probe status.
CLOCK	R	R	R	System clock in seconds.
TIME	R	R	R/W	Time in Hours, minutes and seconds.
DATE	R	R	R/W	Date in Year-Month-Day format
TIMER	R/W	R/W	R/W	Clock activated by PLC, in seconds.
CYTIME	R	R	R	Time to execute a part in hundredths of a second.
PARTC	R/W	R/W	R/W	Part counter of the CNC.
FIRST	R	R	R	Flag to indicate first time of program execution.
KEY	R/W*	R/W	R/W	keystroke code.
KEYSRC	R/W	R/W	R/W	Keystroke source, 0=keyboard, 1=PLC, 2=DNC
ANAIn	R	R	R	Voltage (in volts) of the indicated analog input (n).
ANAOn	W	W	W	Voltage (in volts) to apply to the indicated output (n).
CNCERR	-	R	R	Active CNC error number.
PLCERR	-	-	R	Active PLC error number.
DNCERR	-	R	-	Number of the error generated during DNC communications.
AXICOM	R	R	R	Pair of axes toggled with function G28
TANGAN	R	R	R	Associated with G45. Angular position, in degrees, with respect to programmed path.

Warning:

The "KEY" variable can be "written" (W) at the CNC only via the user channel.

C

HIGH LEVEL PROGRAMMING

DISPLAY STATEMENTS

(Section 14.2)

(ERROR whole number, "error text")	Stops execution of program and displays indicated error.
(MSG "message")	Displays indicated message.
(DGWZ expression 1, ... expression 6)	Define the graphics display area

ENABLING/DISABLING STATEMENTS

(Section 14.3)

(ESBLK and DSBLK)	The CNC executes all the blocks which are found between ESBLK and DSBLK as if they were a single block.
(ESTOP and DSTOP)	Enable (ESTOP) and disable (DSTOP) of the Stop key and the external Stop signal (PLC)
(EFHOLD and DFHOLD)	Enable (EFHOLD) and disable (DFHOLD) of the Feed-Hold input (PLC)

FLOW CONTROLLING STATEMENTS

(Section 14.4)

(GOTO N(expression))	Causes a jump within the same program, to the block defined by label N(expression)
(RPT N(expression), N(expression))	Repeats the execution of the part of a program existing between two blocks defined by means of labels N(expression)
(IF condition <action1> ELSE <action2>)	Analyzes the given condition which must be a relational expression. If the condition is true (result equals 1), <action1> will be executed, otherwise (result equals 0) <action2> will be executed.

SUBROUTINE STATEMENTS

(Section 14.5)

(SUB integer)	Definition of subroutine
(RET)	End of subroutine
(CALL (expression))	Call to subroutine
(PCALL (expression, (assignment statement), (assignment statement),...)	Call to a subroutine. Besides, allows the initialization, by means of assignment statements, of up to 26 local parameters of this subroutine.
(MCALL (expression), (assignment statement), (assignment statement),...)	The same as PCALL, but converting the subroutine indicated into a modal subroutine.
(MDOFF)	Cancellation of modal subroutine
(PROBE (expression),(assignment statement), (assignment statement),...)	Executes a probing canned cycle, its parameters being initialized by means of assignment statements.
(REPOS X, Y, Z, ...)	It must always be used inside interruption subroutines and it facilitates the repositioning of the machine axes to the interruption point.

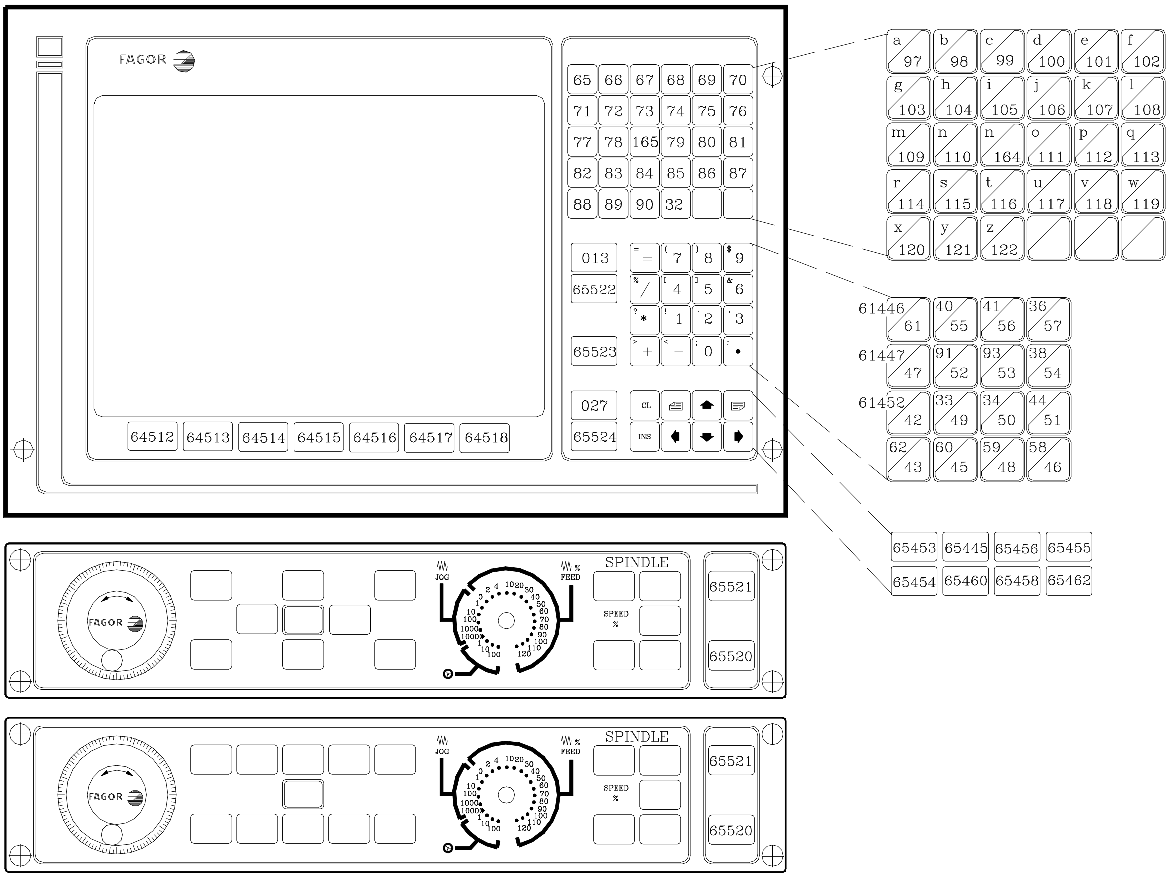
PROGRAM STATEMENTS *(Section 14.6)*

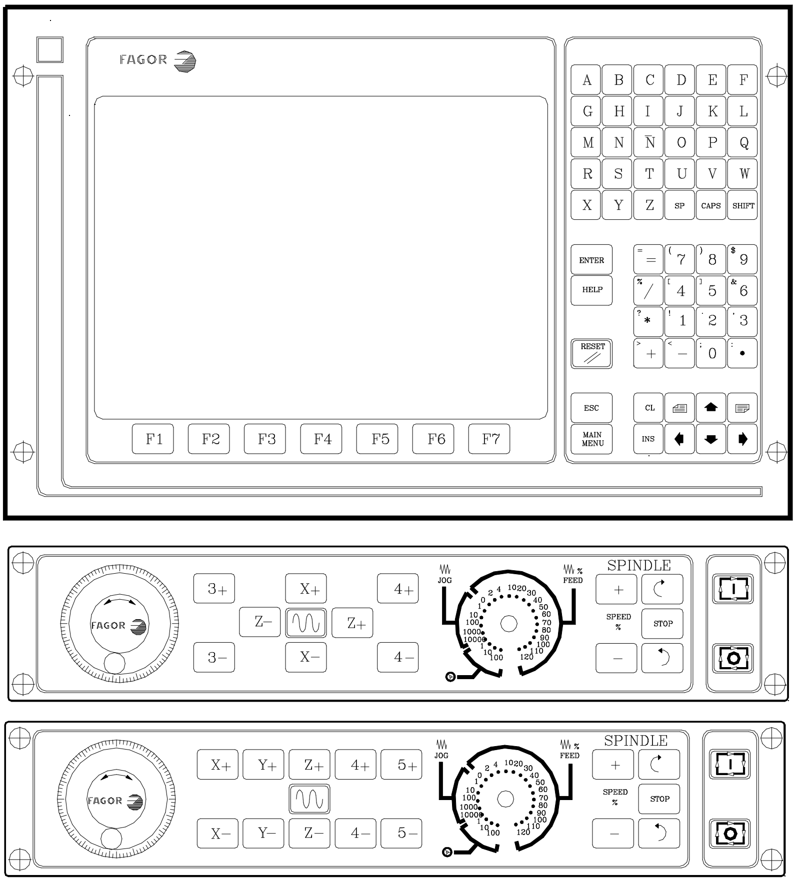
(EXECP(expression),(directory)	Starts the execution of the program
(OPEN P(expression),(destination directory),A/D,"program comment")	Starts generating a new program and allows it to be associated with a program comment.
(WRITE <block text>)	Adds the information contained in <block text> after the last program block of the program which was being generated with OPEN P, as a new program block.

CUSTOMIZING STATEMENTS *(Section 14.7)*

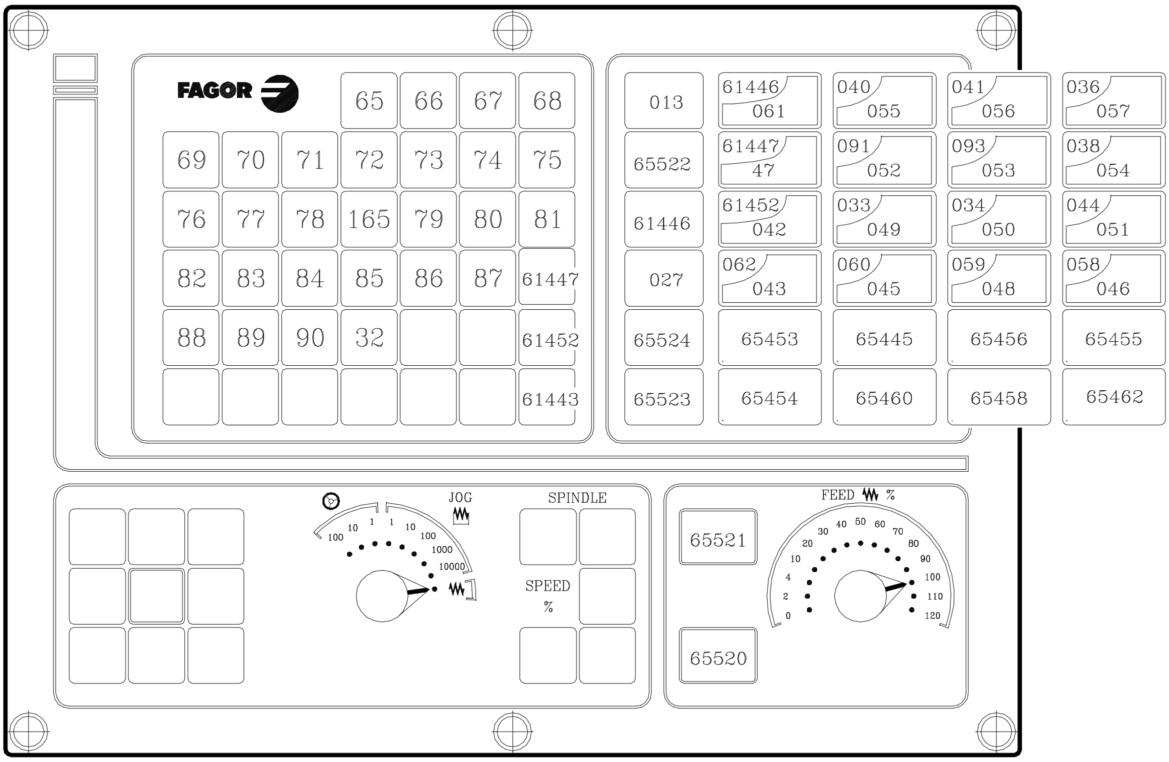
(PAGE(expression))	Displays the user page number (0-255) or system page number (>1000) indicated.
(SYMBOL (expression 1),(expression 2),(expression 3)	Displays the symbol (0-255) indicated by expression 1 Its position on the screen is defined by expression 2 (row,0-639) and by expression 3(column,0-335).
(IB(expression)=INPUT"text",format)	Displays the text indicated in the data input window and stores the data input by the user in the input variable (IBn).
(ODW(expression 1),(expression 2),(expression 3)	Defines and draws a white window on screen (1 row x 14 columns). Its position on screen is defined by expression 2(row) and by expression 3 (column).
(DW (expression 1)=(expression 2), DW(expression 3) = (expression 4),...)	Displays the numerical data indicated by expression 2,4,.. in windows indicated by the value of expression 1,3....
(SK (expression 1)="text 1", (expression 2)="text 2",...)	Defines and displays the new softkey menu indicated.
(WKEY)	Stops the execution of a program until a key is pressed.
(WBUF"text"(expression))	Adds the text and the value of the expression, once this has been evaluated, to the block which is being edited and in the data input window.
(SYSTEM)	Ends the execution of user customized program and returns to standard CNC menu.

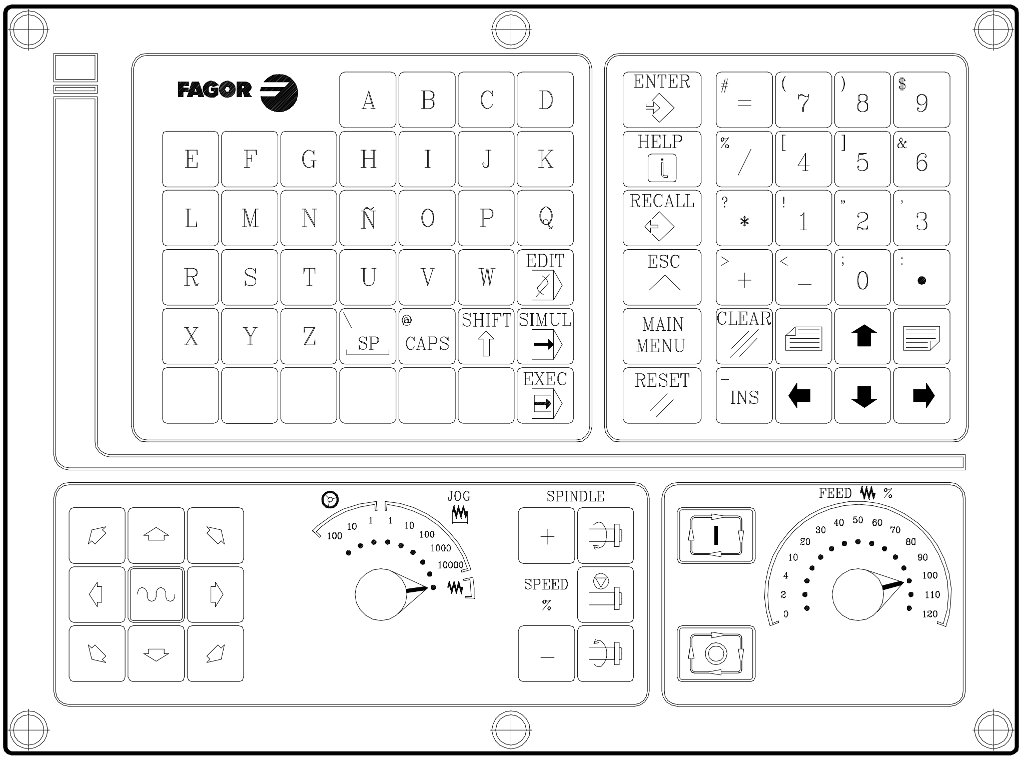
KEY CÓDES


FAGOR
65 66 67 68 69 70
71 72 73 74 75 76
77 78 165 79 80 81
82 83 84 85 86 87
88 89 90 32
013
65522
65523
027
65524
CL
INS
64512 64513 64514 64515 64516 64517 64518
a 97, b 98, c 99, d 100, e 101, f 102
g 103, h 104, i 105, j 106, k 107, l 108
m 109, n 110, n 164, o 111, p 112, q 113
r 114, s 115, t 116, u 117, v 118, w 119
x 120, y 121, z 122
61446 61, 40 55, 41 56, 36 57
61447 47, 91 52, 93 53, 38 54
61452 42, 33 49, 34 50, 44 51
62 43, 60 45, 59 48, 58 46
65453 65445 65456 65455
65454 65460 65458 65462
JOG
FEED
SPINDLE
SPEED %
65521
65520

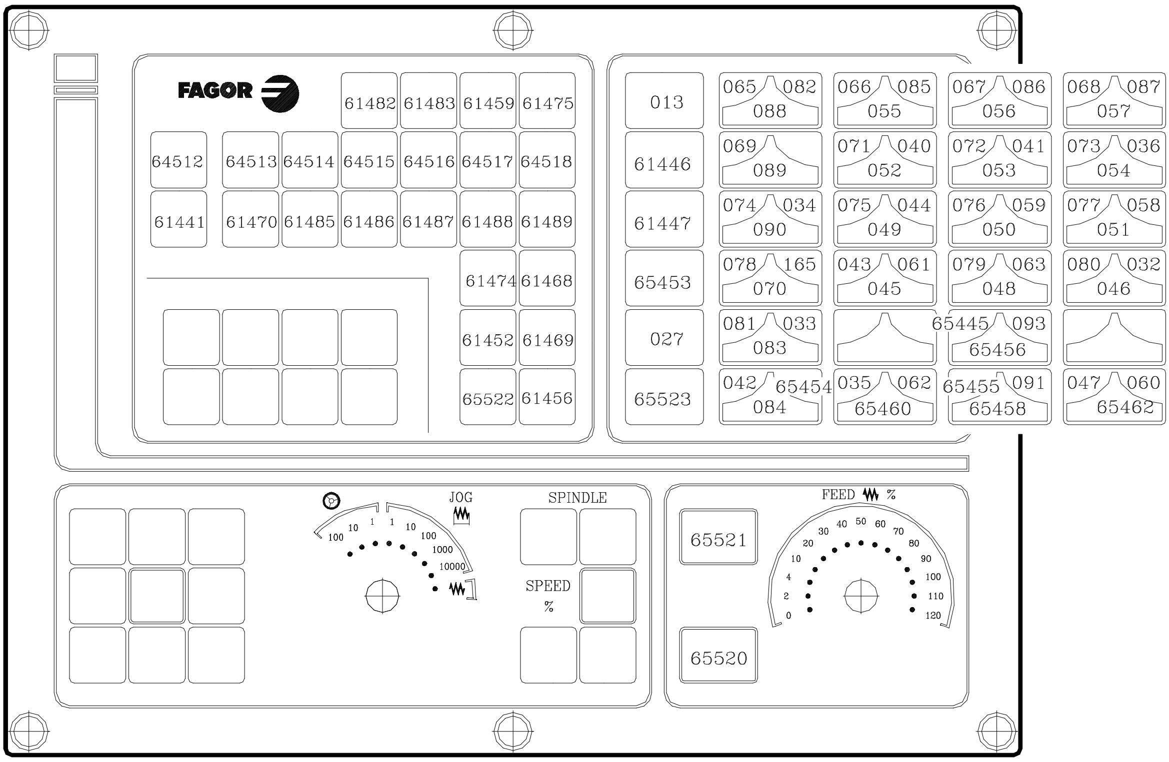

FAGOR
A B C D E F
G H I J K L
M N Ñ O P Q
R S T U V W
X Y Z SP CAPS SHIFT
ENTER
HELP
RESET
ESC
MAIN MENU
CL
INS
F1 F2 F3 F4 F5 F6 F7
3+ X+ 4+
Z– Z+
3– X– 4–
SPINDLE
JOG
FEED
SPEED %
STOP
X+ Y+ Z+ 4+ 5+
X– Y– Z– 4– 5–

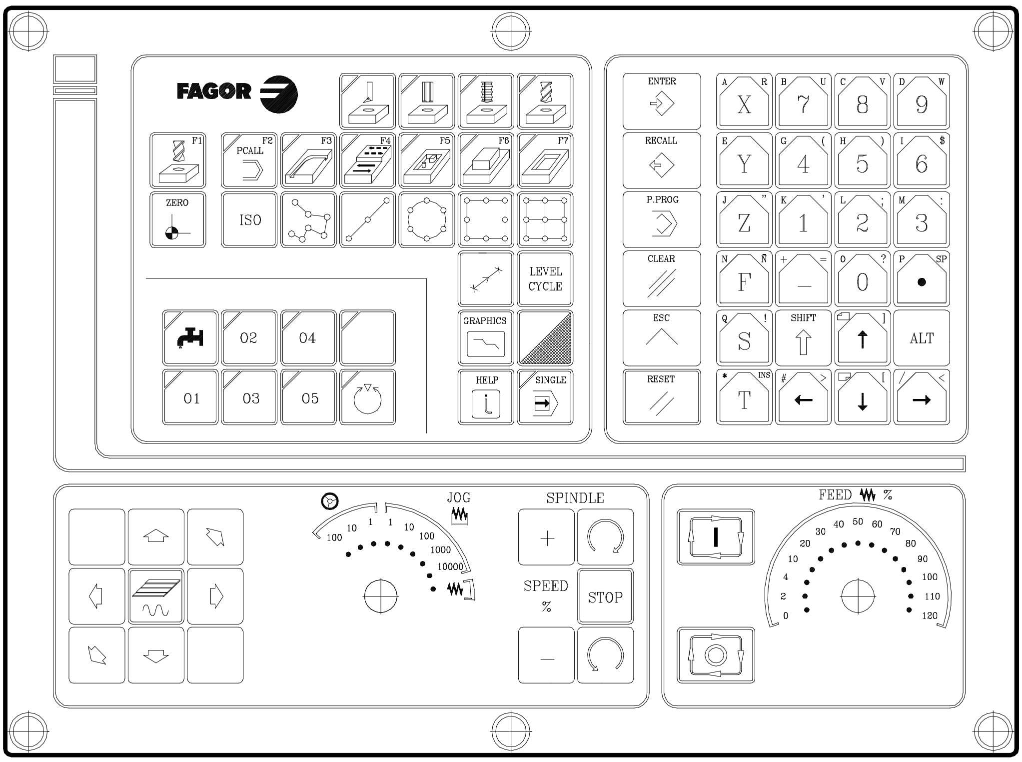
Alphanumeric operator panel (M model)

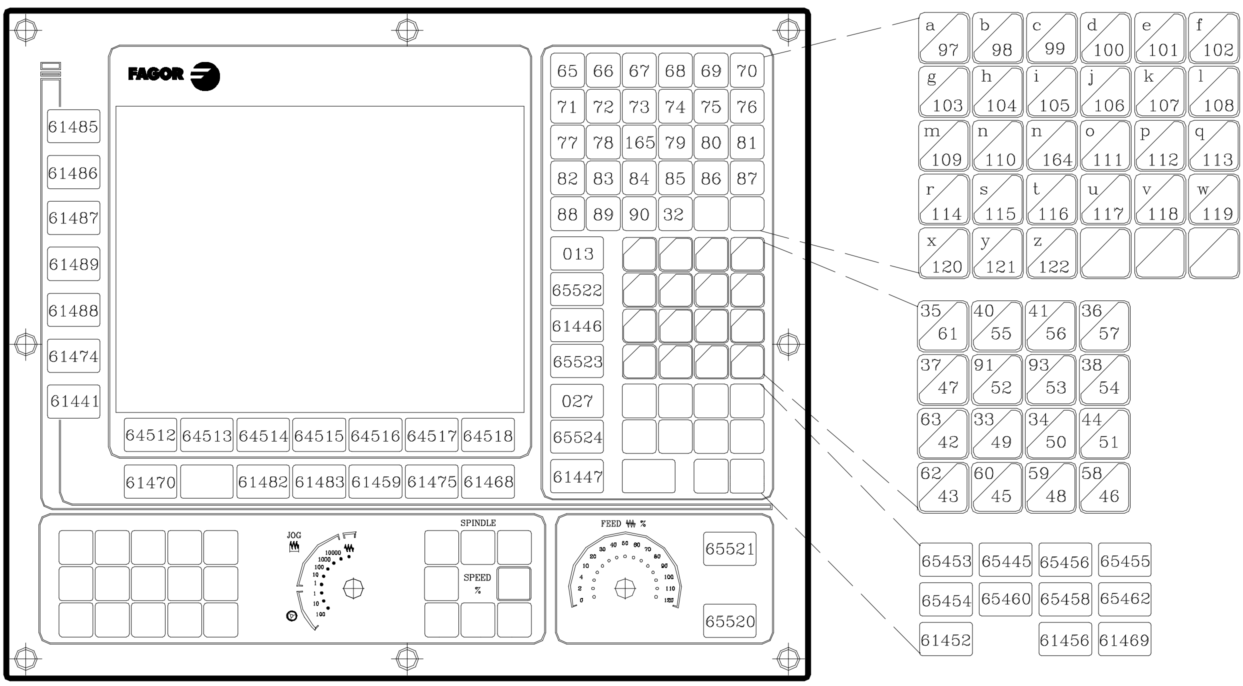


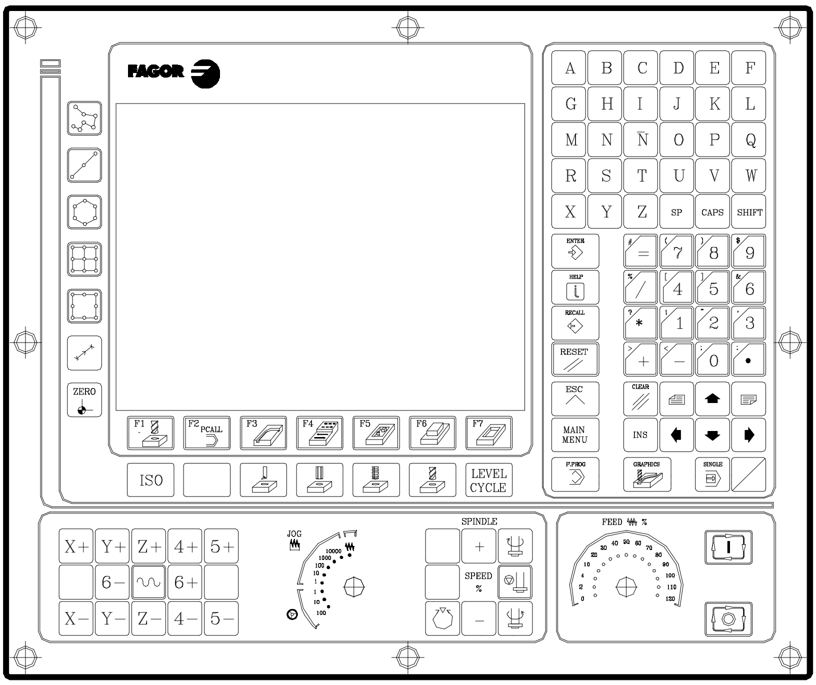


MC operator panel

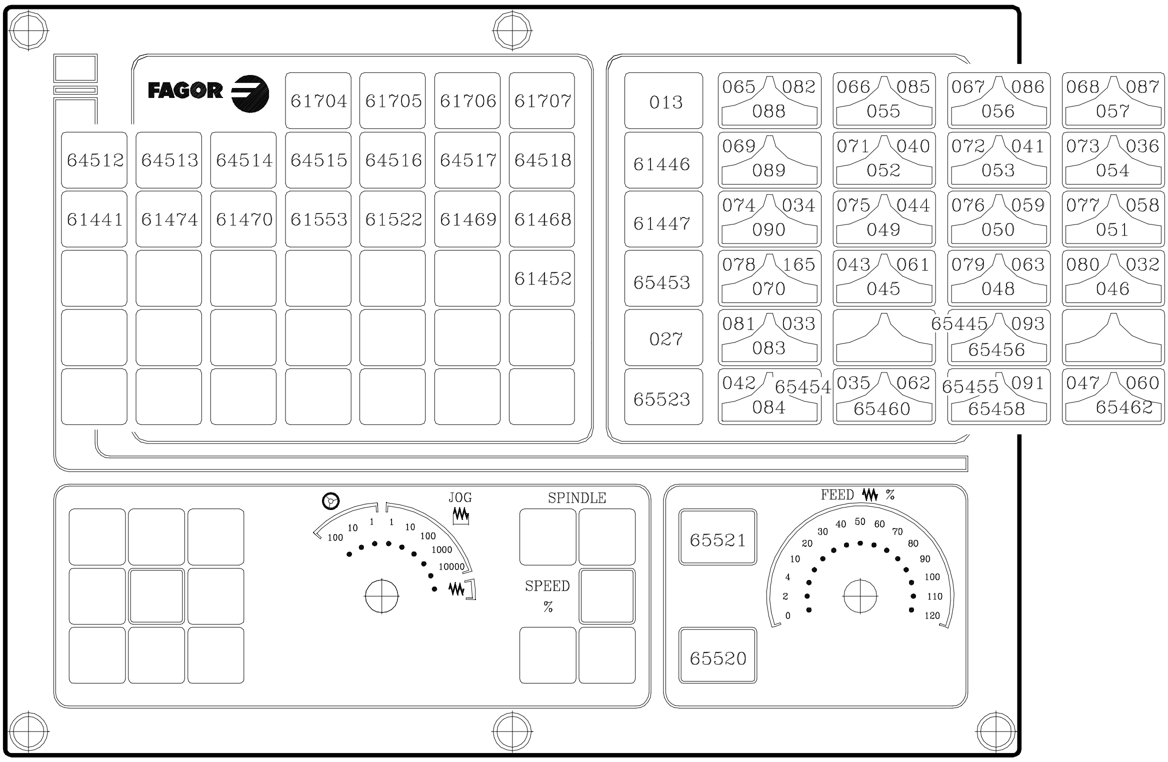


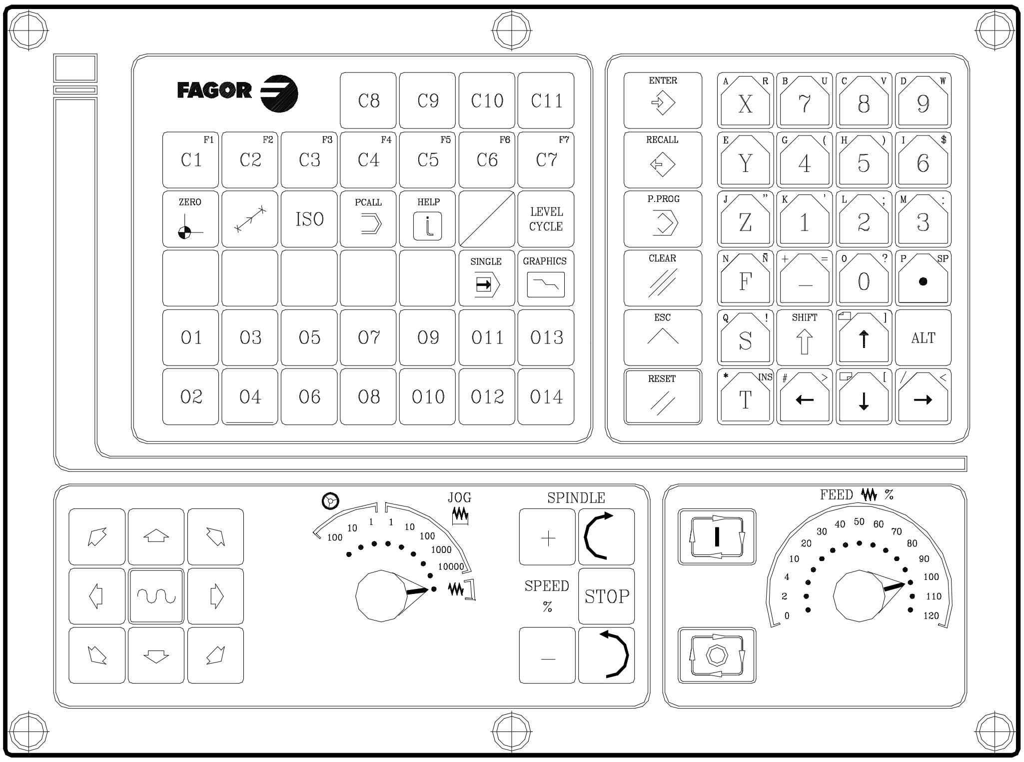



FAGOR
61485
61486
61487
61489
61488
61474
61441
64512 64513 64514 64515 64516 64517 64518
61470 61482 61483 61459 61475 61468
65 66 67 68 69 70
71 72 73 74 75 76
77 78 165 79 80 81
82 83 84 85 86 87
88 89 90 32
013
65522
61446
65523
027
65524
61447
JOG
SPINDLE
SPEED %
FEED %
65521
65520
a 97 b 98 c 99 d 100 e 101 f 102
g 103 h 104 i 105 j 106 k 107 l 108
m 109 n 110 ñ 164 o 111 p 112 q 113
r 114 s 115 t 116 u 117 v 118 w 119
x 120 y 121 z 122
35 61 40 55 41 56 36 57
37 47 91 52 93 53 38 54
63 42 33 49 34 50 44 51
62 43 60 45 59 48 58 46
65453 65445 65456 65455
65454 65460 65458 65462
61452 61456 61469


FAGOR
ZERO
F1 F2 PCALL F3 F4 F5 F6 F7
ISO
LEVEL CYCLE
A B C D E F
G H I J K L
M N Ñ O P Q
R S T U V W
X Y Z SP CAPS SHIFT
ENTER = 7 8 9
HELP / 4 5 6
RECALL * 1 2 3
RESET + − 0 •
ESC CLEAR
MAIN MENU INS
P.PROG GRAPHICS SINGLE
X+ Y+ Z+ 4+ 5+
6− 6+
X− Y− Z− 4− 5−
JOG
SPINDLE
+
SPEED %
−
FEED %

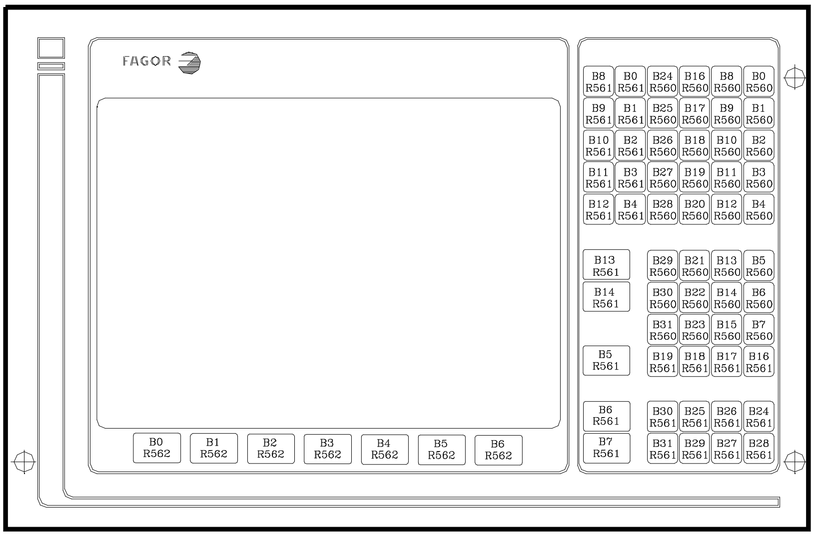
MCO/TCO operator panel

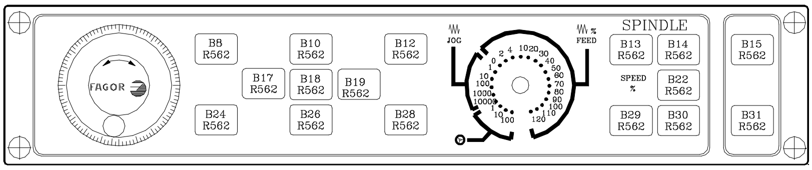


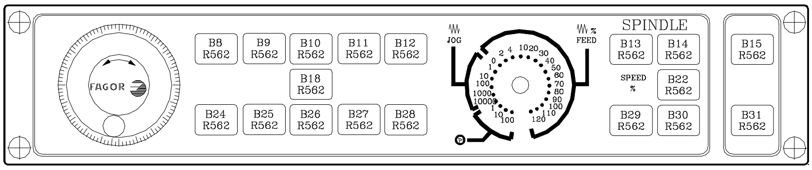


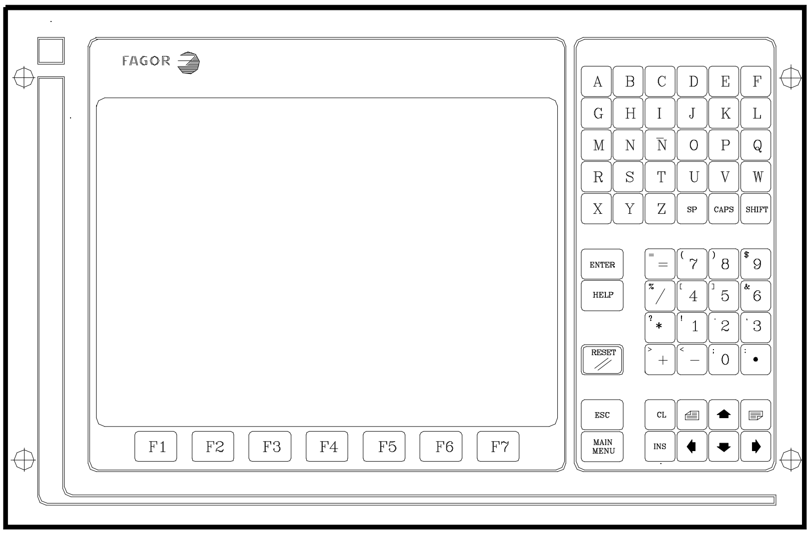
E

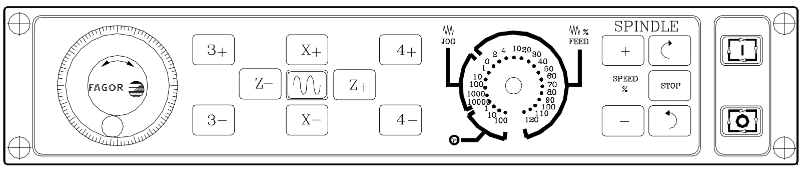
LOGIC OUTPUTS FOR KEY STATUS

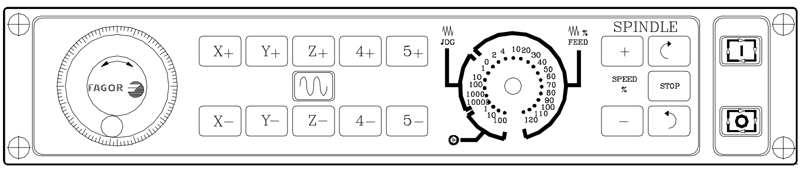




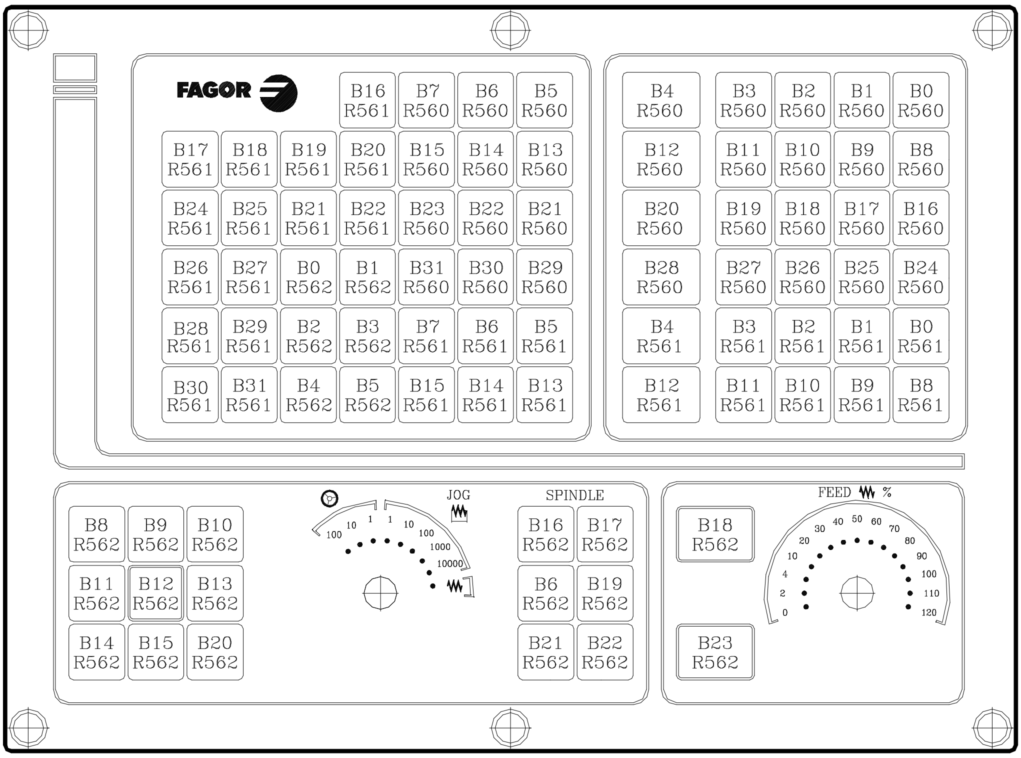


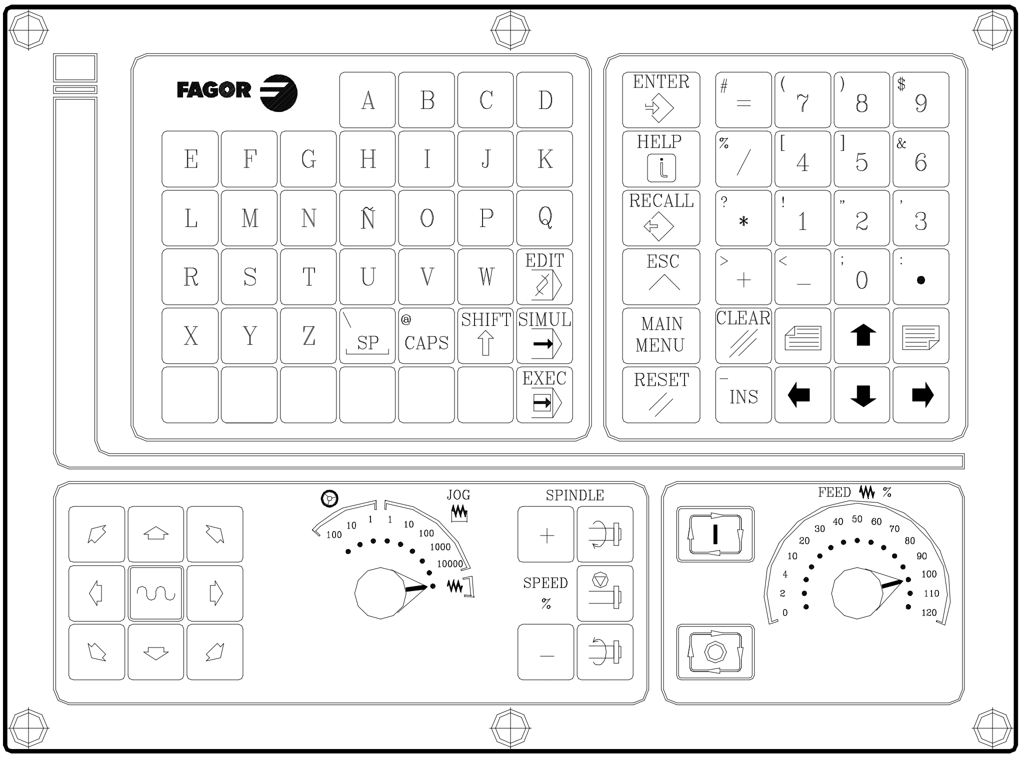




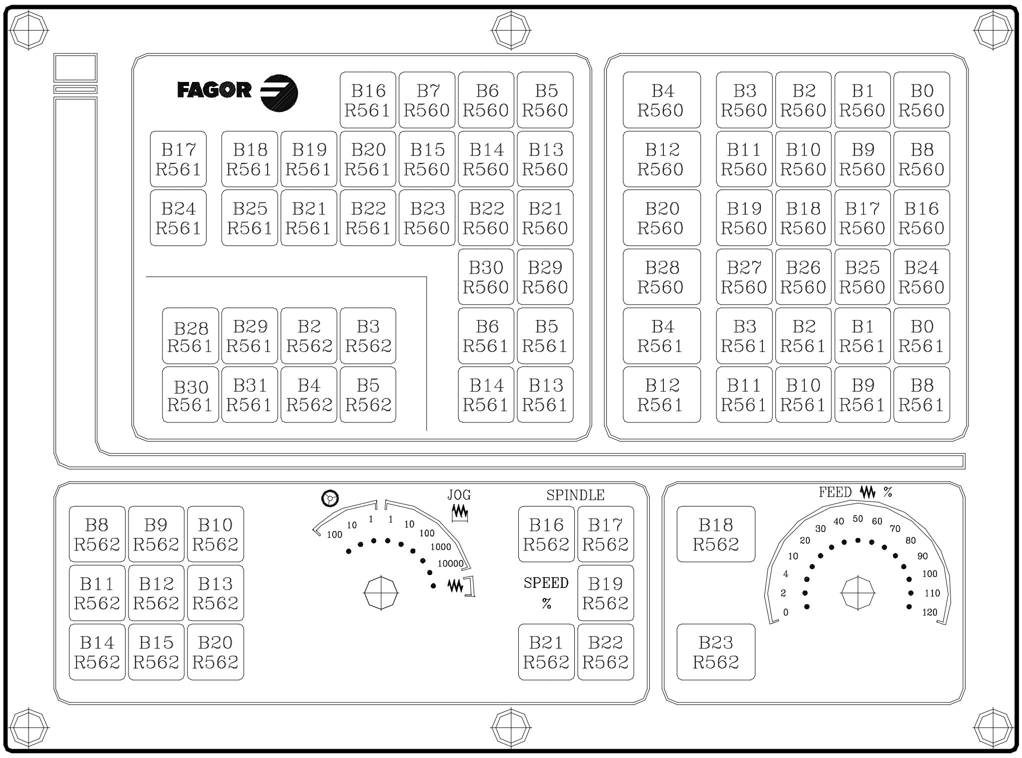


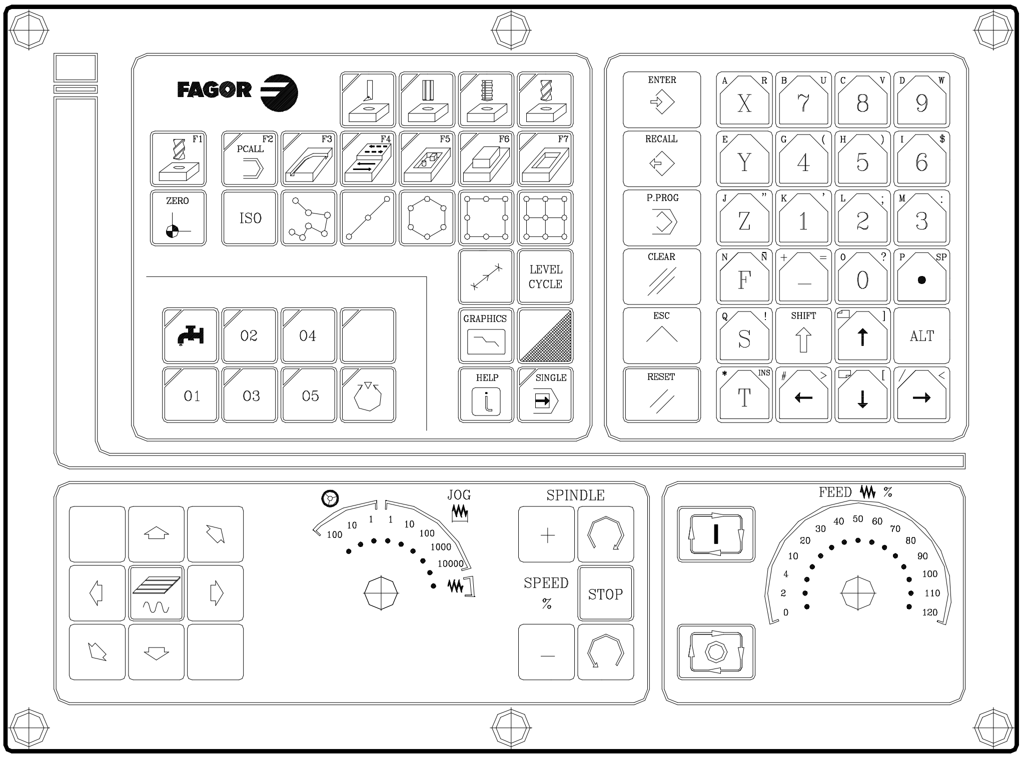
Alphanumeric operator panel (M model)

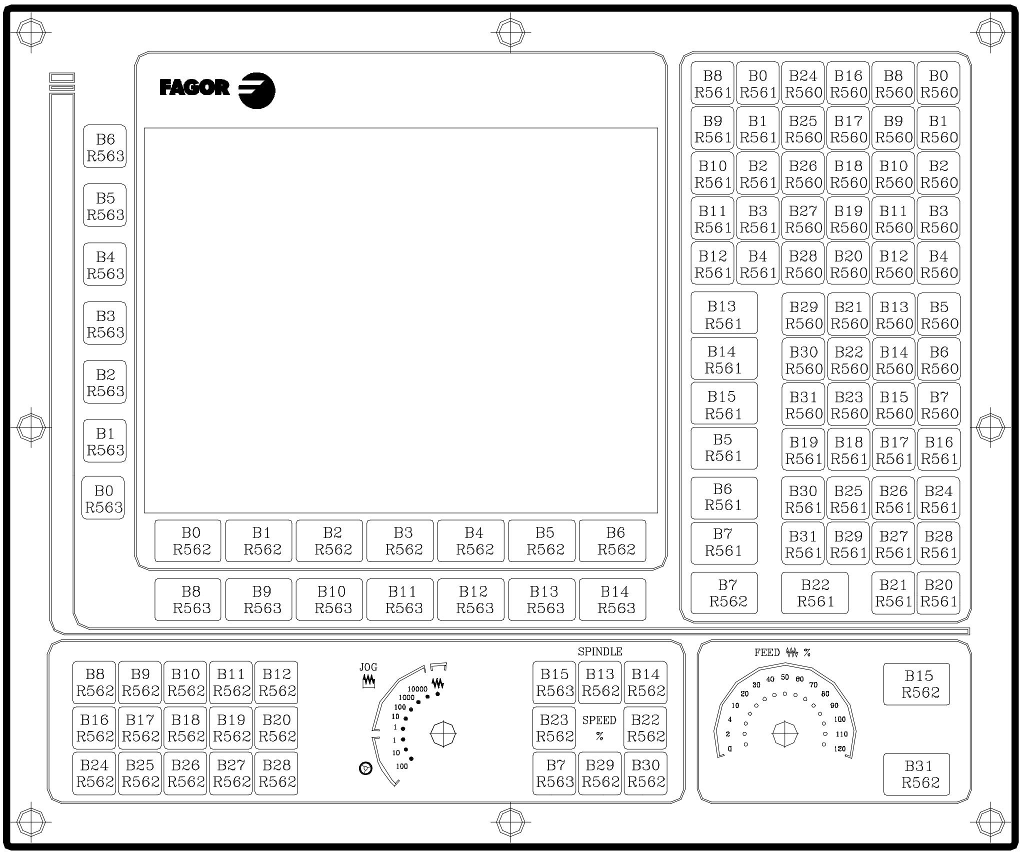


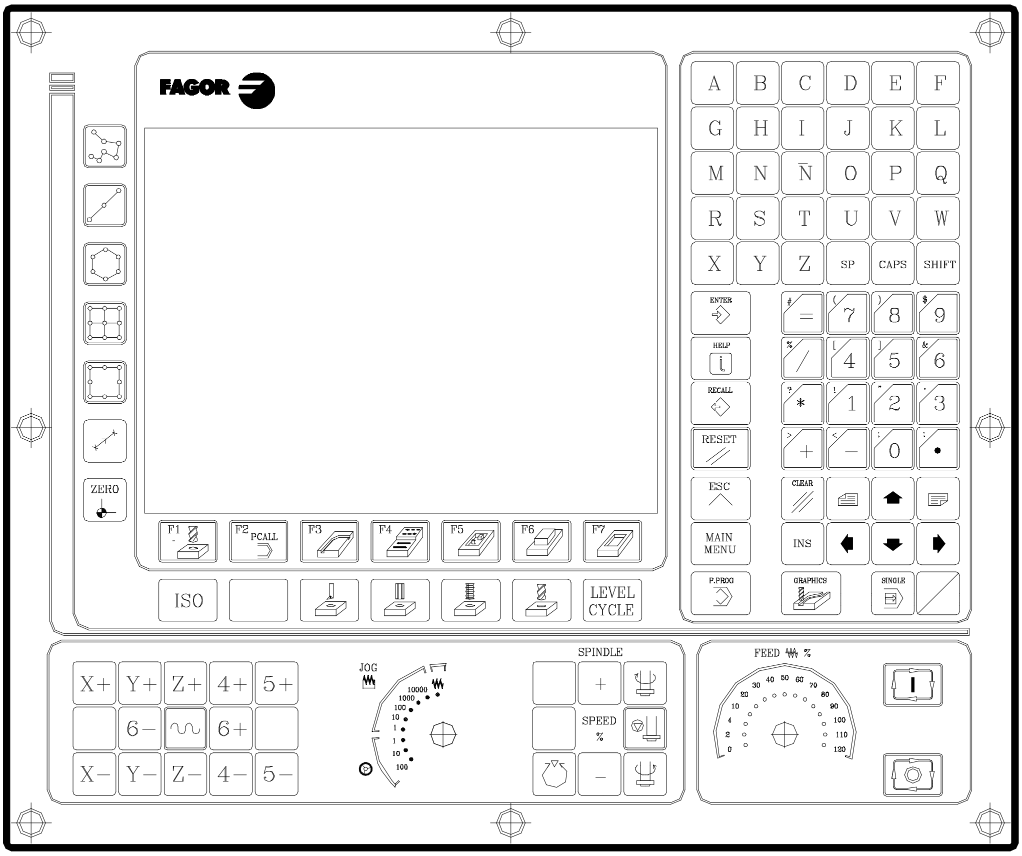


MC operator panel

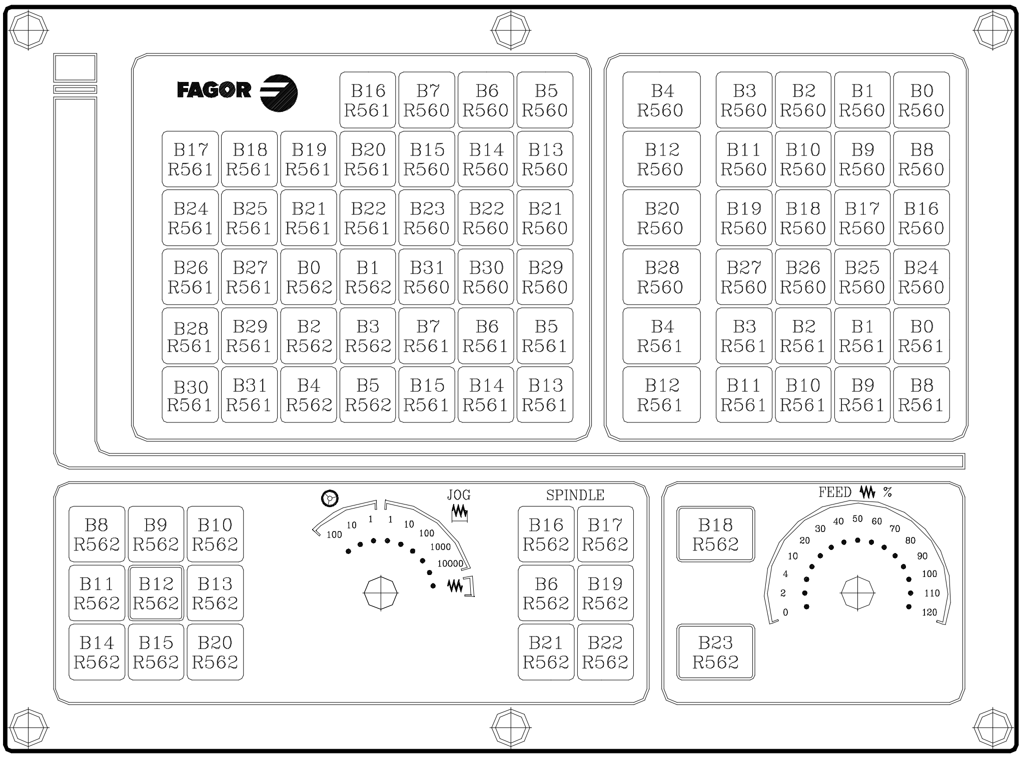


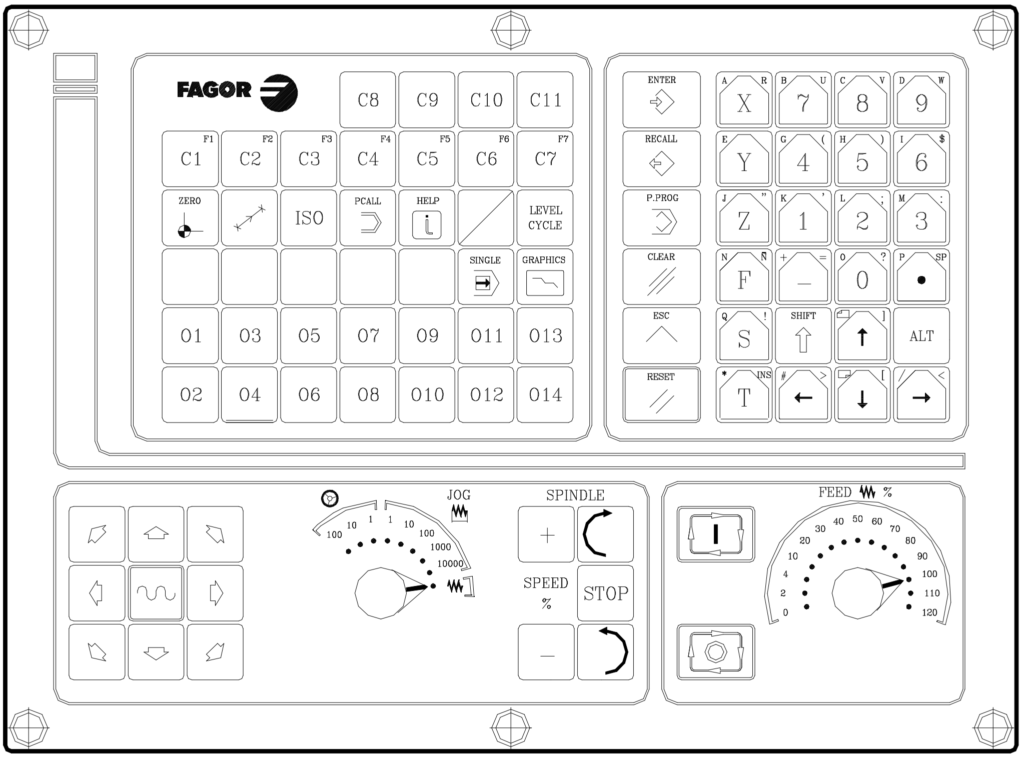



FAGOR
B6 R563
B5 R563
B4 R563
B3 R563
B2 R563
B1 R563
B0 R563
B0 R562 B1 R562 B2 R562 B3 R562 B4 R562 B5 R562 B6 R562
B8 R563 B9 R563 B10 R563 B11 R563 B12 R563 B13 R563 B14 R563
B8 R561 B0 R561 B24 R560 B16 R560 B8 R560 B0 R560
B9 R561 B1 R561 B25 R560 B17 R560 B9 R560 B1 R560
B10 R561 B2 R561 B26 R560 B18 R560 B10 R560 B2 R560
B11 R561 B3 R561 B27 R560 B19 R560 B11 R560 B3 R560
B12 R561 B4 R561 B28 R560 B20 R560 B12 R560 B4 R560
B13 R561 B29 R560 B21 R560 B13 R560 B5 R560
B14 R561 B30 R560 B22 R560 B14 R560 B6 R560
B15 R561 B31 R560 B23 R560 B15 R560 B7 R560
B5 R561 B19 R561 B18 R561 B17 R561 B16 R561
B6 R561 B30 R561 B25 R561 B26 R561 B24 R561
B7 R561 B31 R561 B29 R561 B27 R561 B28 R561
B7 R562 B22 R561 B21 R561 B20 R561
B8 R562 B9 R562 B10 R562 B11 R562 B12 R562
B16 R562 B17 R562 B18 R562 B19 R562 B20 R562
B24 R562 B25 R562 B26 R562 B27 R562 B28 R562
JOG
10000 1000 100 10 1 1 10 100
SPINDLE
B15 R563 B13 R562 B14 R562
B23 R562 SPEED % B22 R562
B7 R563 B29 R562 B30 R562
FEED %
0 2 4 10 20 30 40 50 60 70 80 90 100 110 120
B15 R562
B31 R562


FAGOR
ZERO
F1 F2 PCALL F3 F4 F5 F6 F7
ISO
LEVEL CYCLE
A B C D E F
G H I J K L
M N Ñ O P Q
R S T U V W
X Y Z SP CAPS SHIFT
ENTER = 7 8 9
HELP / 4 5 6
RECALL * 1 2 3
RESET + − 0 .
ESC CLEAR
MAIN MENU INS
P.PROG GRAPHICS SINGLE
X+ Y+ Z+ 4+ 5+
6− 6+
X− Y− Z− 4− 5−
JOG
10000 1000 100 10 1 1 10 100
SPINDLE
+
SPEED %
−
FEED %
0 2 4 10 20 30 40 50 60 70 80 90 100 110 120

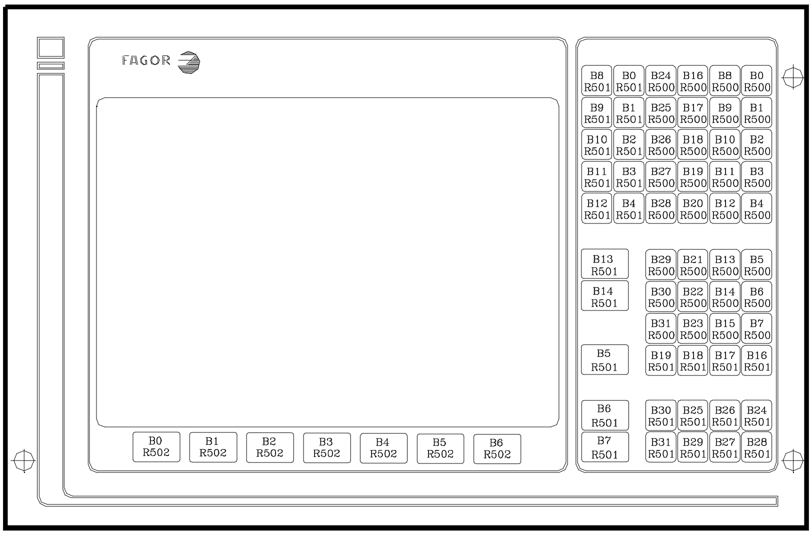
MCO/TCO operator panel

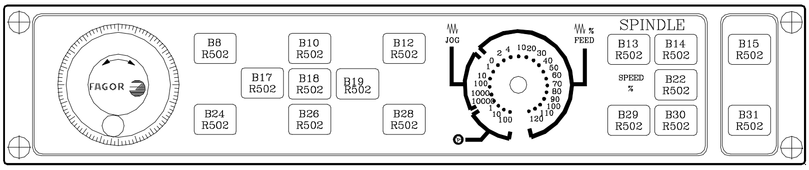


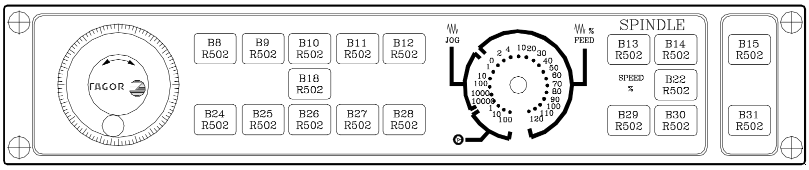


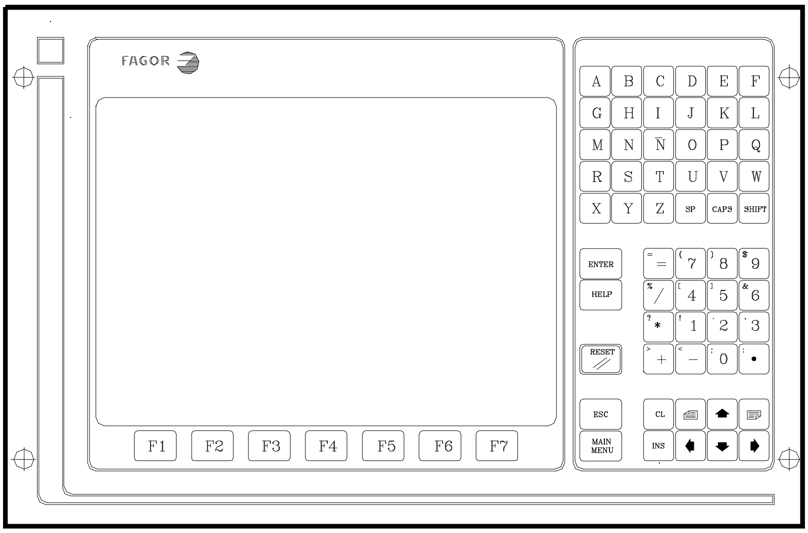
F

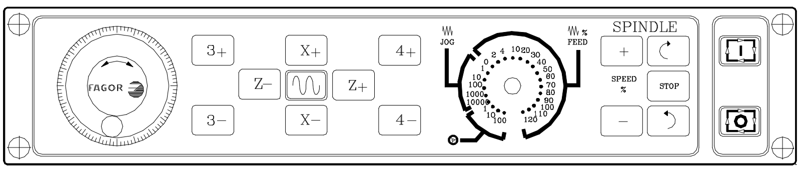
KEYS INHIBITING CODES

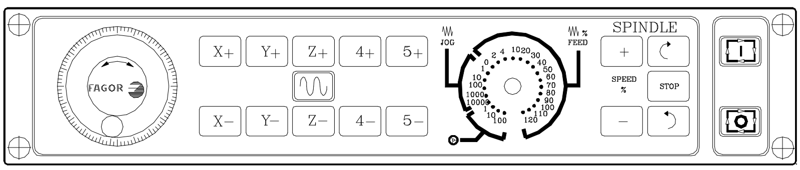




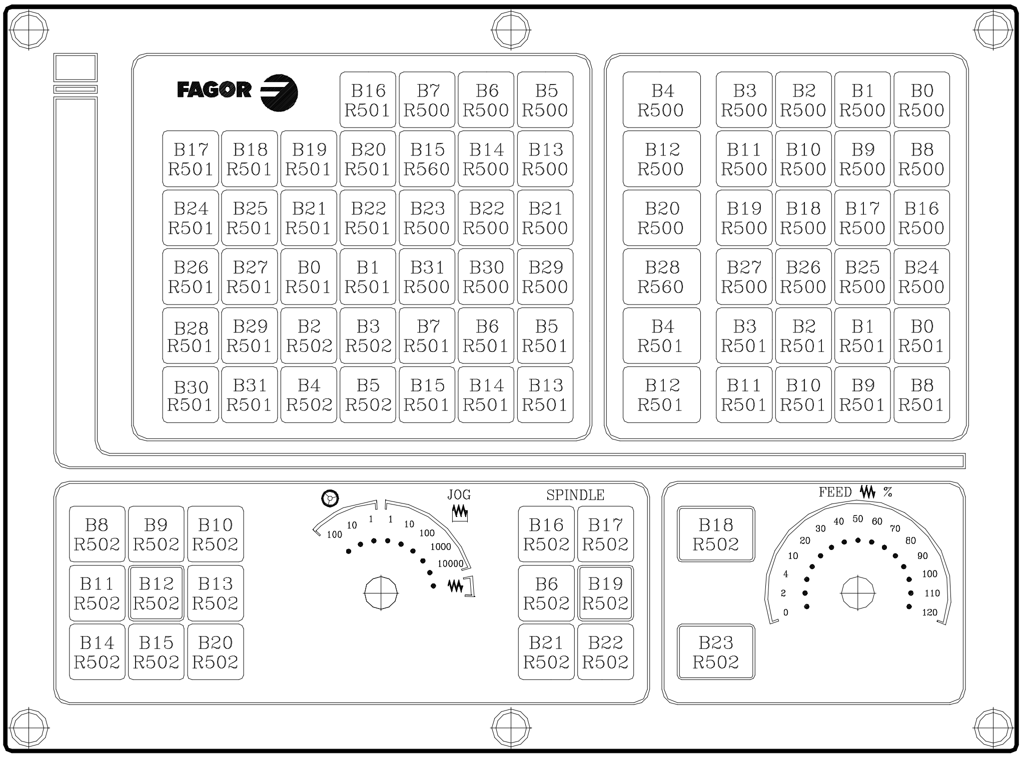


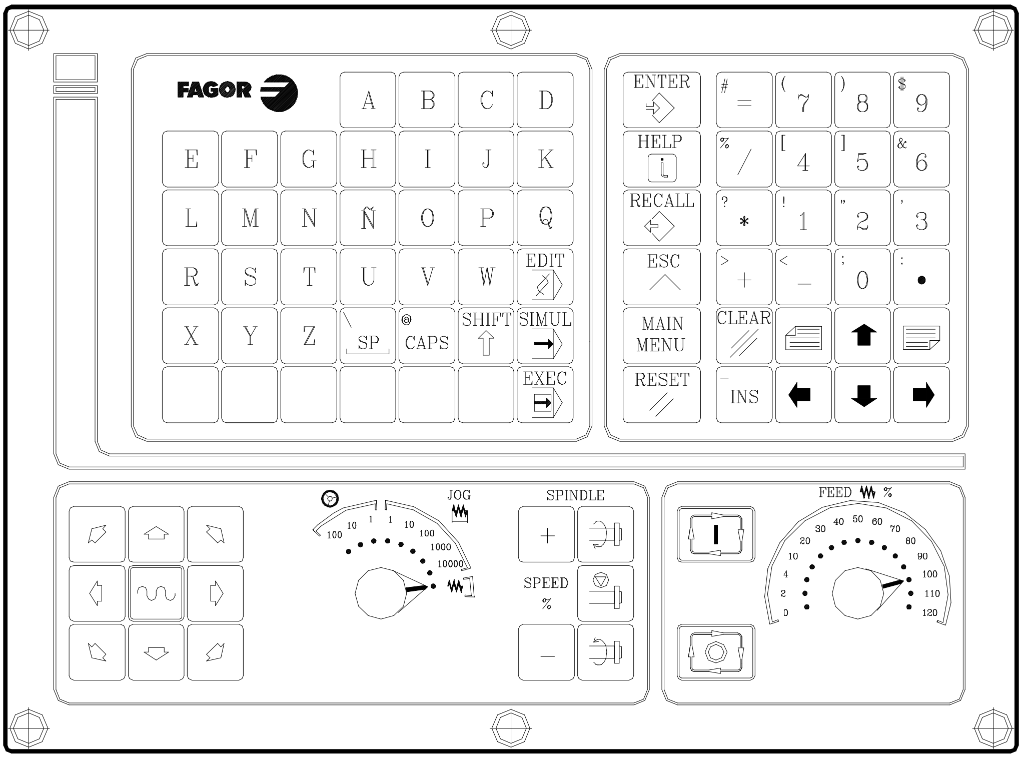




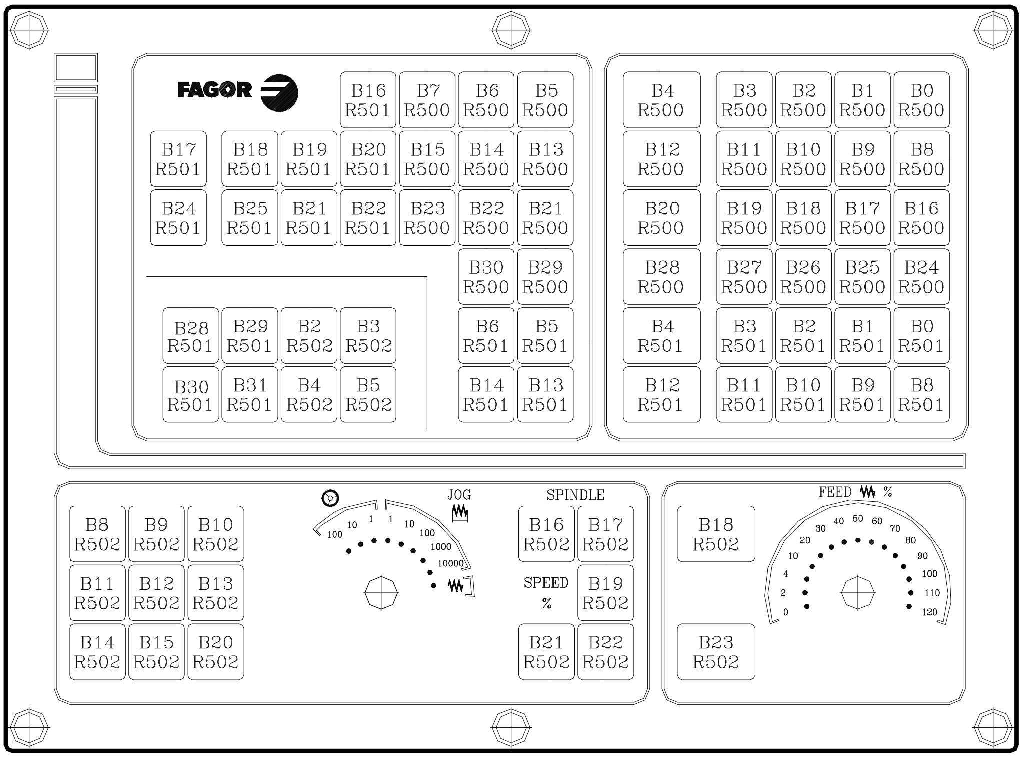


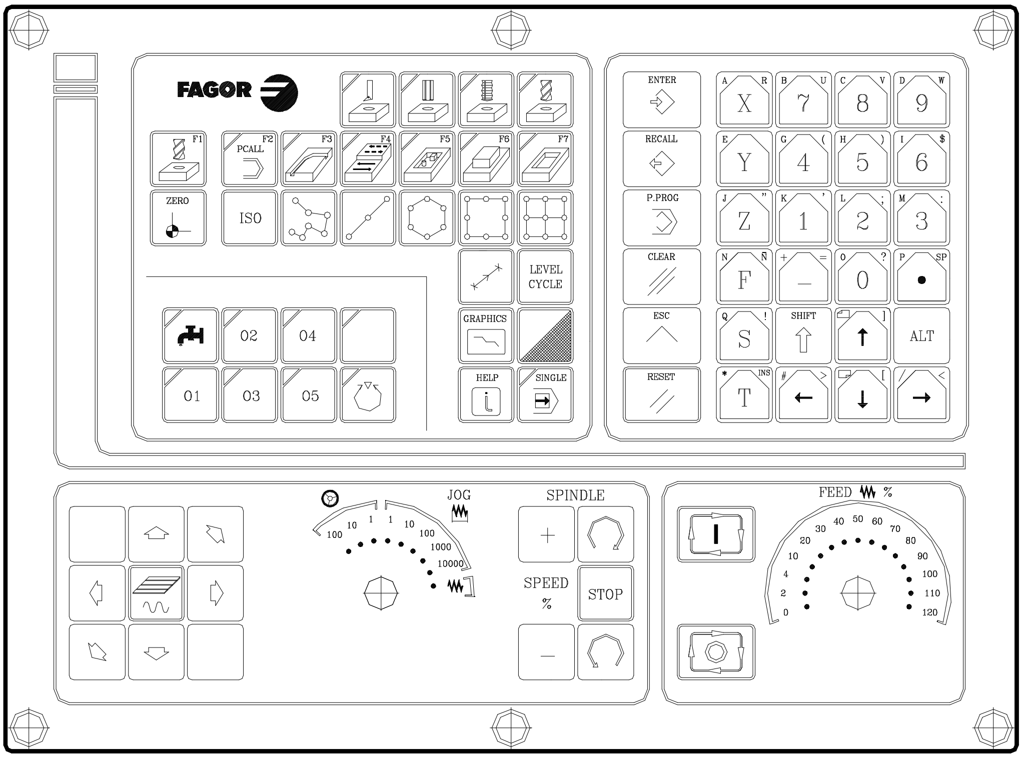
Alphanumeric operator panel (M model)

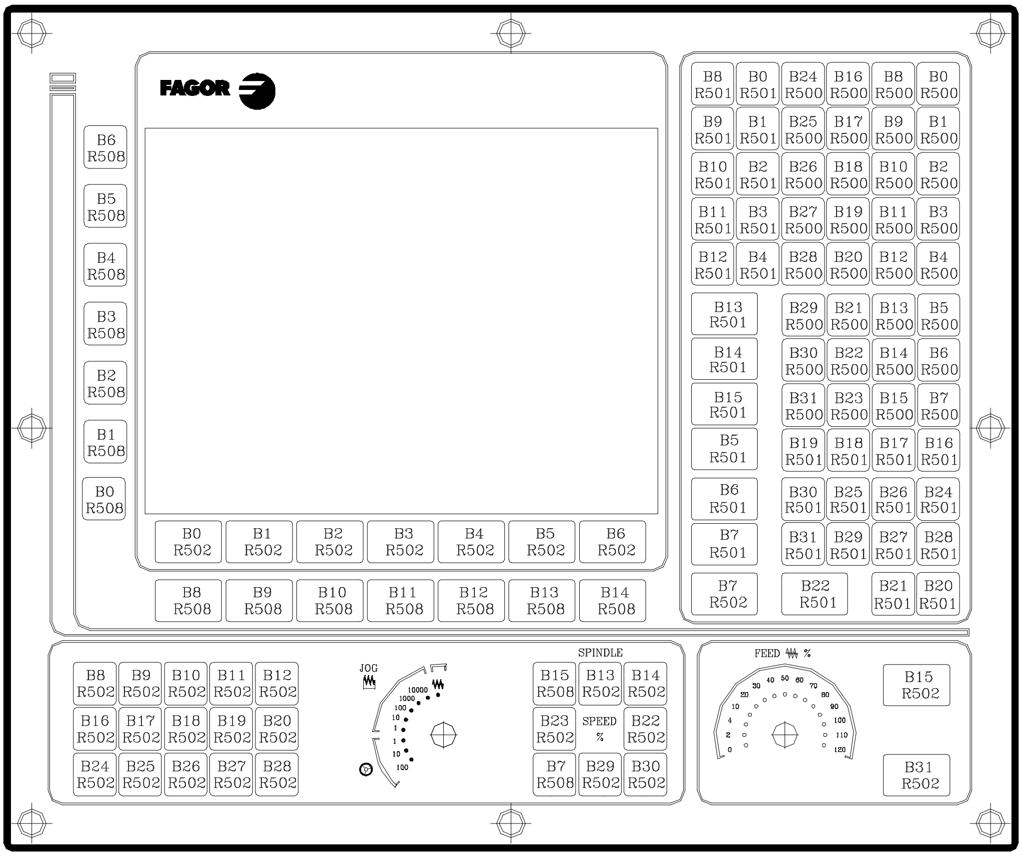


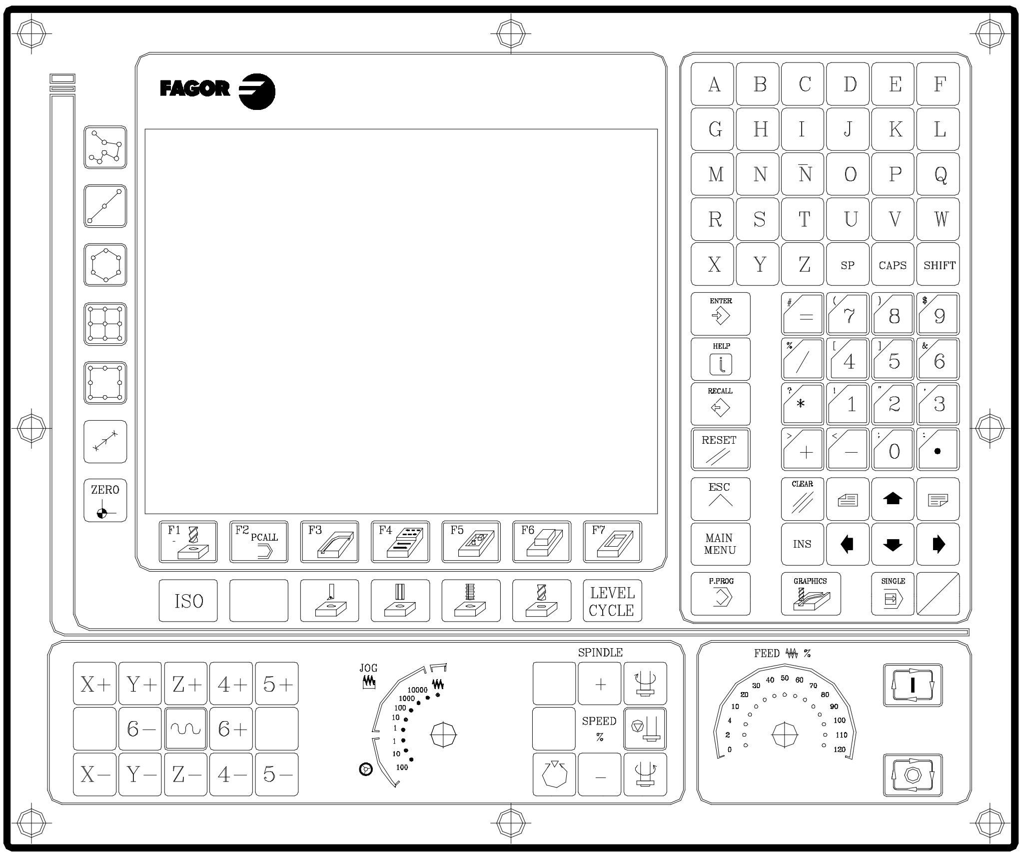


MC operator panel

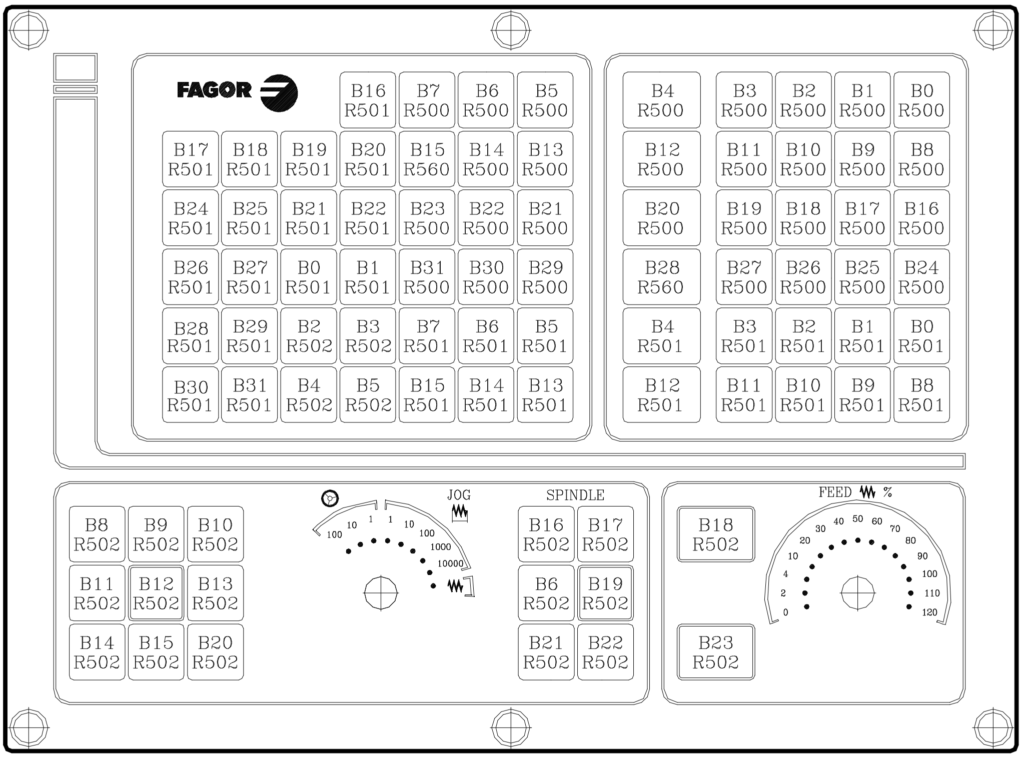


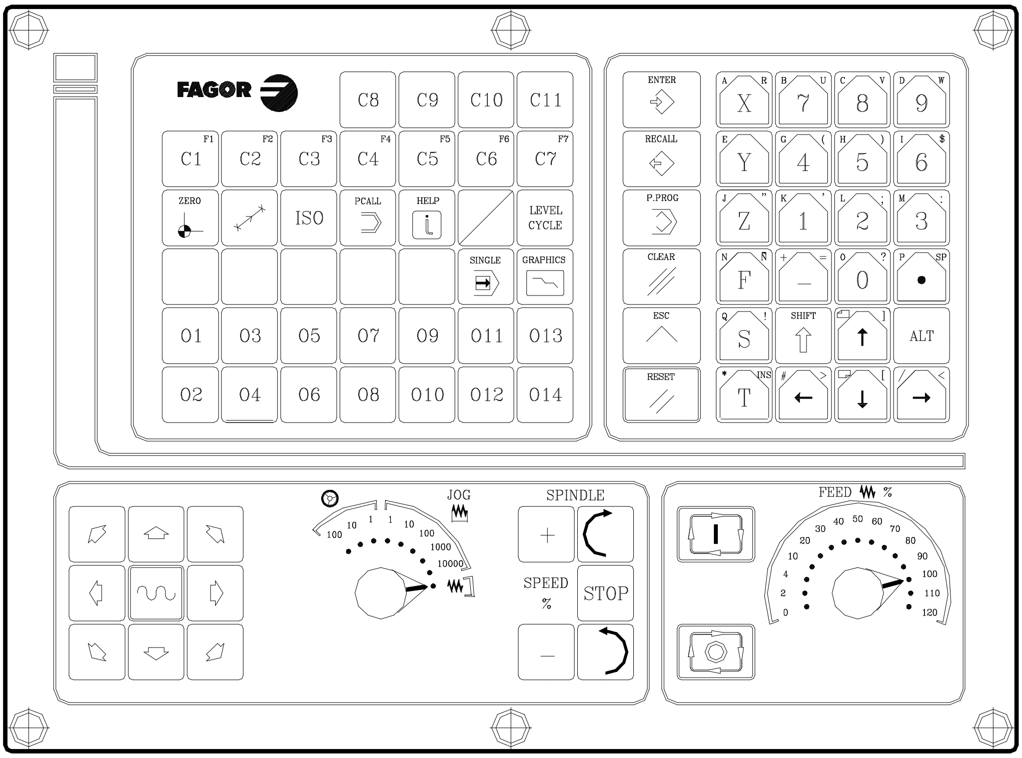



FAGOR
B6 R508
B5 R508
B4 R508
B3 R508
B2 R508
B1 R508
B0 R508
B0 R502 B1 R502 B2 R502 B3 R502 B4 R502 B5 R502 B6 R502
B8 R508 B9 R508 B10 R508 B11 R508 B12 R508 B13 R508 B14 R508
B8 R501 B0 R501 B24 R500 B16 R500 B8 R500 B0 R500
B9 R501 B1 R501 B25 R500 B17 R500 B9 R500 B1 R500
B10 R501 B2 R501 B26 R500 B18 R500 B10 R500 B2 R500
B11 R501 B3 R501 B27 R500 B19 R500 B11 R500 B3 R500
B12 R501 B4 R501 B28 R500 B20 R500 B12 R500 B4 R500
B13 R501 B29 R500 B21 R500 B13 R500 B5 R500
B14 R501 B30 R500 B22 R500 B14 R500 B6 R500
B15 R501 B31 R500 B23 R500 B15 R500 B7 R500
B5 R501 B19 R501 B18 R501 B17 R501 B16 R501
B6 R501 B30 R501 B25 R501 B26 R501 B24 R501
B7 R501 B31 R501 B29 R501 B27 R501 B28 R501
B7 R502 B22 R501 B21 R501 B20 R501
B8 R502 B9 R502 B10 R502 B11 R502 B12 R502
B16 R502 B17 R502 B18 R502 B19 R502 B20 R502
B24 R502 B25 R502 B26 R502 B27 R502 B28 R502
JOG
SPINDLE
B15 R508 B13 R502 B14 R502
B23 R502 SPEED % B22 R502
B7 R508 B29 R502 B30 R502
FEED %
B15 R502
B31 R502


FAGOR
ZERO
F1 F2 PCALL F3 F4 F5 F6 F7
ISO
LEVEL CYCLE
A B C D E F
G H I J K L
M N Ñ O P Q
R S T U V W
X Y Z SP CAPS SHIFT
ENTER = 7 8 9
HELP / 4 5 6
RECALL * 1 2 3
RESET + − 0 •
ESC CLEAR
MAIN MENU INS
P.PROG GRAPHICS SINGLE
X+ Y+ Z+ 4+ 5+
6− 6+
X− Y− Z− 4− 5−
JOG
SPINDLE
+
SPEED %
−
FEED %

MCO/TCO operator panel





G PROGRAMMING ASSISTANCE SYSTEM PAGES

These pages can be displayed by means of the high level mnemonic "PAGE". They all belong to the CNC system and are used as help pages for their respective functions.

GLOSSARY HELP

Page 1000 Preparatory functions G00-G09.
Page 1001 Preparatory functions G10-G19.
Page 1002 Preparatory functions G20-G44.
Page 1003 Preparatory functions G53-G59.
Page 1004 Preparatory functions G60-G69.
Page 1005 Preparatory functions G70-079.
Page 1006 Preparatory functions G80-G89.
Page 1007 Preparatory functions G90-G99.
Page 1008 Miscellaneous (auxiliary) functions M.
Page 1009 Miscellaneous M functions with the symbol for next page.
Page 1010 Coincides with 250 of the directory if it exists.
Page 1011 Coincides with 251 of the directory if it exists.
Page 1012 Coincides with 252 of the directory if it exists.
Page 1013 Coincides with 253 of the directory if it exists.
Page 1014 Coincides with 254 of the directory if it exists.
Page 1015 Coincides with 255 of the directory if it exists.
Page 1016 High level language listing (from A to G)
Page 1017 High level language listing (from H to N)
Page 1018 High level language listing (from 0 to S)
Page 1019 High level language listing (from T to Z)
Page 1020 High level accessible variables (1st part)
Page 1021 High level accessible variables (2nd part)
Page 1022 High level accessible variables (3rd part)
Page 1023 High level accessible variables (4th part)
Page 1024 High level accessible variables (5th part)
Page 1025 High level accessible variables (6th part)
Page 1026 High level accessible variables (7th part)
Page 1027 High level accessible variables (8th part)
Page 1028 High level accessible variables (9th part)
Page 1029 High level accessible variables (10th part)
Page 1030 High level accessible variables (11th part).
Page 1031 High level accessible variables (12th part).
Page 1032 Arithmetic operators.

SYNTAX ASSISTANCE: ISO LANGUAGE

SYNTAX ASSISTANCE: CNC TABLES

SYNTAX ASSISTANCE: HIGH LEVEL

SYNTAX ASSISTANCE: CANNED CYCLES

H

MAINTENANCE

The accumulated dirt inside the unit may act as a screen preventing the proper dissipation of the heat generated by the internal circuitry which could result in a harmful overheating of the CNC and, consequently, possible malfunctions.

On the other hand, accumulated dirt can sometimes act as an electrical conductor and shortcircuit the internal circuitry, especially under high humidity conditions.

To clean the operator panel and the monitor, a smooth cloth should be used which has been dipped into de-ionized water and /or non abrasive dish-washer soap (liquid, never powder) or 75º alcohol.

Do not use highly compressed air to clean the unit because it could generate electrostatic discharges.

The plastics used on the front panel are resistant to :

1.- Grease and mineral oils
2.- Bases and bleach
3.- Dissolved detergents
4.- Alcohol

To check the fuses, first disconnect the power to the CNC.
If the CNC does not turn on when flipping the power switch, check that the fuses are OK and that they are the right ones

Avoid solvents
The action of solvents such as Chlorine hydrocarbons , Benzole , Esters and Ether can damage the plastics used to make the unit's front panel.

Do not open this unit.
Only personnel authorized by Fagor Automation may open this module.

Do not handle the connectors with the unit connected to main AC power.
Before handling these connectors, make sure that the unit is not connected to main AC power.

Note :
Fagor Automation shall not be held responsible for any material or physical damage derived from the violation of these basic safety requirements.

Error solving Manual
(M model)

Ref 0204-ing

FAGOR

INDEX

PROGRAMMING ERRORS

0001 'Línea vacía.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When trying to enter into a program or execute an empty block or containing the label (block number).
2. Within the «Irregular pocket canned cycle with islands (G66)», when parameter "S" (beginning of the profile) is greater than parameter "E" (end of profile).

Solution The solution for each cause is:

1. The CNC cannot enter into the program or execute an empty line. To enter an empty line in the program, use the «;» symbol at the beginning of that block. The CNC will ignore the rest of the block.
2. The value of parameter "S" (block where the profile definition begins) must be lower than the value of parameter "E" (block where the profile definition ends).

0002 'Improper data'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When editing an axis coordinate after the cutting conditions (F, S, T or D) or the «M» functions.
2. When the marks of the block skip (conditional block /1, /2 or /3) are not at the beginning of the block.
3. When programming a block number greater than 9999 while programming in ISO code.
4. When trying to define the coordinates of the machining starting point in the finishing operation (G68) of the «Irregular pocket canned cycle».
5. While programming in high-level, the value of the RPT instruction exceeds 9999.

Solution The solution for each cause is:

1. Remember the programming order
2. Remember the programming order
   - Block skip (conditional block /1, /2 or /3).
   - Label (N).
   - «G» functions.
   - Axis coordinates. (X, Y, Z…).
   - Machining conditions (F, S, T, D).
   - «M» functions.
3. Correct the syntax of the block. Program the labels between 0 and 9999
4. No point can be programmed within the definition of the finishing cycle (G68) for the «Irregular pocket canned cycle». The CNC selects the point where it will start machining. The programming format is: G68 B...L...Q...I...R...K...V... And then the cutting conditions.
5. Correct the syntax of the block. Program a number of repetitions between 0 and 9999

0003 'Improper data order.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The machining conditions or the tool data have been programmed in the wrong order.

Solution Remember that the programming order is… F...S...T...D......

All the data need not be programmed.

0004 'No more information allowed in the block.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When editing a «G» function after an axis coordinate.

2. When trying to edit some data after a «G» function (or after its associated parameters) which must go alone in the block (or which only admits its own associated data).
3. When assigning a numeric value to a parameter that does not need it.

Solution The solution for each cause is:

1. Remember the programming order
   - Block skip (conditional block /1, /2 or /3).
   - Label (N).
   - «G» functions.
   - Axis coordinates. (X, Y, Z…).
   - Machining conditions (F, S, T, D).
   - «M» functions.
2. There are some «G» functions which carry associated data in the block. Maybe, this type of functions do not let program other type of information after their associated parameters. On the other hand, neither machining conditions, (F, S), tool data (T, D) nor «M» functions may be programmed.
3. There are some «G» functions having certain parameters associated to them which do not need to be defined with values.

0005 'Repeated information'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The same data has been entered twice in a block.

Solution Correct the syntax of the block. The same data cannot be defined twice in a block.

0006 'Improper data format'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While defining the parameters of a machining canned cycle, a negative value has been assigned to a parameter which only admits positive values.

Solution Verify the format of the canned cycle. In some canned cycles, there are parameters which only accept positive values.

0007 'Incompatible G functions.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

When programming in the same block two «G» functions which are incompatible with each other.

When trying to define a canned cycle in a block containing a nonlinear movement (G02, G03, G08, G09, G33).

Solution The solution for each cause is:

1. There are groups of «G» functions which cannot go together in the block because they involve actions incompatible with each other. For example:
   G01/G02: Linear and circular interpolation
   G41/G42: Left-hand or right-hand tool radius compensation.
   This type of functions must be programmed in different blocks.
2. A canned cycle must be defined in a block containing a linear movement. In other words, to define a cycle, a "G00" or a "G01" must be active. Nonlinear movements (G02, G03, G08 and G09) may be defined in the blocks following the profile definition.

0008 'Nonexistent G function'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A nonexistent «G» function has been programmed.

Solution Check the syntax of the block and verify that a different «G» function is not being edited by mistake.

0009 'No more G functions allowed'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A «G» function has been programmed after the machining conditions or after the tool data.

Solution Remember that the programming order is:

- Block skip (conditional block /1, /2 or /3).
- Label (N).
- «G» functions.
- Axis coordinates. (X, Y, Z…).
- Machining conditions (F, S, T, D).
- «M» functions.

0010 'No more M functions allowed'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause More than 7 «M» functions have been programmed in a block.

Solution The CNC does not let program more than 7 «M» functions in a block. To execute any other functions, write them in a separate block. The «M» functions may go alone in a block.

0011 'This G or M function must be alone.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The block contains either a «G» or an «M» function that must go alone in the block.

Solution Write it alone in the block.

0012 'Program F, S, T, D before the M functions.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A machining condition (F, S) or tool data (T, D) has been programmed after the «M» functions.

Solution Remember that the programming order is:
… F...S...T...D...M...

Up to 7 «M» functions may be programmed .

All the data need not be programmed.

0013 'Program G30 D +/-359.9999'

No explanation required

0014 'Do not program labels by parameters.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A label (block number) has been defined with a parameter.

Solution Programming the block number is optional, but it cannot be defined with a parameter It can only be defined with a number between 0 and 9999.

0015 'Number of repetitions not possible.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A repetition has been programmed wrong or the block does not admit repetitions.

Solution High level instructions do not admit a number of repetitions at the end of the block. To do a repetition, assign to the block to be repeated a label (block number) and use the RPT instruction.

0016 'Program: G15 axis.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the function «Longitudinal axis selection (G15)» the parameter for the axis has not been programmed.

Solution Check the syntax of the block. The definition of the "G15" function requires the name of the new longitudinal axis.

0017 'Program: G16 axis-axis.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the function «Main plane selection by two axes (G16)» one of the two parameters for the axes has not been programmed.

Solution Check the syntax of the block. The definition of the "G16" function requires the name of the axes defining the new work plane.

0018 'Program: G22 K(1/2/3/4) S(0/1/2).'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the function «Enable/Disable work zones (G22)» the type of enable or disable of the work zone has not been defined or it has been assigned the wrong value.

Solution The parameter for enabling or disabling the work zones "S" must always be programmed and it may take the following values.
- S=0: The work zone is disabled.
- S=1: It is enabled as a no-entry zone.
- S=2: It is enabled as a no-exit zone.

0019 'Program: work zone K1, K2, K3 or K4.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. A "G20", "G21" or "G22" function has been programmed without defining the work zone K1, K2, K3 or K4.
2. The programmed work zone is smaller than 0 or greater than 4.

Solution The solution for each cause is:

1. The programming format for functions "G20", "G21" and "G22" is:

   G20 K...X...C±5.5 Definition of lower work zone limits
   G21 K...X...C±5.5 Definition of upper work zone limits.
   G22 K...S... Enable/disable work zones.

   Where:

   K : Is the work zone.
   X...C Are the axes where the limits are defined.
   S Is the type of work zone enable.

2. The "K" work zone may only have the values of K1, K2, K3 or K4.

0020 'Program G36-G39 with R+5.5.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the "G36" or "G39" function, the "R" parameter has not been programmed or it has been assigned a negative value.

Solution To define "G36" or "G39", parameter "R" must also be defined and with a positive value).

G36 R= Rounding radius.
G39 R= Distance between the end of the programmed path and the point to be chamfered.

0021 'Program: G72 S5.5 or axis (axes).'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When programming a general scaling factor (G72) without the scaling factor to apply.
2. When programming a particular scaling factor (G72) to several axes, but the axes have been defined in the wrong order.

Solution Remember that the programming format for this function is:

G72 S5.5" When applying a general scaling factor (to all axes).
G72 X…C5.5" When applying a particular scaling factor to one or several axes.

0022 'Program: G73 Q (angle) I J (center).'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the «Pattern rotation (G73)» function have been programmed wrong. The causes may be:

1. The rotation angle has not been defined.
2. Only one of the rotation center coordinates has been defined.
3. The rotation center coordinates have been defined in the wrong order.

Solution The programming format for this function is:

G73 Q (angle) [I J] (center)

The "Q" value must always be programmed.

The "I", "J" values are optional, but if programmed, both must be programmed.

0023 'Block incompatible when defining a profile.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the set of blocks defining a pocket profile, there is a block containing a «G» function that cannot be part of the profile definition.

Solution The "G" functions available in the profile definition of a pocket (2D/3D) are:

G00: Beginning of the profile.
G01: Linear interpolation.
G02/G03: Clockwise/counterclockwise interpolation.
G06: Circle center in absolute coordinates.
G08: Arc tangent to previous path.
G09: Three point arc.
G36: Controlled corner rounding
G39: Chamfer.
G53: Programming with respect to home.
G70/G71: Inch/metric programming.
G90/G91: Programming in absolute/incremental coordinates.
G93: Polar origin preset.

And also, in the 3D pocket profile:

G16: Main plane selection by two axes.
G17: Main plane X-Y and longitudinal Z.
G18: Main plane Z-X and longitudinal Y.
G19: Main plane Y-Z and longitudinal X.

0024 'High level blocks not allowed when defining a profile.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause Within the set of blocks defining a pocket profile, a high level block has been programmed.

Solution The pocket profile must be defined in ISO code. High level instructions are not allowed (GOTO, MSG, RPT ...).

0025 'Program: G77 axes (2 to 6) or G77 S.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the "axis slaving function (G77)» the parameters for the axes are missing or in "spindle synchronization (G77S) functions the "S" parameter is missing.

Solution In the "axis slaving" function, program at least two axes and in the "spindle synchronization" function, always program the "S" parameter.

0026 'Program: G93 IJ.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the «Polar origin preset (G93)» function, some of the parameters for the new polar origin have not been programmed.

Solution Remember that the programming format for this function is:

G93 I...J...

The "I", "J" values are optional, but if programmed, both must be programmed and they indicate the new polar origin.

0027 'G49 T X Y Z S, X Y Z A B C ', or, ' X Y Z Q R S.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the «Incline plane definition (G49)» function, a parameter has been programmed twice.

Solution Check the syntax of the block. The programming formats are:

T X Y Z S   X Y Z A B C   X Y Z Q R S

0028 'G2 or G3 not allowed when programming a canned cycle.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A canned cycle has been attempted to execute while the "G02", "G03" or "G33" functions were active.

Solution To execute a canned cycle, "G00" or "G01" must be active. A "G02" or "G03" function may be programmed previously in the program history. Check that these functions are not active when the canned cycle is defined.

0029 'G60: [A] /X I K/(2) [P Q R S T U V].'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the «Multiple machining in a straight line (G60)» have been programmed wrong. These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.
3. Some data might be superfluous.

Solution In this type of machining, two of the following parameters must always be programmed:

X Path length.
I Step between machining operations.
K : Number of machining operations.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0030 'G61-2: [A B] /X I J/(2) Y J D (2)/ [P Q R S T U V].'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the «Multiple machining in a parallelogram pattern (G61)» or «Multiple machining in a grid pattern (G62)» cycle have been programmed wrong. These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.
3. Some data might be superfluous.

Solution This type of machining requires the programming of two parameters of each group (X, I, K) and (Y, J, D).

X/Y Path length.
I/J Step between machining operations.
K /D Number of machining operations.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0031 'G63: X Y /I K/(1) [C P][P Q R S T U V].'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the «Multiple machining in a circle (G63)» cycle have been programmed wrong. These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.
3. Some data might be superfluous.

Solution This type of machining requires the programming of:

X/Y Distance from the center to the first hole.

And one of the following data:

I Angular step between machining operations.
K : Number of machining operations.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0032 'G64: X Y /I K/(1) [C P][P Q R S T U V.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the «multiple machining in an arc (G64)» cycle have been programmed wrong. These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.
3. Some data might be superfluous.

Solution This type of machining requires the programming of:

X/Y Distance from the center to the first hole.
B Total angular travel.

And one of the following data:

I Angular step between machining operations.
K : Number of machining operations.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0033 'G65: X Y /A I/(1) [C P].'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the «Multiple machining programmed by means of an arc chord (G65)» cycle have been programmed wrong. These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.
3. Some data might be superfluous.

Solution This type of machining requires the programming of:

X/Y Distance from the center to the first hole.

And one of the following data:

A Angle of the matrix of the chord with the abscissa axis (in degrees).
I Chord length.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0034 'G66: [D H][R I][C J][F K] S E [Q].'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the «Irregular pocket canned cycle with islands (G66)» have been programmed wrong. These may be the probable causes:

1. A parameter has been programmed which does not match the calling format.
2. Some mandatory parameter is missing.
3. The parameters of the cycle have not been edited in the correct order.

Solution This machining cycle requires the programming of :

S First block of the description of the geometry of the profiles making up the pocket.
E End block of the description of the geometry of the profiles making up the pocket.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. Also, the following parameters cannot be defined:

H if D has not been defined.
I if R has not been defined.
J if C has not been defined.
K : if F has not been defined.

The (X...C) position where the machining takes place cannot be programmed either.

0035 'G67: [A] B [C] [I] [R] [K] [V].'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the roughing (2D/3D pocket) or semi-finishing (3D pocket) operation have been programmed wrong in the «Irregular pocket canned cycle with islands». These may be the probable causes:

1. A parameter has been programmed which does not match the calling format.
2. Some mandatory parameter is missing.
3. The parameters of the cycle have not been edited in the correct order.

Solution This machining cycle requires the programming of :

Roughing operation (2D or 3D pockets)

B Cutting pass
I Total pocket depth.
R Coordinate of the reference plane.

Semi-finishing operation (3D pockets)

B Cutting pass
I Total pocket depth (if no roughing operation has been defined).
R Coordinate of the reference plane (if no roughing operation has been defined).

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place cannot be programmed in this cycle.

0036 'G68: [B] [L] [Q] [J] [I] [R] [K].'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters for the finishing operation (2D/3D pocket) have been programmed wrong in the «Irregular pocket cycle with islands. These may be the probable causes:

1. A parameter has been programmed which does not match the calling format.
2. Some mandatory parameter is missing.
3. The parameters of the cycle have not been edited in the correct order.

Solution This machining cycle requires the programming of :

2D pockets

- B Cutting pass (if no roughing operation has been defined).
- I Total pocket depth (if no roughing operation has been defined).
- R Coordinate of the reference plane (if no roughing operation has been defined).

3D pockets

- B Cutting pass
- I Total pocket depth (if no roughing or semi-finishing operation has been defined).
- R Coordinate of the reference plane (if no roughing or semi-finishing operation has been defined).

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place cannot be programmed in this cycle.

0037 'G69: I B [C D H J K L R].'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the «Deep hole drilling cycle with variable peck (G69)». These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.

Solution This type of machining requires the programming of:

- I Machining depth.
- B Drilling peck.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0038 'G81-84-85-86-89: I [K].'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters have been programmed wrong in the following cycles: drilling (G81), tapping (G84), reaming (G85) or boring (G86/G89). This could be because parameter "I : Machining depth" is missing in the canned cycle being edited.

Solution This type of machining requires the programming of:

- I Machining depth.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0039 'G82: I K.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters have been programmed wrong in the «Drilling cycle with dwell (G82)». This could be because some parameter is missing.

Solution Both parameters must be programmed in this cycle:

- I Machining depth.
- K : Dwell at the bottom.

To program a drilling operation without dwell at the bottom, use function G81.

The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0040 'G83: I J.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters have been programmed wrong in the «Deep hole drilling with constant peck (G83)». This could be because some parameter is missing.

Solution This type of machining requires the programming of:

I Machining depth.
J Number of pecks.

The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0041 'G87: I J K B [C] [D] [H] [L] [V].'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters have been programmed wrong in the «Rectangular pocket canned cycle (G87)». These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.

Solution This type of machining requires the programming of:

I Pocket depth.
J Distance from the center to the edge of the pocket along the abscissa axis.
K : Distance from the center to the edge of the pocket along the ordinate axis.
B Defines the machining pass along the longitudinal axis.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0042 'G88: I J B [C] [D] [H] [L] [V].'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters have been programmed wrong in the «Circular pocket canned cycle (G88)». These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.

Solution This type of machining requires the programming of:

I Pocket depth.
J Pocket radius.
B Defines the machining pass along the longitudinal axis.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0043 'Incomplete Coordinates.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. During simulation or execution, when trying to make a movement defined with only one coordinate of the end point or without defining the arc radius while a «circular interpolation (G02/G03) is active.
2. During editing, when editing a circular movement (G02/G03) by defining only one coordinate of the end point or not defining the arc radius.

Solution The solution for each cause is:

1. A "G02" or "G03" function may be programmed previously in the program history. In this case, to make a move, both coordinates of the end point and the arc radius must be defined. To make a linear movement, program "G01".
2. To make a circular movement (G02/G03), both coordinates of the end point and the arc radius must be programmed.

0044 'Incorrect Coordinates.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. An attempt has been made to execute a block syntactically incorrect (G1 X20K-15)
2. The “I” parameter is missing in the definition of a machining canned cycle (G81-G89) Machining depth.

Solution The solution for each cause is:

1. Correct the syntax of the block.
2. This type of machining requires the programming of:

   I Machining depth.

   The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0045 ‘Polar coordinates not allowed.’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause When «Programming with respect to home (G53)», the end point has been defined in polar or cylindrical coordinates or in Cartesian coordinates with an angle.

Solution When programming with respect to home, only Cartesian coordinates may be programmed.

0046 ‘Axis does not exist.’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When editing a block whose execution involves the movement of a nonexistent axis.
2. Sometimes, this error comes up while editing a block that is missing a parameter of the «G» function. This is because some parameters with an axis name have a special meaning inside certain «G» functions. For example: G69 I...B....
   In this case, parameter “B” has a special meaning after “I“. If the “I” parameter is left out, the CNC assumes “B” as the position where the machining takes place on that axis. If that axis does not exist, it will issue this error message.

Solution The solution for each cause is:

1. Check that the axis name being edited is correct.
2. Check the block syntax and make sure that all the mandatory parameters have been programmed.

0047 ‘Program axes.’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause No axis has been programmed in a function requiring an axis.

Solution Some instructions require the programming of axes (REPOS, G14, G20, G21…).

0048 ‘Incorrect order of axes.’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The axis coordinates have not been programmed in the correct order or an axis has been programmed twice in the same block.

Solution Remember that the correct programming order for the axes is:

X...Y...Z...U...V...W...A...B...C...

All axes need not be programmed:

0049 ‘Point incompatible with active plane.’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When trying to do a circular interpolation, the end point is not in the active plane.
2. When trying to do a tangential exit in a path that is not in the active plane.

Solution The solution for each cause is:

1. Maybe a plane has been defined with “G16”, “G17”, “G18” or “G19”. In this case, circular interpolations can only be carried out on the main axes defining that plane. To define a circular interpolation in another plane, it must be defined beforehand.

2. Maybe a plane has been defined with “G16”, “G17”, “G18” or “G19”. In this case, corner rounding, chamfers and tangential entries/exits can only be carried out on the main axes defining that plane. To do it in another plane, it must be defined beforehand.

0050 ‘Program positions on active plane.’

No explanation required

0051 ‘Perpendicular axis included in active plane.’

No explanation required

0052 ‘Center of circle programmed incorrectly.’

No explanation required

0053 ‘Program pitch.’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the «Electronic threading cycle (G33)» the parameter for the thread pitch is missing.

Solution Remember that the programming format for this function is:

G33 X...C...L...

Where: “L” is the thread pitch.

0054 ‘Pitch programmed incorrectly.’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A helical interpolation has been programmed with the wrong or negative pitch.

Solution Remember that the programming format is:

G02/G03 X...Y...I...J...Z...K...

Where: “K” is the helical pitch (always positive value).

0055 ‘Positioning axes or Hirth axes not allowed’

No explanation required

0056 ‘The axis is already slaved.’

No explanation required

0057 ‘Do not program a slaved axis.’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When trying to move an axis alone while being slaved to another one.
2. When trying to slave an axis that is already slaved using the G77 function «Electronic axis slaving».

Solution The solution for each cause is:

1. A slaved axis cannot be moved separately. To move a slaved axis, its master axis must be moved. Both axes will move at the same time.
   Example: If the Y axis is slaved to the X axis, an X axis move must be programmed in order to move the Y axis (together with the X axis).
   To unslave the axes, program “G78”.
2. An axis cannot be slaved to two different axes at the same time. To unslave the axes, program “G78”.

0058 ‘Do not program a GANTRY axis.’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When trying to move an axis alone while being slaved to another one as a GANTRY axis
2. When defining an operation on a GANTRY axis. (Definition of work zone limits, planes, etc.).

Solution The solution for each cause is:

1. A GANTRY axis cannot be moved separately. To move a GANTRY axis, its associated axis must be moved. Both axes will move at the same time.
   Example: If the Y axis is a GANTRY axis associated with the X axis, an X axis move must be programmed in order to move the Y axis (together with the X axis).
   GANTRY axes are defined by machine parameter.
2. The axes defined as GANTRY cannot be used in the definition of operations or movements. These operations are defined with the main axis that the GANTRY axis is associated with.

0059 'Eje HIRTH: program only integer values.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A rotation of a HIRTH axis has been programmed with a decimal value.

Solution HIRTH axes do not accept decimal angular values. They must be full degrees.

0060.'Invalid action.'

No explanation required

0061 'ELSE not associated with IF.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. While editing in High level language, when editing the "ELSE" instruction without having previously programmed an "IF".
2. When programming in high level language, an "IF" is programmed without associating it with any action after the condition.

Solution Remember that the programming formats for this instruction are:

(IF (condition) <action1>)
(IF (condition) <action1> ELSE <action2>)

If the condition is true, it executes the < action1>, otherwise, it executes < action2>.

0062 'Program label N(0-9999).'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, a block number out of the 0-9999 range has been programmed in the "RPT" or "GOTO" instruction.

Solution Remember that the programming format of these instructions is:

(RPT N(block number), N(block number))
(GOTO N(block number))

The block number (label) must be between 0 and 9999.

0063 'Program subroutine number 1 thru 9999.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, a subroutine number out of the 0-9999 range has been programmed in the "SUB" instruction.

Solution Remember that the programming format for this instruction is:

(SUB (integer))

The subroutine number must be between 0 and 9999.

0064 'Repeated subroutine.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause There has been an attempt to define a subroutine already existing in another program of the memory.

Solution In the CNC memory, there could not be more than one subroutine with the same identifying number even if they are contained in different programs.

0065 'The main program cannot have a subroutine.'

Detection In execution or while executing programs transmitted via DNC.

Cause The various causes might be:

1. An attempt has been made to define a subroutine in the MDI execution mode.
2. A subroutine has been defined in the main program.

Solution The solution for each cause is:

1. Subroutines cannot be defined from the «MDI execution» option of the menu.
2. Subroutines must be defined after the main program or in a separate program. They cannot be defined before or inside the main program.

0066 'Expecting a message.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level, the "MSG" or "ERROR" instruction has been edited but without the message to be displayed.

Solution Remember that the programming format of these instructions is:

(MSG "message")
(ERROR integer, "error message")

Although it can also be programmed as follows:

(ERROR integer)
(ERROR "error message")

0067 'OPEN is missing.'

Detection In execution or while executing programs transmitted via DNC.

Cause While programming in high level, a "WRITE" instruction has been edited, but the OPEN instruction has not been written previously to tell it where that instruction has to be executed.

Solution The "OPEN" instruction must be edited before the "WRITE" instruction to «tell» the CNC where (in which program) it must execute the "WRITE" instruction.

0068 'Expecting a program number.'

No explanation required

0069 'Program does not exist.'

Detection In execution or while executing programs transmitted via DNC.

Cause Inside the «Irregular pocket with islands cycle (G66)», it has been programmed that the profiles defining the irregular pocket are in another program (parameter "Q"), but that program does not exist.

Solution Parameter "Q" defines which program contains the definition of the profiles that, in turn, define the irregular pocket with islands. If this parameter is programmed, that program number must exist and it must contain the labels defined by parameters "S" and "E".

0070 'Program already exists.'

Detection In execution or while executing programs transmitted via DNC.

Cause This error comes up during execution when using the "OPEN" instruction (While programming in high level language) to create an already existing program.

Solution Change the program number or use parameters A/D in the "OPEN" instruction:

(OPEN P.........,A/D,…)

Where:

A: Appends new blocks after the existing ones.
D: Deletes the existing program and it opens it as a new one.

0071 'Expecting a parameter'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When defining the function «Modification of canned cycle parameters (G79)», the parameter to be modified has not been indicated.
2. While editing the machine parameter table, the wrong parameter number has been entered (maybe the "P" character is missing) or another action is being carried out (moving around in the table) before quitting the table editing mode.

Solution The solution for each cause is:

1. To define the "G79" function, the cycle parameter to be modified must be indicated as well as its new value.
2. Enter the parameter number to be edited or press [ESC] to quit this mode.

0072 'Parameter does not exist.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, the "ERROR" instruction has been edited, but the error number to be displayed has been defined either with a local parameter greater than 25 or with a global parameter greater than 299.

Solution The parameters used by the CNC are:

Local: 0-25
Global: 100-299

0073 'Parameter range protected. Cannot be written.'

No explanation required

0074 'Variable not accessible from CNC.'

No explanation required

0075 'Read-only variable.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An attempt has been made to assign a value to a read-only variable.

Solution Read-only variables cannot be assigned any values through programming. However, their values can be assigned to a parameter.

0076 'Write-only variable.'

No explanation required

0077 'Analog output not available.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An attempt has been made to write to an analog output currently being used by the CNC.

Solution The selected analog output may be currently used by an axis or a spindle. Select another analog output between 1 and 8.

0078 'Program channel 0(CNC),1(PLC) or 2(DNC).'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, the "KEYSCR" instruction has been programmed, but the source of the keys is missing.

Solution When programming the "KEYSCR" instruction, the parameter for the source of the keys must always be programmed:

(KEYSCR=0) : CNC keyboard
(KEYSCR=1) : PLC
(KEYSCR=2) : DNC

The CNC only lets modifying the contents of this variable if it is «zero»

0079 'Program error number 0 thru 9999.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, the "ERROR" instruction has been programmed, but the error number to be displayed is missing.

Solution Remember that the programming format for this instruction is:

(ERROR integer, "error message")

Although it can also be programmed as follows:

(ERROR integer)
(ERROR "error message")

0080 'Operator missing.'

No explanation required

0081 'Incorrect expression.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, an expression has been edited with the wrong format.

Solution Correct the syntax of the block.

0082 'Incorrect operation.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. While programming in high level language, the assignment of a value to a parameter is incomplete.
2. While programming in high level language, the call to a subroutine is incomplete.

Solution Correct (complete) the format to assign a value to a parameter or a call to a subroutine.

0083 'Incomplete operation.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. While programming in high level language, the "IF" instruction has been edited without the condition between brackets.
2. While programming in high level language, the "DIGIT" instruction has been edited without assigning a value to some parameter.

Solution The solution for each cause is:

1. Remember that the programming formats for this instruction are:
   (IF (condition) <action1>)
   (IF (condition) <action1> ELSE <action2>)
   If the condition is true, it executes the < action1>, otherwise, it executes < action2>.
2. Correct the syntax of the block. All the parameters defined within the "DIGIT" instruction must have a value assigned to them.

0084 'Expecting "=".'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, a symbol or data has been entered that does not match the syntax of the block.

Solution Enter the "=" symbol in the right place.

0085 'Expecting ")".'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, a symbol or data has been entered that does not match the syntax of the block.

Solution Enter the ")" symbol in the right place.

0086 'Expecting "(".'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, a symbol or data has been entered that does not match the syntax of the block.

Solution Enter the "(" symbol in the right place.

0087 'Expecting ",".'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. While programming in high level language, a symbol or data has been entered that does not match the syntax of the block.
2. While programming in high level language, an ISO-coded instruction has been programmed.
3. While programming in high level language, an operation has been assigned either to a local parameter greater than 25 or to a global parameter greater 299.

Solution The solution for each cause is:

1. Enter the "," symbol in the right place.
2. A block cannot contain high level language instructions and ISO-coded instructions at the same time.

3. The parameters used by the CNC are:

Local:	0-25.
Global:	100-299.

Other parameters out of this range cannot be used in operations.

0088 'Operation limit exceeded.'

No explanation required

0089 'Logarithm of zero or negative number.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An operation has been programmed which involves the calculation of a negative number or a zero.

Solution Only logarithms of numbers greater than zero can be calculated. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0090 'Square root of a negative number.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An operation has been programmed which involves the calculation of the square root of a negative number.

Solution Only the square root of numbers greater than zero can be calculated. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0091 'Division by zero.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An operation has been programmed which involves a division by zero.

Solution Only divisions by numbers other than zero are allowed. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0092 'Base zero with positive exponent.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An operation has been programmed which involves elevating zero to a negative exponent (or zero).

Solution Zero can only be elevated to positive exponents greater than zero. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0093 'Negative base with decimal exponent.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An operation has been programmed which involves elevating a negative number to a decimal exponent.

Solution Negative numbers can only be elevated to integer exponents. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0094 'ASIN/ACOS range exceeded.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An operation has been programmed which involves calculating the arcsine or arccosine of a number out of the ±1 range.

Solution Only the arcsine (ASIN) or arccosine (ACOS) of numbers between ±1 can be calculated. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0095 'Program row number.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While editing a customizing program, a window has been programmed with the "ODW" instruction, but the vertical position of the window on the screen is missing.

Solution The vertical position of the window on the screen is defined by rows (0-25).

0096 'Program column number.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While editing a customizing program, a window has been programmed with the "ODW" instruction, but the horizontal position of the window on the screen is missing.

Solution The horizontal position of the window on the screen is defined by columns (0-79).

0097 'Program another softkey.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While editing a customizing program, the programming format for the "SK" instruction has not been respected.

Solution Correct the syntax of the block. The programming format is:
(SK1=(text 1), SK2=(text 2)…)

If the "," character is entered after a text, the CNC expects the name of another softkey.

0098 'Program softkeys 1 thru 7.'

Detection While executing in the user channel.

Cause In the block syntax, a softkey has been programmed out of the 1 to 7 range.

Solution Only softkeys within the 1 to 7 range can be programmed.

0099 'Program another window.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While editing a customizing program, the programming format for the "DW" instruction has not been respected.

Solution Correct the syntax of the block. The programming format is:
(DW1=(assignment), DW2=(assignment)…)

If the "," character is entered after an assignment, the CNC expects the name of another window.

0100 'Program windows 0 thru 25.'

Detection While executing in the user channel.

Cause In the block syntax, a window has been programmed out of the 0 to 25 range.

Solution Only windows within the 0 to 25 range can be programmed.

0101 'Program rows 0 thru 20.'

Detection While executing in the user channel.

Cause In the block syntax, a row has been programmed out of the 0 to 20 range.

Solution Only rows within the 0 to 20 range can be programmed.

0102 'Program columns 0 thru 79.'

Detection While executing in the user channel.

Cause In the block syntax, a column has been programmed out of the 0 to 79 range.

Solution Only columns within the 0 to 79 range can be programmed.

0103 'Program pages 0 thru 255.'

Detection While executing in the user channel.

Cause In the block syntax, a page has been programmed out of the 0 to 255 range.

Solution Only pages within the 0 to 255 range can be programmed.

0104 'Program INPUT.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, an "IB" instruction has been edited without associating an "INPUT" to it.

Solution Remember that the programming formats for this instruction are:
(IB (expression) = INPUT "text", format)
(IB (expression) = INPUT "text")

0105 'Program inputs 0 thru 25.'

Detection While executing in the user channel.

Cause In the block syntax, an input has been programmed out of the 0 to 25 range.

Solution Only inputs within the 0 to 25 range can be programmed.

0106 'Program numerical format.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, an "IB" instruction has been edited with non-numeric format.

Solution Remember that the programming format for this instruction is:
(IB (expression) = INPUT "text", format)

Where «format» must be a signed number with 6 entire digits and 5 decimals at the most.

If the "," character is entered after the text, the CNC expects the format.

0107 'Do not program formats greater than 6.5 .'

Detection While executing in the user channel.

Cause While programming in high level language, an "IB" instruction has been edited in a format with more than 6 entire digits or more than 5 decimals.

Solution Remember that the programming format for this instruction is:
(IB (expression) = INPUT "text", format)

Where «format» must be a signed number with 6 entire digits and 5 decimals at the most.

0108 'This command can only be executed in the user channel.'

Detection During execution.

Cause An attempt has been made to execute a block containing information that can only be executed through the user channel.

Solution There are specific expressions for customizing programs that can only be executed inside the user program.

0109 'C. User: do not program geometric help, compensation or cycles.'

Detection While executing in the user channel.

Cause An attempt has been made to execute a block containing geometric aide, tool radius/length compensation or machining canned cycles.

Solution Inside a customizing program the following cannot be programmed:
Neither geometric assistance nor movements.
Neither tool radius nor length compensation.
Canned cycles.

0110 'Local parameters not allowed.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause Some functions can only be programmed with global parameters.

Solution Global parameters are the ones included in the 100-299 range.

0111 'Block cannot be executed while running another program'

Detection While executing in MDI mode

Cause An attempt has been made to execute a customizing instruction from MDI mode while the user channel program is running.

Solution Customizing instructions can only be executed through the user channel.

0112 'WBUF can only be executed in user channel while editing'

Detection During normal execution or execution through the user channel.

Cause An attempt has been made to execute the "WBUF" instruction.

Solution The “WBUF” instruction cannot be executed. It can only be used in the editing stage through the user input.

0113 ‘Table limits exceeded.’

Detection While editing tables.

Cause The various causes might be:

1. In the tool offset table, an attempt has been made to define a tool offset with a greater number than allowed by the manufacturer.
2. In the parameter tables, an attempt has been made to define a nonexistent parameter.

Solution The tool offset number must be smaller than the one allowed by the manufacturer.

0114 ‘Offset: D3 R L I K.’

Detection While editing tables.

Cause In the tool offset table, the parameter editing order has not been respected.

Solution Enter the table parameters in the right order.

0115 ‘Tool: T4 D3 F3 N5 R5(.2).’

Detection While editing tables.

Cause In the tool table, the parameter editing order has not been respected.

Solution Enter the table parameters in the right order.

0116 ‘Origin: G54-59 axes (1-5).’

Detection While editing tables.

Cause In the Zero offset table, the zero offset to be defined (G54-G59) has not be selected.

Solution Enter the table parameters in the right order. To fill out the zero offset table, first select the offset to be defined (G54-G59) and then the zero offset position for each axis.

0117 ‘Function: M4 S4 bits(8).’

Detection While editing tables.

Cause In the «M» function table, the parameter editing order has not been respected.

Solution Edit table following the format:
M1234 (associated subroutine) (customizing bits)

0118 ‘G51 [A] E’

Detection In execution or while executing programs transmitted via DNC.

Cause In the «Look-Ahead (G51)» function, the parameter for the maximum contouring error is missing.

Solution This type of machining requires the programming of:
E : Maximum contouring error.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0119 ‘Leadscrew: Coordinate-error.’

Detection While editing tables.

Cause In the leadscrew compensation tables, the parameter editing order has not been respected.

Solution Enter the table parameters in the right order.
P123 (position of the axis to be compensated) (leadscrew error at that point)

0120 ‘Incorrect axis.’

Detection While editing tables.

Cause In the leadscrew compensation tables, an attempt has been made to edit a different axis from the one corresponding to that table.

Solution Each axis has its own table for leadscrew compensation. The table for each axis can only contain the positions for that axis.

0121 'Program P3 = value.'

Detection While editing tables.

Cause In the machine parameter table, the editing format has not been respected.

Solution Enter the table parameters in the right order.
P123 = (parameter value)

0122 'Magazine: P(1-255) = T(1-9999).'

Detection While editing tables.

Cause In the tool magazine table, the editing format has not been respected or some data is missing.

Solution Enter the table parameters in the right order.

0123 'Tool T0 does not exist.'

Detection While editing tables.

Cause In the tool table, an attempt has been made to edit a tool as T0.

Solution No tool can be edited as T0. The first tool must be T1.

0124 'Offset D0 does not exist.'

Detection While editing tables.

Cause In the tool table, an attempt has been made to edit a tool offset as D0.

Solution No tool offset can be edited as D0. The first tool offset must be D1.

0125 'Do not modify the active tool or the next one.'

Detection During execution.

Cause In the tool magazine table, an attempt has been made to change the active tool or the next one.

Solution During execution, neither the active tool nor the next one may be changed.

0126 'Tool not defined.'

Detection While editing tables.

Cause In the tool magazine table, an attempt has been made to assign to the magazine position a tool that is not defined in the tool table.

Solution Define the tool in the tool table.

0127 'Magazine is not RANDOM.'

Detection While editing tables.

Cause There is no RANDOM magazine and, in the tool magazine table, the tool number does not match the tool magazine position.

Solution When the tool magazine is not RANDOM, the tool number must be the same as the magazine position (pocket number).

0128 'The position of a special tool is set.'

Detection While editing tables.

Cause In the tool magazine table, an attempt has been made to place a tool in a magazine position reserved for a special tool.

Solution When a special tool occupies more than one position in the magazine, it has a reserved position in the magazine. No other tool can be placed in this position.

0129 'Next tool only possible in machining centers.'

Detection During execution.

Cause A tool change has been programmed with M06, but the machine is not a machining center. (it is not expecting the next tool).

Solution When the machining is not a machining center, the tool change is done automatically when programming the tool number «T».

0130 'Write 0/1.'

Detection While editing machine parameters

Cause An attempt has been made to assign the wrong value to a parameter.

Solution The parameter only admits values of 0 or 1.

0131 'Write +/-.'

Detection While editing machine parameters

Cause An attempt has been made to assign the wrong value to a parameter.

Solution The parameter only admits values of + or -.

0132 'Write YES/NO.'

Detection While editing machine parameters

Cause An attempt has been made to assign the wrong value to a parameter.

Solution The parameter only admits values of YES or NO.

0133 'Write ON/OFF.'

Detection While editing machine parameters

Cause An attempt has been made to assign the wrong value to a parameter.

Solution The parameter only admits values of ON or OFF.

0134 'Values 0 thru 2.'

0135 'Values 0 thru 3.'

0136 'Values 0 thru 4.'

0137 'Values 0 thru 9.'

0138 'Values 0 thru 29.'

0139 'Values 0 thru 100.'

0140 'Values 0 thru 255.'

0141 'Values 0 thru 9999.'

0142 'Values 0 thru 32767.'

0143 'Values within +/-32767.'

0144 'Values 0 thru 65535.'

Detection While editing machine parameters

Cause The various causes might be:

1. An attempt has been made to assign the wrong value to a parameter.
2. During execution, when inside the program a call has been made to a subroutine (MCALL, PCALL) with a value greater than allowed.

0145 'Format +/- 5.5.'

Detection While editing machine parameters

Cause An attempt has been made to assign the wrong value to a parameter.

Solution The parameter only admits values with the format:

0146 'Word does not exist.'

No explanation required

0147 'Numerical format exceeded.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A data or parameter has been assigned a value greater than the established format.

Solution Correct the syntax of the block. Most of the time, the numeric format will be 5.4 (5 integers and 4 decimals).

0148 'Text too long.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, the "ERROR" or "MSG" instruction has been assigned a text with more than 59 characters.

Solution Correct the syntax of the block. The "ERROR" and "MSG" instructions cannot be assigned texts longer than 59 characters.

0149 'Incorrect message.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, the text associated with the "ERROR" or "MSG" instruction has been edited wrong.

Solution Correct the syntax of the block. The programming format is:

(MSG "message")
(ERROR number, "message")

The message must be between " ".

0150 'Incorrect number of bits.'

Detection While editing tables.

Cause The various causes might be:

1. In the «M» function table, in the section on customizing bits:
   The number does not have 8 bits.
   The number does not consist of 0's and 1's.
2. In the machine parameter table, an attempt has been made to assign the wrong value of bit to a parameter.

Solution The solution for each cause is:

1. The customizing bits must consist of 8 digits of 0's and 1's.
2. The parameter only admits 8-bit or 16-bit numbers.

0151 'Negative numbers not allowed.'

No explanation required

0152 'Incorrect parametric programming.'

Detection During execution.

Cause The parameter has a value that is incompatible with the function it has been assigned to.

Solution This parameter may have taken the wrong value, in the program history. Correct the program so this parameter does not reach the function with that value.

0153 'Decimal format not allowed.'

No explanation required

0154 'Insufficient memory.'

Detection During execution.

Cause The CNC does not have enough memory to internally calculate the paths.

Solution Sometimes, this error is taken care of by changing the machining conditions.

0155 'Help not available.'

No explanation required

0156 'Don't program G33 ,G34, G95 or M19 S with no spindle encoder'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A "G33", "G34", "G95", "G95" or "M19 S" has been programmed without having an encoder on the spindle.

Solution If the spindle does not have an encoder, functions "M19 S", "G33", "G34" or "G95" cannot be programmed. Spindle machine parameter "NPULSES (P13)" indicates the number of encoder pulses per turn.

0157 'G79 not allowed when there is no active canned cycle.'

Detection During execution.

Cause An attempt has been made to execute the «Modification of canned cycle parameters (G79)» function without any canned cycle being active.

Solution The "G79" function modifies the values of a canned cycle; therefore, there must be an active canned cycle and the "G79" must be programmed in the influence zone of that canned cycle.

0158 'Tool T must be programmed with G67 and G68.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the «Irregular pocket canned cycle with islands (G66)» the tool has not been defined for roughing "G67" (2D/3D pockets) for semi-finishing "G67" (3D pocket) or finishing "G68" (2D/3D pocket).

Solution The irregular pocket canned cycle with islands requires the programming of the roughing tool "G67" (2D/3D pockets), the semi-finishing tool "G67" (3D pocket) and the finishing tool "G68" (2D/3D pocket).

0159 'Inch programming limit exceeded.'

Detection During execution.

Cause An attempt has been made to execute in inches a program edited in millimeters.

Solution Enter function G70 (inch programming) or G71 (mm programming) at the beginning of the program.

0160 'G79 not allowed when executing the canned cycle.'

No explanation required

0161 'G66 must be programmed before G67 and G68.'

Detection During execution.

Cause A roughing operation "G67" (2D/3D pockets), a semi-finishing operation "G67" (3D pocket) or a finishing operation "G68" (2D/3D pocket) has been programmed without having previous programmed the call to an «Irregular pocket canned cycle with islands (G66)».

Solution When working with irregular pockets, before programming the aforementioned cycles, the call to the «Irregular canned cycle with islands (G66)» must be programmed.

0162 'No negative radius allowed with absolute coordinates'

Detection During execution.

Cause While operating with absolute polar coordinates, a movement with a negative radius has been programmed.

Solution Negative radius cannot be programmed when using absolute polar coordinates.

0163 'The programmed axis is not longitudinal.'

Detection During execution.

Cause An attempt has been made to modify the coordinates of the point where the canned cycle is to be executed using the «Modification of the canned cycle parameters (G79)»function.

Solution With "G79", the parameters defining a canned cycle may be modified, except the coordinates of the point where it will be executed. To change those coordinates, program only the new coordinates.

0164 'Wrong password.'

Detection While assigning protections.

Cause [ENTER] has been pressed before selecting the type of code to be assigned a password.

Solution Use the softkeys to select the type of code to which a password is to be assigned.

0165 'Password: use uppercase/lowercase letters or digits.'

Detection While assigning protections.

Cause A bad character has been entered in the password.

Solution The password can only consist of letters (upper and lower case) or digits.

0166 'Only one HIRTH axis per block is allowed.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A movement has been programmed which involves the movement of two HIRTH axes simultaneously.

Solution The CNC does not admit movements involving more than one HIRTH axis at a time. HIRTH axes must move one at a time.

0167 ‘Rot. axis position: absolute values (G90) within 0-359.9999.’

Detection During execution.

Cause A movement of a positioning-only rotary axis has been programmed. The movement has been programmed in absolute coordinates (G90) and the target coordinate of the movement is not within the 0 to 359.9999 range.

Solution Positioning-only rotary axes: In absolute coordinates, only movements within the 0 to 359.9999 range are possible.

0168 'Rotary axis: Absolute values (G90) within +/-359.9999.'

Detection During execution.

Cause A movement of a rotary axis has been programmed. The movement has been programmed in absolute coordinates (G90) and the target coordinate of the movement is not within the 0 to 359.9999 range.

Solution Rotary axes: In absolute coordinates, only movements within the 0 to 359.9999 range are possible.

0169 ‘Modal subroutines cannot be programmed.’

Detection While executing in MDI mode

Cause An attempt has been made to call upon a modal subroutine (MCALL).

Solution MCALL modal subroutines cannot be executed from the menu option «MDI execution».

0170 ‘Program symbols 0 thru 255 in positions 0-639, 0-335. ‘

No explanation required

0171 ‘The window must be previously defined.’

Detection During normal execution or execution through the user channel.

Cause An attempt has been made to write in a window (DW) that has not been previously defined (ODW).

Solution It is not possible to write in a window that has not been previously defined. Check that the window to write in (DW) has been previously defined.

0172 ‘The program is not accessible’

Detection During execution.

Cause An attempt has been made to execute a program that cannot be executed.

Solution The program may be protected against execution. To know whether a program may be executed, check for the “X” character on the attributes column. If this character is missing, the program cannot be executed.

0173 ‘It is not possible to program angle + angle.’

No explanation required

0174 ‘Circular (helical) interpolation not possible.’

Detection During execution.

Cause An attempt has been made to execute a helical interpolation while the «LOOK-AHEAD (G51)» function was active.

Solution Helical interpolations are not possible while the «LOOK-AHEAD (G51)» function is active.

0175 ‘Analog inputs: ANAI(1-8) = +/-5 Volts.’

Detection During execution.

Cause An analog input has taken a value out of the ±5V range.

Solution Analog inputs may only take values within the ±5V range.

0176 ‘Analog outputs: ANAO(1-8) = +/-10 Volts.’

Detection During execution.

Cause An analog output has been assigned a value out of the ±10V range.

Solution Analog outputs may only take values within the ±10V range.

0177 'A gantry axis cannot be part of the active plane.'

No explanation required

0178 'G96 only possible with analog spindle.'

Detection During execution.

Cause The "G96" function has been programmed but either the spindle speed is not controlled or the spindle does not have an encoder.

Solution To operate with the "G96" function, the spindle speed must be controlled (SPDLTYPE(P0)=0) and the spindle must have an encoder (NPULSES(P13) other than zero).

0179 'Do not program more than 4 axes simultaneously.'

No explanation required

0180 'Program DNC1/2, HD or CARD A (optional).'

Detection While editing or executing.

Cause While programming in high level language, in the "OPEN" and "EXEC" instructions, an attempt has been made to program a parameter other than DNC1/2, HD or CARD A, or the DNC parameter has been assigned a value other than 1 or 2.

Solution Check the syntax of the block.

0181 'Program A (append) or D (delete).'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the "OPEN" instruction the A/D parameter is missing.

Solution Check the syntax of the block. The programming format is:
(OPEN P.........,A/D,…)

Where:

A Appends new blocks after the existing ones.
D: Deletes the existing program and it opens it as a new one.

0182 'Option not available.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A «G» function has been defined which is not a software option.

0183 'Cycle does not exist.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the "DIGIT" instruction, a digitizing cycle has been defined which is not available.

Solution The "DIGIT" instruction only admits two types of digitizing:
(DIGIT 1,…) : Grid pattern digitizing cycle.
(DIGIT 2,…) : Arc pattern digitizing cycle.

0184 'T with subroutine: program only T and D.'

No explanation required

0185 'Tool offset does not exist'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause Within the block syntax, a tool offset has been called upon which is greater than the ones allowed by the manufacturer.

Solution Program a new smaller tool offset.

0188 'Function not possible from PLC.'

Detection During execution.

Cause From the PLC channel and using the "CNCEX" instruction, an attempt has been made to execute a function that is incompatible with the PLC channel execution.

Solution The installation manual (chapter 11.1.2) offers a list of the functions and instructions that may be executed through the PLC channel.

0189 'The live tool does not exist.'

No explanation required

0190 ‘Programming not allowed while in tracing mode.’

Detection During execution.

Cause Among the blocks defining the «Tracing and digitizing canned cycles (TRACE)», there is block that contains a «G» function which does not belong in the profile definition.

Solution The «G» functions available in the profile definition are:

G00	G01	G02	G03	G06	G08	G09	G36
G39	G53	G70	G71	G90	G91	G93	

0191 ‘Do not program tracing axes.’

Detection During execution.

Cause An attempt has been made to move an axis that has been defined as a tracing axis using the “G23” function.

Solution The tracing axes are controlled by the CNC. To deactivate the tracing axes, use the “G25” function..

0192 ‘Incorrect active plane and longitudinal axis.’

Detection During execution.

Cause While programming in high level language, an attempt has been made to execute a probing cycle using the “PROBE” instruction, but the longitudinal axis is included in the active plane.

Solution The “PROBE” probing canned cycles are executed on the X, Y, Z axes, the active plane being formed by two of them. The other axis must be perpendicular and it must be selected as the longitudinal axis.

0193 ‘G23 has not been programmed.’

Detection During execution.

Cause Digitizing “G24” has been activated or a tracing contour “G27” has been programmed, but without previously activating the tracing function “G23”.

Solution To digitize or operate with a contour, the tracing function must be activated previously.

0194 ‘Repositioning not allowed.’

Detection During execution.

Cause The axes cannot be repositioned using the “REPOS” instruction because the subroutine has not been activated with one of the interruption inputs.

Solution Before executing the “REPOS” instruction, one of the interruption inputs must be activated.

0195 ‘Axes X, Y or Z slaved or synchronized.’

Detection During execution.

Cause While programming in high level language, an attempt has been made to execute a probing cycle using the “PROBE” instruction, but one of the X, Y or Z axis is slaved or synchronized.

Solution To execute the “PROBE”” instruction, the X, Y, Z axes must not be slaved or synchronized. To unslave the axes, program “G78”.

0196 ‘Axes X, Y and Z must exist.’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, an attempt has been made to edit the “PROBE” instruction, but one of the X, Y or Z axis is missing.

Solution To operate with the “PROBE” instruction, the X, Y, Z axes must be defined.

0198 ‘Deflection out of range.’

Detection During execution.

Cause In the tracing cycle “G23”, a nominal probe deflection has been defined which is greater than the value set by machine parameter.

Solution Program a smaller nominal probe deflection.

0199 'Rotary axis preset: values between 0 and 359.9999.'

Detection While presetting coordinates.

Cause An attempt has been made preset the coordinates of a rotary axis with a value out of the 0 to 359.9999 range.

Solution The preset value of rotary axes must be within the 0 to 359.9999 range.

0200 'Program: G52 axis +/-5.5'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause When programming the «Movement against a hard stop (G52)», either the axis to be moved has not been programmed or several axes have been programmed.

Solution When programming "G52", the axis to be moved must be indicated. Only one axis may be programmed at a time.

0201 'Program only one positioning axis in G01.'

No explanation required

0202 'Program G27 only when tracing a profile.'

Detection During execution.

Cause A tracing contour (G27) has been defined, but the tracing function is neither bi-dimensional nor three-dimensional.

Solution The «Definition of a tracing contour (G27)» function must only be defined when tracing or digitizing in two or three dimensions.

0203 'G23-G27 not allowed during INSPECTION.'

No explanation required

0204 'Incorrect tracing method.'

Detection During execution.

Cause While executing a manual tracing "G23", an attempt has been made to jog a «follower» axis with the jog keys or the electronic handwheels.

Solution When executing a manual tracing, the axes selected as followers are moved by hand. The rest may be jogged with the jog keys or the electronic handwheels.

0205 'Incorrect digitizing method.'

Detection During execution.

Cause Point-to-point digitizing has been defined, but the CNC is not in jog mode (it is in either in simulation or execution mode, instead).

Solution To execute point-to-point digitizing, the CNC must be in jog mode.

0206 'Values 0 thru 6.'

Detection While editing machine parameters

Cause An attempt has been made to assign the wrong value to a parameter.

Solution The parameter only admits values between 0 and 6.

0207 'Complete Table.'

Detection While editing tables.

Cause In the tables for «M» functions or tool offsets, an attempt has been made to define more data than those allowed by the manufacturer by means of machine parameters. When loading a table via DNC, the CNC does not delete the previous table, it replaces the existing values and it copies the new data in the free positions of the table.

Solution The maximum number of data that can be defined is limited by the machine parameters:

Maximum number of «M» functions	NMISCFUN(P29).
Maximum number of	NTOOL(P23).
Maximum number of tool offset	NTOFFSET(P27).
Maximum number of magazine positions	NPOCKET(P25).

To load a new table via DNC, the previous table should be deleted.

0208 'Program A from 0 to 255'

Detection During execution.

Cause In the «LOOK-AHEAD (G51)» function, parameter "A" (% of acceleration to be applied) has been programmed with a value greater than 255.

Solution Parameter "A" is optional, but when programmed, it must have a value between 0 and 255.

0209 'Program nesting not allowed.'

Detection During execution.

Cause From a running program, an attempt has been made to execute another program with the "EXEC" instruction which in turn also has an "EXEC" instruction.

Solution Another program cannot be called upon from a program being executed using the "EXEC" instruction.

0210 'No compensation is permitted.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An attempt has been made to activate or cancel tool radius compensation (G41, G42, G40) in a block containing a nonlinear movement.

Solution Tool radius compensation must be activated/deactivated in linear movements (G00, G01).

0211 'Do not program a zero offset without cancelling the previous one.'

Detection During execution.

Cause An attempt has been made to define an incline plane using the «Definition of the incline plane (G49)» function while another one was already defined.

Solution To define a new incline plane, the one previously defined must be canceled first. To cancel an incline plane, program "G49" without parameters.

0212 'Programming not permitted while G48-G49 are active.'

Detection During execution.

Cause While programming in high level language, an attempt has been made to execute a probing cycle with the "PROBE" instruction while function "G48" or "G49" was active.

Solution The digitizing cycles "PROBE" are carried out on the X, Y, Z axes. Therefore, neither the "G48" nor the "G49" function may be active when executing them.

0213 'A second spindle is required for G28, G29, G77 or G78.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An attempt has been made to select the work spindle with "G28/G29" or synchronize spindles with "G77/G78", but the machine only has one work spindle.

Solution If the machine only has one work spindle, the "G28, G29, G77 and G78" functions cannot be programmed.

0214 'Invalid G function when selecting a profile'

Detection While restoring a profile.

Cause Within the group of blocks selected to restore the profile, there is a block containing a «G» code that does not belong in the profile definition.

Solution The «G» functions available in the profile definition are:

G00	G01	G02	G03	G06	G08	G09
G36	G37	G38	G39	G90	G91	G93

0215 'Invalid G function after first point of profile'

Detection While restoring a profile.

Cause Within the selected blocks for restoring the profile, and after the starting point of a profile, there is a block containing a «G» function that does not belong in the profile definition.

Solution The «G» functions available in the profile definition are:

G00	G01	G02	G03	G06	G08	G09
G36	G37	G38	G39	G90	G91	G93

0216 ‘Nonparametric assignment after first point of profile’

Detection While restoring a profile.

Cause Within the selected blocks for restoring the profile, and after the starting point of a profile, a nonparametric assignment has been programmed in high level language (a local or global parameter).

Solution The only high level instructions that can be edited are assignments to local parameters (P0 thru P25) and global parameters (P100 thru P299).

0217 ‘Invalid programming after first point of profile’

Detection While restoring a profile.

Cause Within the selected blocks for restoring the profile, and after the starting point of a profile, there is a high level block that is not an assignment.

Solution The only high level instructions that can be edited are assignments to local parameters (P0 thru P25) and global parameters (P100 thru P299).

0218 ‘The axis cannot be programmed after first point of profile’

Detection While restoring a profile.

Cause Within the selected blocks for restoring the profile, and after the starting point of a profile, a position has been defined on an axis that does not belong to the active plane. A surface coordinate may have been defined after the starting point of the profile.

Solution The surface coordinate of the profiles is only defined in the starting block of the first profile, the one corresponding to the starting point of the outside profile.

0219 ‘First point programmed wrong when selecting profile’

Detection While selecting a profile.

Cause The starting point of the profile has been programmed wrong. One of the two coordinates defining its position is missing.

Solution The starting point of a profile must be defined on the two axes forming the active plane.

0226 ‘A tool cannot be programmed with G48 active’

Detection During execution.

Cause A tool change has been programmed while the «TCP transformation (G48)» function is active.

Solution A tool change cannot take place while TCP transformation is active. To make a tool change, cancel TCP transformation first.

0227 ‘Program Q between +/-359.9999.’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the «Electronic threading (G33)» function, the entry angle “Q” has been programmed with a value out of the ±359.9999 range.

Solution Program an entry angle within the ±359.9999 range.

0228 ‘Do not program "Q" with parameter M19TYPE=0.’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the «Electronic threading (G33)» function, an entry angle “Q” has been programmed, but the type of spindle orientation available does not allow this operation.

Solution In order to define an entry angle, spindle machine parameter M19TYPE(P43) must be set to «1».

0229 ‘Program maximum X’
0230 ‘Program minimum Y’
0231 ‘Program maximum Y’
0232 ‘Program minimum Z’
0233 ‘Program maximum Z’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, in the “DGWZ” instruction, the indicated limit is missing or it has been defined with a non-numerical value.

Solution Check the syntax of the block.

0234 'Wrong graphic limits'

Detection During execution.

Cause One of the lower limits defined with the "DGWZ" instruction is greater than its corresponding upper limit.

Solution Program the upper limit of the graphics display area greater than the lower ones.

0235 'Do not program the axis in tangential control'

No explanation required

0236 'Do not program the longitudinal axis or the axis of the active plane'

No explanation required

0237 'Program values between +/-359.9999.'

Detection During execution.

Cause A G30 offset has been programmed greater than the maximum allowed. For example G30 D380

Solution The offset must be within ±359.9999.

0238 'Do not program G30 without synchronizing the spindles in speed'

Detection During execution.

Cause An attempt has been made to synchronize the spindles in "G30" offset without having them synchronized in speed.

Solution First, synchronize the spindle in speed using G77S.

0239 'Do not synchronize the spindles while the "C" axis is active'

Detection During execution.

Cause An attempt has been made to synchronize the spindle, but the "C" axis is not active.

Solution Activate the "C" axis first.

0240 'Do not activate the "C" axis while the spindles are synchronized'

Detection During execution.

Cause An attempt has been made to activate the "C" axis while the spindles were synchronized.

Solution First, cancel the spindle synchronization (G78 S).

0241 'Do not program G77 S, G78 S if there is no encoder at the spindle'

Detection During execution.

Cause An attempt has been made to synchronize the spindles (G77 S or G78 S) and one of them does not have an encoder or Sercos feedback.

Solution Both spindles must have an encoder or Sercos feedback.

0242 'Do not synchronize spindles with M19TYPE=0'

Detection During execution.

Cause An attempt has been made to synchronize the spindles (G77 S or G78 S) and one of them has parameter M19TYPE=0.

Solution Both spindles must have parameter M19TYPE=1

0243 'Values 0 thru 15.'

No explanation required

BLOCK PREPARATION AND EXECUTION ERRORS

1000 'There is no enough path information.'

Detection During execution.

Cause The program contains too many blocks without information about the path to apply tool radius compensation, rounding, chamfer or tangential entry or exit.

Solution In order to carry out these operations, the CNC needs to know in advance the path to follow; therefore, there cannot be more than 48 blocks in a row without information about the path to follow.

1001 'Plane change in rounding/chamfering.'

Detection During execution.

Cause A plane change has been programmed on the path following the definition of a "controlled corner rounding G36" or "chamfer (G39)".

Solution The plane cannot be changed while executing a rounding or a chamfer. The path following the definition of a rounding or chamfer must be in the same plane that the rounding or the chamfer.

1002 'Rounding radius too large.'

Detection During execution.

Cause In the "Controlled corner rounding (G36)" function, the programmed rounding radius is larger than one of the paths where it has been defined.

Solution The rounding radius must be smaller than the paths that define it.

1003 'Rounding in last block.'

Detection During execution.

Cause A "Controlled rounding radius (G36) or "Chamfer (G39) has been defined on the last path of the program or when the CNC does not find information about the path following the definition of the rounding or chamfer.

Solution A rounding or chamfer must be defined between two paths.

1004 'Tangential output programmed wrong'

Detection During execution.

Cause The move following the definition of a tangential output (G38) is a circular path.

Solution

The move following the definition of a tangential output must be a straight path.

1005 'Chamfer programmed wrong.'

Detection During execution.

Cause The move following the definition of a "Chamfer (G39)" is a circular path.

Solution The move following the definition of a chamfer must be a straight path.

1006 'Chamfer value too large.'

Detection During execution.

Cause In the "Chamfer (G39)" function, the programmed chamfer value is larger than one of the paths where it has been defined.

Solution The chamfer size must be smaller than the paths that define it.

1007 'G8 defined wrong.'

Detection During execution.

Cause The various causes might be:

1. When a full circle has been programmed using the function "Arc tangent to previous path (G08)"
2. When the tangent path ends in a point of the previous path or its extension (in a straight line).
3. In an irregular pocket canned cycle with islands, when programming function "G08" in the block following the definition of the beginning of the profile (G00).

Solution The solution for each cause is:

1. Function “G08” does not allow programming full circles.
2. Tangent path must not end in a point of the previous path or in its extension (in a straight line).
3. The CNC does not have information about the previous path and cannot execute the tangent arc.

1008 ‘There is no information about the previous path’

Detection During execution.

Cause An arc tangent to the previous path has been programmed using function “G08”, but there is no information about the previous path.

Solution To do a path tangent to the previous one, there must be information about the previous path and it must be within the 48 blocks preceding the tangent path.

1009 ‘There is no information for tangent arc in pockets with islands.’

Detection During execution.

Cause Within the set of blocks defining the profile of an irregular pocket with islands, a tangent arc has been programmed, but some data is missing or there is not enough information about the previous path.

Solution Check the data that defines the profile.

1010 ‘Wrong plane for tangent path.’

Detection During execution.

Cause A plane change has been programmed between the definition of the function “arc tangent to previous path (G08)” and the previous path.

Solution A plane cannot be changed between two paths

1011 ‘Jog movement out of limits.’

Detection During execution.

Cause After defining an incline plane, the tool positions at a point out of the work limits; the operator tries to move an axis with the JOG keys, the tool does not position within the area defined by the work limits.

Solution Jog the axis that allows to position the tool within the work limits.

1012 ‘G48 cannot be programmed while G43 is active’

Detection During execution.

Cause An attempt has been made to activate TCP (G48) while tool length compensation (G43) was active.

Solution To activate TCP transformation (G48), tool length compensation must be OFF because TCP already applies its own specific tool length compensation.

1013 ‘G43 cannot be programmed while G48 is active’

Detection During execution.

Cause An attempt has been made to activate tool length compensation (G43) while TCP (G48) was active.

Solution To activate tool length compensation (G43) cannot be activated while TCP transformation (G48) is ON because TCP already applies its own specific tool length compensation.

1014 ‘G49 cannot be programmed if it’’s already active’

No explanation required

1015 ‘The tool is not defined in the tool table’

Detection During execution.

Cause A tool change has been defined, but the new tool is not defined in the tool table.

Solution Define the new tool in the tool table.

1016 ‘The tool is not in the tool magazine’

Detection During execution.

Cause A tool change has been defined, but the new tool is not defined in position of the tool magazine table.

Solution Define the new tool in the tool magazine table.

1017 'There is no empty pocket in the tool magazine'

Detection During execution.

Cause A tool change has been defined and there is no empty pocket for the tool that is currently in the spindle.

Solution Perhaps, the new tool has been defined as special in the tool table and there are more than one pockets reserved to it in the magazine. In this case, that position is set for that tool and no other tool can occupy it. To avoid this error, an empty pocket (position) should be left in the tool magazine.

1018 'A tool change has been programmed without M06'

Detection During execution.

Cause An M06 has not been programmed after having looked for a tool and before searching again.

Solution This error occurs when having a machining center (general machine parameter TOFFM06(P28)=YES) that has a cyclic tool changer (general machine parameter CYCATC(P61)=YES). In this case, the tool change must be done with an m06 after searching for a tool and before searching for the next one.

1019 'There is no tool of the same family for replacement.'

Detection During execution.

Cause The real life of the requested tool exceeds its nominal life. The CNC has tried to replace it with another one of the same family, but it has not found any.

Solution Replace the tool or define another one of the same family.

1020 'Do not change the active or pending tool using high level language.'

Detection During execution.

Cause While programming in high level language and using the "TMZT" variable, an attempt has been made to assign the current or next tool to a magazine position.

Solution Use the "T" function to change the active tool or the next one. The "TMZT" variable cannot be used to move the active tool or the next one to the magazine.

1021 'No tool offset has been programmed in the canned cycle.'

Detection During execution.

Cause The "PROBE" canned cycle for tool calibration has been programmed, but no tool offset has been selected.

Solution To execute the "Tool calibration canned cycle (PROBE), a tool offset must be selected where the probing cycle information will be stored.

1022 'Tool radius programmed incorrectly'

No explanation required

1023 'G67. Tool radius too large.'

Detection During execution.

Cause In the "Irregular pocket canned cycle with islands (G66)", a tool has been selected whose radius is too large for the roughing operation "G67" (2D pocket). The tool cannot get in anywhere in the pocket.

Solution Select a tool of a smaller radius.

1024 'G68. Tool radius too large.'

Detection During execution.

Cause In the "Irregular pocket canned cycle with islands (G66)", a tool has been selected whose radius is too large for the finishing operation "G68" (2D pocket). Somewhere in the machining operation, the distance between the outside profile and the profile of an island is smaller than the tool diameter.

Solution Select a tool of a smaller radius.

1025 'A tool with no radius has been programmed'

Detection During execution.

Cause In the "Irregular pocket canned cycle with islands (G66), a (G67/G68) operation has been programmed with no radius.

Solution Correct the tool definition in the tool table or select another one for that operation.

1026 'A step has been programmed that is larger than the tool diameter'

Detection During execution.

Cause In the "Rectangular pocket canned cycle (G87)", in the "circular pocket canned cycle (G68) or in an operation of the "irregular pocket canned cycle with islands (G66)", the "C" parameter has been programmed with a value larger than that of the tool that will be used for that operation.

Solution Correct the syntax of the block. The machining step "C" must be smaller than or equal to the tool diameter.

1027 'A tool cannot be programmed with G48 active'

Detection During execution.

Cause A tool change has been programmed while the «TCP transformation (G48)» function is active.

Solution A tool change cannot take place while TCP transformation is active. To make a tool change, cancel TCP transformation first.

1028 'Do not switch axes over while G23, G48 or G49 is active'

Detection During execution.

Cause An attempt has been made to switch over to an axis or back (G28/G29) while function "G23", "G48" or "G49" was active.

Solution The axes cannot be swapped while function "G23", "G48" or "G49" is active.

1029 'Do not swap axes that are already swapped.'

Detection During execution.

Cause An attempt has been made to swap (G28) an axis that was already swapped with another one.

Solution An axis already swapped with another one cannot be swapped with a third one. It must be switched back first (G29 axis)

1030 'The "M" for the automatic gear change does not fit'

Detection During execution.

Cause Using automatic gear change, 7 "M" functions and the "S" function (involving a gear change) have been programmed. In this case, the CNC cannot include the "M" for automatic gear change in that block.

Solution Program an "M" function or the "S" function in a separate block.

1031 'No subroutine is allowed with automatic gear change.'

Detection During execution.

Cause On machines having an automatic gear change, when programming a spindle speed "S" that involves a gear change and the "M" function of the automatic gear change has a subroutine associated with it.

Solution When having an automatic gear change, the "M" functions corresponding to the gear change cannot have a subroutine associated with it.

1032 'Spindle gear (range) not defined in M19.'

Detection During execution.

Cause "M19" has been programmed, but none of the gear change functions "M41", "M42", "M43" or "M44" are active.

Solution On power-up, the CNC does not assume any range; Therefore, if the gear change function is not generated automatically (spindle parameter AUTOGEAR(P6)=NO), the auxiliary gear change functions ("M41", "M42", "M43" or "M44") must be programmed.

1033 ‘Wrong gear change.’

Detection During execution.

Cause The various causes might be:

1. When trying to make a gear change and the machine parameters for gears (MAXGEAR1, MAXGEAR2, MAXGEAR3, or MAXGEAR4) are set wrong. All the gears (ranges) have not been used and the unused ones have been set to a maximum speed of zero rpm.
2. When programming a gear change (“M41”, “M42”, “M43” or “M44”) and the PLC has not responded with the relevant active gear signal (GEAR1, GEAR2, GEAR3 or GEAR4).

Solution The solution for each cause is:

1. When not using all four gears, the lower ones must be used starting with “MAXGEAR1” and the unused gears must be assigned the value of the highest one used.
2. Check the PLC program.

1034 ‘”S” has been programmed, but no gear is active.’

Detection During execution.

Cause An attempt has been made to start the spindle, but no gear is selected.

Solution On power-up, the CNC does not assume any range; Therefore, when programing a spindle speed and the gear change function is not generated automatically (spindle parameter AUTOGEAR(P6)=NO), the auxiliary gear change functions (“M41”, “M42”, “M43” or “M44”) must be programmed.

1035 ‘Programmed “S” too high’

Detection During execution.

Cause An “S” has been programmed with a higher value than allowed by the last active gear.

Solution Program a lower spindle speed “S” .

1036 ‘”S” has not been programmed in G95 or in threading’

Detection During execution.

Cause “mm(inches)/revolution (G95)” or “electronic threading (G33)”has been programmed, but no spindle speed has been selected.

Solution An “S” must be programmed to work in mm/rev (G95) or for an electronic threading (G33).

1038 ‘The spindle has not been oriented’

Detection During execution.

Cause A threading cycle is to be executed without having oriented the active spindle (main or secondary) first.

1040 ‘Canned cycle does not exist’

Detection While executing in MDI mode

Cause When trying to execute a canned cycle (G8x) after interrupting a program during the execution of a canned cycle (G8x) and then changing the plane.

Solution Do not interrupt the program while executing a canned cycle.

1041 ‘Mandatory parameter missing in canned cycle’

Detection During execution.

Cause The various causes might be:

1. In the “Irregular canned cycle with islands” some parameter is missing.
   2D POCKETS
   - In the roughing operation “G67”, either parameter “I” or “R” is missing.
   - There is no roughing operation and in the finishing operation “G68”, either parameter “I” or “R” is missing.

   3D POCKETS
   - In the roughing operation “G67”, either parameter “I” or “R” is missing.
   - There is no roughing operation and in the semifinishing operation “G67”, either parameter “I” or “R” is missing.

- There is neither roughing nor semifinishing operation and in the finishing operation "G68", either parameter "I" or "R" is missing.
- In the finishing operation "G68", parameter "B" is missing.

2. In the "Digitizing canned cycle" some parameter is missing.

Solution Correct the definition of parameters.

Pocket with islands (finishing operation)

In the irregular pocket canned cycle with islands, parameters "I" and "R" must be programmed in the roughing operation. If there is no roughing operation, they must be defined in the finishing operation (2D) or in the semifinishing operation (3D). If there is no semifinishing operation (3D), they must be defined in the finishing operation. In the 3D pocket, parameter "B" must be defined in the finishing operation.

Digitizing cycles

Check the syntax of the block. The programming formats are:
(DIGIT 1,X,Y,Z,I,J,K,B,C,D,F)
(DIGIT 2,X,Y,Z,I,J,K,A,B,C,F)

1042 'Wrong parameter value in canned cycle'

Detection During execution.

Cause The various causes might be:

1. In the "Irregular pocket canned cycle with islands", when a parameter has been defined with a wrong value in the finishing operation "G68". Perhaps, a parameter that only takes positive values has been assigned a negative value (or zero).
2. In the "Irregular pocket canned cycle with islands", when in the drilling operation (G69) parameter "B", "C" or "H" has been defined with a zero value.
3. In the rectangular (G87) or circular (G88) pocket canned cycles, either parameter "C" or a pocket dimension has been defined with a zero value.
4. In the "Deep hole drilling canned cycle with variable peck (G69), parameter "C" has been defined with zero value.
5. In the digitizing canned cycle, a parameter has been assigned a wrong value. Perhaps, a parameter that only takes positive values has been assigned a negative value (or zero).

Solution Correct the definition of parameters.

Pocket with islands (finishing operation)

"Q" parameter	Only takes a value of 0, 1 or 2.
"B" parameter	Only takes values other than zero.
"J" parameter	It must be smaller than the radius of the tool used for that operation.

GRID pattern digitizing.

"B" parameter	Only takes positive values greater than zero.
"C" parameter	Only takes positive values other than zero.
"D" parameter	It only admits values of 0 or 1.

ARC pattern digitizing.

"J" and "C" parameter	Only takes positive values greater than zero.
"K", "A" and "B" parameter	It only admits positive values.

1043 'Wrong depth profile in pocket with islands.'

Detection During execution.

Cause In the "Irregular pocket canned cycle with islands" (3D):

- The depth profiles of two sections of the same contour (simple or composite) cross each other.
- A contour cannot be finished with the programmed tool (spherical path with non-spherical tool).

Solution The depth profiles of two sections of the same profile cannot cross each other. On the other hand, the depth profile must be defined after the plane profile and the same starting point must be used in both profiles. Check that the tip of the selected tool is the best for the programmed depth profile.

1044 'Plane profile intersects itself in a pocket with islands'

Detection During execution.

Cause Within the set of profiles that define a pocket with islands, one of the profiles intersects itself.

Solution Check the definition of the profiles. The profile of a pocket with islands cannot intersect itself.

1045 'Error when programming a drilling operation in a pocket with islands.'

Detection During execution.

Cause In the "Irregular pocket canned cycle with islands (G66), a canned cycle has been programmed that is not for drilling.

Solution In the drilling operation, only canned cycle "G81", "G82", "G83" or "G69" may be programmed.

1046 'Wrong tool position before the canned cycle'

Detection During execution.

Cause When calling a canned cycle, the tool is positioned between the reference plane and the final depth coordinate of one of the operations.

Solution When calling a canned cycle, the tool must be positioned above the reference plane.

1047 'Open plane profile in pocket with islands'

Detection During execution.

Cause Within the set of profiles that define a pocket with islands, one of the profiles does not start and end at the same point.

Solution Check the definition of the profiles. The profiles that define the pockets with islands must be closed. The error may occurred because "G01" has not been programmed after the beginning, with "G00", of one of the profiles.

1048 'Part surface coordinate not programmed in pocket with islands'

Detection During execution.

Cause The part surface coordinate of the pocket has not been programmed at the first point of the geometry definition.

Solution The data for the surface coordinate must be defined in the first definition block of the pocket profile (in absolute coordinates).

1049 'Wrong reference plane coordinate in canned cycle'

Detection During execution.

Cause In an operation of the "Irregular pocket canned cycle with islands (G66), the coordinate of the reference plane is located between the part surface coordinate and the final depth coordinate of one of the operations.

Solution The reference plane must be located above the part surface. This error comes up sometimes because the part surface position has been programmed in incremental coordinates. (The pocket surface data must be programmed in absolute coordinates).

1050 'Wrong value to be assigned to a variable'

Detection During execution.

Cause Using parameters, the value assigned to a variable is too high.

Solution Check the program history to make sure that this parameter does not have that value when it reaches the block where this assignment is made.

1051 'Wrong access to PLC variable.'

Detection During execution.

Cause From the CNC, an attempt has been made to read a PLC variable that is not defined in the PLC program.

1052 'Access to a variable with wrong index'

Detection During editing

Cause While programming in high level language, an operation has been carried out either with a local parameter greater than 25 or with a global parameter greater 299.

Solution The parameters used by the CNC are:

Local: 0-25.
Global: 100-299.

Other parameters out of these ranges cannot be used in operations.

1053 'Local parameters not accessible'

Detection While executing in the user channel.

Cause An attempt has been made to execute a block with an operation that uses local parameters.

Solution The program that is executed in the user channel does not allow operations with local parameters (P0 to P25).

1054 'Limit of local parameters exceeded'

Detection During execution.

Cause While programming in high level language, more than 6 nesting levels have been used with the "PCALL" instruction. More than 6 calls have been made in the same loop using the "PCALL" instruction.

Solution Only up to 6 nesting levels are allowed for local parameters within the 15 nesting levels of the subroutines. Calling with a "PCALL" instruction generates a new nesting level for local parameters (and a new one for subroutines).

1055 'Nesting exceeded'

Detection During execution.

Cause While programming in high level language, more than 15 nesting levels have been used with the "CALL", "PCALL" or "MCALL" instruction. More than 15 calls have been made in the same loop using the "CALL", "PCALL" or "MCALL" instruction.

Solution Only 15 nesting levels allowed. Calling with the "CALL", "PCALL" and "MCALL" instructions generates a new nesting level.

1056 'RET not associated with subroutine.'

Detection During execution.

Cause The "RET" instruction has been edited, but the "SUB" instruction has not been edited before.

Solution To using the "RET" instruction (subroutine), the subroutine must begin with the "SUB (subroutine number)".

1057 'Undefined subroutine'

Detection During execution.

Cause A (CALL, PCALL…) has been made to a subroutine that was not defined in the CNC memory.

Solution Check that the name of the subroutine is correct and that the subroutine exists in the CNC memory (not necessarily in the same program where the call is).

1058 'Undefined probing canned cycle'

Detection During execution.

Cause Using the "PROBE" instruction, a probing cycle has been defined which is not available.

Solution The available "PROBE" canned cycles are 1 to 9.

1059 'Jump to an undefined label'

Detection During execution.

Cause While programming in high level language, the "GOTO N..." instruction has been programmed, but the programmed block number (N) does not exist.

Solution When programming the "GOTO N..." instruction, the block it refers to must be defined in the same program.

1060 'Undefined label'

Detection During execution.

Cause The various causes might be:

1. While programming in high level language, the instrucción "RPT N..., N..." instruction has been programmed, but a programmed block number (N) does not exist.
2. When programming "G66 … S...E..." in an "Irregular pocket canned cycle with islands (G66) and one of the data defining the beginning or the end of the profiles is missing.

Solution The solution for each cause is:

1. When programming the "RPT N..., N..." instruction, the blocks it refers to must be defined in the same program.
2. Check the program. Place the label for parameter "S" at the beginning of the profile definition and the label for parameter "E" at the end of the profile definition.

1061 'Label cannot be searched'

Detection While executing in MDI mode

Cause While programming in high level language, either an "RPT N..., N..." or "GOTO N..." instruction has been defined.

Solution While operating in MDI mode, "RPT" or "GOTO" type instructions cannot be programmed.

1062 'Subroutine in an unavailable program.'

Detection During execution.

Cause A call has been made to a subroutine that it is located in a program being used by the DNC.

Solution Wait for the DNC to finish using the program. If the subroutine is to be used often, it should be stored in a separate program.

1063 'The program cannot be opened.'

Detection During execution.

Cause While executing a program in infinite mode, an attempt has been made to execute another infinite program from the current one using the "EXEC" instruction.

Solution Only one infinite program may be executed at a time.

1064 'The program cannot be executed'

Detection During execution.

Cause An attempt has been made to execute a program from another with the "EXEC" instruction, but the program does not exist or is protected against execution.

Solution The program to be executed with the "EXEC" instruction must exist in the CNC memory and must be executable.

1065 'Beginning of compensation without straight path'

Detection During execution.

Cause The first movement in work plane after activating tool radius compensation (G41/G42) is not a linear movement.

Solution The first movement after activating radius compensation (G41/G42) must be linear.

1066 'End of compensation without straight path'

Detection During execution.

Cause The first movement in work plane after deactivating tool radius compensation (G40) is not a linear movement.

Solution The first movement after deactivating radius compensation (G40) must be linear.

1067 'Compensation radius too large.'

Detection During execution.

Cause While working with tool radius compensation (G41/G42), an inside radius has been programmed with a smaller radius than that of the tool.

Solution use a tool with a smaller radius. When working with tool radius compensation, the arc radius must larger than that of the tool. Otherwise, the tool cannot machine the programmed path.

1068 'Step on linear path'

Detection During execution.

Cause When operating with tool compensation (G41/G42), the profile has a straight section that cannot be machined because the tool diameter is too large.

Solution use a tool with a smaller radius.

1069 'Circular path defined incorrectly'

No explanation required

1070 'Step on circular path'

Detection During execution.

Cause When operating with tool compensation (G41/G42), the profile has a curved section that cannot be machined because the tool diameter is too large.

Solution use a tool with a smaller radius.

1071 'Plane change in tool radius compensation.'

Detection During execution.

Cause When operating with tool compensation (G41/G42), another work plane has been selected.

Solution To change the work plane, tool radius compensation must be off (G40).

1072 'Tool radius compensation not possible with positioning-only rotary axis.

Detection During execution.

Cause An attempt has been made to move a positioning-only axis with tool radius compensation (G41/G42).

Solution Tool radius compensation not allowed for positioning-only rotary axes. Use "G40" to cancel tool radius compensation.

1076 'Coordinate angle programmed wrong.'

Detection During execution.

Cause When programming in angle-coordinate format, an axis movement has been programmed with an angle perpendicular to that axis. (For example, the main plane is formed by the XY axes and the X axis movement is programmed at a 90º angle).

Solution Check and correct the definition of the movement in the program. If using parameters, check that the parameters have the correct values when arriving to the definition of the movement.

1077 'Either the arc radius is too small or a full circle has been programmed'

Detection During execution.

Cause The various causes might be:

1. When programming a full circle using the "G02/G03 X Y R" format.
2. When programming using the "G02/G03 X Y R" format, the distance to the arc's end point is greater than the diameter of the programmed circle.

Solution The solution for each cause is:

1. This format cannot be used to make full circles. Program the coordinates of the end point different from those of the starting point.
2. The diameter of the circle must be greater than the distance to the arc's end point.

1078 'Negative radius in polar coordinates'

Detection During execution.

Cause Working with incremental polar coordinates, a block is executed where the end position has a negative radius.

Solution Incremental polar coordinate programming allows negative radius, but the (absolute) end point of the radius must be positive,

G74 'There is no subroutine associated with G74'

Detection While executing a home search.

Cause The various causes might be:

1. When trying to search home on all the axes manually, but there is no associated subroutine indicating the home searching sequence (order).
2. "G74" has been programmed, but there is no associated subroutine indicating the home searching sequence (order).

Solution The solution for each cause is:

1. An associated subroutine is required to execute the "G74" function.
2. If "G74" is to be executed from a program, the home searching order must be defined.

1080 'Plane change in tool inspection'

Detection While executing the "tool inspection" option.

Cause the work plane has been chanted and the original one has not been restored before resuming the execution.

Solution The plane that was active before inspecting the tool must be restored before resuming the execution.

1081 'Block not allowed in tool inspection.'

Detection While executing the "tool inspection" option.

Cause An attempt has been made to execute the "RET" instruction.

Solution This instruction cannot be executed in the "tool inspection" option.

1082 'The probe signal has not been received.'

Detection During execution.

Cause The various causes might be:

1. When programming a "PROBE" canned cycle, the probe has moved the maximum safety distance of the cycle without the CNC receiving the probe signal.
2. When programming the "G75" function, it has reached the end point and the CNC has not received the signal from the probe. (Only when general machine parameter PROBERR(P119)=YES).

Solution The solution for each cause is:

1. Check that the probe is connected properly.
   The maximum probing distance (in PROBE cycles) depends on the safety distance "B". To increase this distance, increase the safety distance.
2. If PROBERR(P119)=NO, this error will not be issued when the end point is reached without having received the probe signal (only with "G75").

1083 'Range exceeded'

Detection During execution.

Cause The distance for the axes to travel is very long and the programmed feedrate is too low.

Solution Program a higher speed for that movement.

1084 'Arc programmed wrong'

Detection During execution.

Cause The various causes might be:

1. When the arc programmed using "G02/G03 X Y I J" cannot go through the defined end point.
2. When programming an arc using "G09 X Y I J" the three points are in line or two of them are the same.
3. When trying to do a rounding tangential entry on a path that is not in the active plane.
4. When programming a tangential exit and the next path is tangent (being on its straight extension) to the path preceding the tangential exit.
   If the error comes up in the block calling the "Irregular canned cycle with islands" is because one of the cases mentioned earlier occurs in the set of blocks defining the profiles of a pocket with islands.

Solution The solution for each cause is:

1. Correct the syntax of the block. The coordinates of the end point or of the radius are defined wrong.

2. The three points used to define an arc must be different and cannot be in line.
3. Maybe a plane has been defined with "G16", "G17", "G18" or "G19". In this case, corner rounding, chamfers and tangential entries/exits can only be carried out on the main axes defining that plane. To do it in another plane, it must be defined beforehand.
4. The path after a tangential exit may be tangent, but it cannot be on the extension (in a straight line) of the previous path.

1085 'Helical path programmed wrong'

Detection During execution.

Cause When programming an arc using "G02/G03 X Y I J Z K", the programmed arc is impossible. The desired height cannot be reached with the programmed helical pitch.

Solution Correct the syntax of the block. The height of the interpolation and the coordinates of the end point in the plane must be related taking the helical pitch into account.

1086 'The spindle cannot be homed.'

Cause Spindle machine parameter REFEED1(P34) = 0.

1087 'Circle with zero radius'

Detection During execution.

Cause The various causes might be:

1. When programming an arc using "G02/G03 X Y I J", an arc has been programmed with a zero radius.
2. When operating with tool radius compensation, an inside arc has been programmed with the same radius as that of the tool.

Solution The solution for each cause is:

1. Arcs with zero radius are not allowed. Program a radius other than zero.
2. When working with tool radius compensation, the arc radius must larger than that of the tool. Otherwise, the tool cannot machine the programmed path (because to do so, the tool would have to make an arc of zero radius).

1088 'Range exceeded in zero offset.'

Detection During execution.

Cause A zero offset has been programmed and the value of the end position is too high.

Solution Check that the values assigned to the zero offsets (G54-G59) are correct. If the zero offsets have been assigned values from the program using parameters, check that the parameter values are correct. If an absolute (G54-G57) and an incremental (G58-G59) zero offset has been programmed, check that the sum of both does not exceed the machine limits.

1089 'Range exceeded in zone limit.'

Detection During execution.

Cause When programming zone limits "G20" or "G21" with parameters, the parameter value is greater than the maximum allowed for that function

Solution Check the program history to make sure that this parameter does not have that value when it reaches the block where the limits have been defined.

1090 'Point inside the forbidden zone 1.'

Detection During execution.

Cause An attempt has been made to move an axis to a point located inside the work area 1 that is defined as "no entry" zone.

Solution In the program history, work zone 1 (defined with G20/G21) has been set as "no entry" zone " (G22 K1 S1). To cancel this work zone, program "G22 K1 S0"

1091 'Point inside the forbidden zone 2.'

Detection During execution.

Cause An attempt has been made to move an axis to a point located inside the work area 2 that is defined as "no entry" zone.

Solution In the program history, work zone 2 (defined with G20/G21) has been set as "no entry" zone " (G22 K1 S1). To cancel this work zone, program "G22 K2 S0"

1092 'Insufficient acceleration for the speed programmed in threading.'

Detection During execution.

Cause A thread has been programmed and there isn't enough room to accelerate and decelerate.

Solution Program a lower speed.

1093 'Only one Hirth axis can be moved at a time'

No explanation required

1094 'Probe calibrated wrong'

No explanation required

1095 'Probing axes out of alignment .'

Detection During the probe calibration process

Cause An axis has moved to touch a cube and one of the axis that did not move registers a deflection greater than allowed by machine parameter MINDEFLE(P66). This is because the probing axes are not parallel enough to the axes of the machine.

Solution Correct the parallelism between the probing axes and those of the machine.

1096 'Point inside the forbidden zone 3.'

Detection During execution.

Cause An attempt has been made to move an axis to a point located inside the work area 3 that is defined as "no entry" zone.

Solution In the program history, work zone 3 (defined with G20/G21) has been set as "no entry" zone " (G22 K3 S1). To cancel this work zone, program "G22 K3 S0"

1097 'Point inside the forbidden zone 4.'

Detection During execution.

Cause An attempt has been made to move an axis to a point located inside the work area 4 that is defined as "no entry" zone.

Solution In the program history, work zone 4 (defined with G20/G21) has been set as "no entry" zone " (G22 K4 S1). To cancel this work zone, program "G22 K4 S0"

1098 'Work zone limits defined wrong'

Detection During execution.

Cause The upper limits (G21) of the defined work zone are the same or smaller than the lower ones (G20) of the same work zone.

Solution Program the upper limits (G21) of the work zone greater than the lower ones (G20).

1099 'Do not program a slaved axis.'

Detection During execution.

Cause When operating in polar coordinates, a movement has been programmed that involves an axis that is slaved to another one.

Solution The movements in polar coordinates are made with the main axes of the work plane; therefore, the axes that define the plane cannot be slaved to each other or to a third one. To unslave the axes, program "G78".

1100 'Travel limits of spindle 1 exceeded'

Detection During execution.

Cause An attempt has been made to exceed the physical turning limits of the spindle. As a result, the PLC activates the spindle mark "LIMIT+S" or "LIMIT-S". ("LIMIT+S2" or "LIMIT-S2" when working with the second spindle).

1101 'Spindle 1 locked'

Detection During execution.

Cause The CNC tries to output the command to the drive when the spindle input SERVOSON is still low. The error may be due to an error in the PLC program where this signal is not properly treated or that the value of the spindle parameter DWELL(P17) is not high enough.

1102 'Following error of spindle 1 out of limit'

Detection During execution.

Cause When the spindle is working in closed loop (M19), its following error is greater than the values indicated by spindle parameter MAXFLWE1(P21) and MAXFLE2(P22) The possible causes for this error are:

Servo drive error
- Faulty drive.
- Enable signals missing.
- Power supply missing.
- Drive adjusted incorrectly.
- The velocity command signal is not received.

Motor error
- Faulty motor.
- Power cables.

Feedback failure
- Defective feedback.
- Defective feedback cable.

Mechanical failure
- Mechanical stiffness.
- Spindle mechanically locked.

CNC error
- Defective CNC.
- Parameters adjusted incorrectly.

1103 'Do not synchronize spindles without homing them first'

Detection During execution.

Cause An attempt has been made to synchronize the spindle without homing them first.

Solution Before activating the synchronization, both spindles must be homed using the "M19" function.

1104 ' Do not program G28 or G29 while spindle synchronization is active'

Detection During execution.

Cause An attempt has been made to swap spindles (G28/G29) while the spindles were synchronized.

Solution First, cancel spindle synchronization (G78S).

1105 'Do not change the ranges (gears) while the spindle are synchronized'

Detection During execution.

Cause While the spindles are synchronized, a gear changing "M" function (M41 to M44) has been executed or the programmed "S" involves a gear change (with automatic gear changer).

Solution First, cancel spindle synchronization (G78S).

1106 'Travel limits of spindle 2 exceeded'

Same as error 1000, but for the second spindle.

1107 'Spindle 2 locked'

Same as error 1001, but for the second spindle.

1108 'Following error of spindle 2 out of limit'

Same as error 1002, but for the second spindle.

1109 'Axis software limit overrun'

No explanation required

1110-1118 'Range of the * axis exceeded'

Detection During execution.

Cause A movement has been defined with parameters and the parameter value is greater than the maximum travel distance of the axis.

Solution Check the program history to make sure that this parameter does not have that value when it reaches the block where this movement is programmed.

1119-1127 'The * axis cannot be synchronized'

Detection During execution.

Cause The various causes might be:

1. When trying to synchronize two axes from the PLC and one axis is already slaved to another one using the "G77" function.
2. When programming or trying to move an axis that is slaved to another one.

1128-1136 'Maximum feedrate of the * axis exceeded'

Detection During execution.

Cause The resulting feedrate of one of the axes after applying an individual scaling factor exceeds the maximum value indicated by axis machine parameter MAXFEED (P42).

1137-1145 'Wrong feedrate parameter of the * axis'

Detection During execution.

Cause "G00" programmed with parameter G00FEED(P38)=0 or "G1 F00" with axis parameter MAXFEED(P42) = 0.

1146-1154 '* axis locked'

Detection During execution.

Cause The CNC tries to output the command to the drive when the spindle input SERVO(n)ON is still low. The error may be due to an error in the PLC program where this signal is not properly treated or that the value of the axis parameter DWELL(P17) is not high enough.

1155-1163 'Maximum axis limits of the * axis exceeded'

Detection During execution.

Cause A coordinate has been programmed that is out of the limits defined by axis parameters LIMIT+(P5) and LIMIT-(P6).

1164-1172 'Work zone 1 of the * axis exceeded'

Detection During execution.

Cause An attempt has been made to move an axis to a point located out of the work area 1 that is defined as "no exit" zone.

Solution In the program history, work zone 1 (defined with G20/G21) has been set as "no exit" zone " (G22 K1 S2). To cancel this work zone, program "G22 K1 S0"

1173-1181 'Work zone 2 of the * axis exceeded'

Detection During execution.

Cause An attempt has been made to move an axis to a point located out of the work area 2 that is defined as "no exit" zone.

Solution In the program history, work zone 2 (defined with G20/G21) has been set as "no exit" zone " (G22 K2 S2). To cancel this work zone, program "G22 K2 S0"

1182-1190 'Following error of * axis out of limit'

Detection During execution.

Cause The following error of the axis is greater than the values indicated by axis parameter MAXFLWE1(P21) or maxflwe2(P22). The possible causes for this error are:

Servo drive error
- Faulty drive.
- Enable signals missing.
- Power supply missing.
- Drive adjusted incorrectly.
- The velocity command signal is not received.

Motor error
- Faulty motor.
- Power cables.

Feedback failure
- Defective feedback.
- Defective feedback cable.

Mechanical failure
- Mechanical stiffness.
- Spindle mechanically locked.

CNC error
- Defective CNC.
- Parameters adjusted incorrectly.

1191-1199 'Difference of following errors of the slaved axis * tool large '

Cause The "n" axis is electronically coupled to another one or is a slaved Gantry axis and the difference between the following errors of the "n" axis and the one it is coupled to is greater than the value set by the machine parameter for the "n" axis MAXCOUPE(P45).

1200-1208 'Travel limits of the * axis exceeded'

Detection During execution.

Cause An attempt has been made to exceed the physical travel limits. As a result, the PLC activates the axis mark "LIMIT+1" or "LIMIT-1".

1209-1217 '* axis servo error'

Cause The real feedrate of the axis, after the time indicated by axis parameter FBALTIME(P12), is below 50% or over 200% of the one programmed.

1218-1226 'Work zone 3 of the * axis exceeded'

Detection During execution.

Cause An attempt has been made to move an axis to a point located out of the work area 3 that is defined as "no exit" zone.

Solution In the program history, work zone 3 (defined with G20/G21) has been set as "no exit" zone " (G22 K3 S2). To cancel this work zone, program "G22 K3 S0"

1227 'Wrong profile intersection in pocket with islands.'

Detection During execution.

Cause In the "Irregular pocket canned cycle with islands (G66)", there are two plane profiles that either have the starting point or a section in common.

Solution Define the profiles again. Two plane profiles cannot start at the same point or have sections in common.

1228-1236 'Work zone 4 of the * axis exceeded'

Detection During execution.

Cause An attempt has been made to move an axis to a point located out of the work area 4 that is defined as "no exit" zone.

Solution In the program history, work zone 4 (defined with G20/G21) has been set as "no exit" zone " (G22 K4 S2). To cancel this work zone, program "G22 K4 S0"

1237 'Do not change the entry angle inside a thread'

Detection During execution.

Cause A thread joint has been defined and an entry angle "Q" has been programmed between two threads.

Solution When joining threads, only the first one may have an entry angle "Q".

1238 'Range of write-protected parameters. P297, P298'

Detection During execution.

Cause When trying to execute the function: "Definition of incline plane (G49)", parameters P297 and P298 are write-protected with machine parameters ROPARMIN(P51) and ROPARMAX(P52).

Solution While defining an incline plane, the CNC updates parameters P297 and P298. Therefore, these two parameters cannot be write-protected.

1239 'Point inside the forbidden zone 5.'

Detection During execution.

Cause An attempt has been made to move an axis to a point located inside the work area 5 that is defined as "no entry" zone.

Solution In the program history, work zone 5 (defined with G20/G21) has been set as "no entry" zone " (G22 K5 S1). To cancel this work zone, program "G22 K5 S0"

1240-1248 'Work zone 5 of the * axis exceeded'

Detection During execution.

Cause An attempt has been made to move an axis to a point located out of the work area 5 that is defined as "no exit" zone.

Solution In the program history, work zone 5 (defined with G20/G21) has been set as "no exit" zone " (G22 K5 S2). To cancel this work zone, program "G22 K5 S0"

1249 'Variable pitch thread programmed wrong'

Detection During execution.

Cause We are trying to make a variable-pitch thread with the following conditions:

- The "K" increment is positive and equal to or greater than 2L.
- The "K" increment is positive and with one of the calculated pitches, it exceeds the maximum feedrate (parameter MAXFEED) of one of the threading axis.
- The "K" increment is negative and one of the calculated pitches 0 or negative.

1250 'The K value is too large in G34'

Detection During execution.

Cause The ratio between the initial and final pitches of the variable-pitch thread (G34) to be executed

is greater than 32767.

1251 'Two variable-pitch threads cannot be joined in round corner'

Detection During motionless simulation, except when graphics are active

Cause To variable-pitch threads cannot be joined in round corner unless the second one is of the type: G34 … L0 K0.

G34 'G34 without a pitch is only allowed after a variable-pitch thread.

Detection During motionless simulation, except when graphics are active

Cause G34 L0 cannot be programmed after a movement, no G34, or in square corner.

1253 'Retrace function unavailable'

No explanation required

HARDWARE ERRORS

2000 'External emergency activated.'

Detection During execution.

Cause PLC input I1 is set to "0" (maybe the E-stop button) or the PLC mark M5000(/EMERGEN) is set to "0".

Solution Check at the PLC why the inputs are at "0". (Possible lack of power).

2001-2009 '* axis feedback error'

Detection During execution.

Cause The CNC does not receive feedback signal from the axes.

Solution Check that the connections are properly made.

NOTE: This error comes up on differential axes DIFFBACK(P9) =YES and sinusoidal axes SINMAGNI(P10) other than 0 when parameter FBACKAL(P11)=ON Setting parameter FBACKAL(P11)=OFF avoids this error, but this is only temporary solution.

2010 'Spindle feedback error'

Detection During execution.

Cause The CNC does not receive feedback signal from the spindle.

Solution Check that the connections are properly made.

NOTE: This error comes up on differential axes DIFFBACK(P14)=YES when parameter FBACKAL(P15)=ON. Setting parameter FBACKAL(P15)=OFF avoids this error, but this is only temporary solution.

2011 'Maximum temperature exceeded'

Detection Any time.

Cause The CNC's internal temperature has been exceeded. The causes may be:

- Electrical cabinet poorly ventilated.
- Axis board with some defective component.

Solution Turn the CNC and wait until it cools off. If the error persists, a component of the board may be defective. In that case, replace the board. Contact the Service Department.

2012 'There is no voltage at the axis board'

Detection During execution.

Cause 24V are missing at the output supply of the axis board. The fuse may be blown.

Solution Power the outputs of the axis board (24v). If the fuse is blown, replace it.

2013 'There is no voltage at the I/O 1 board.'
2014 'There is no voltage at the I/O 2 board.'
2015 'There is no voltage at the I/O 3 board.'

Detection During execution.

Cause 24V are missing at the output supply of the corresponding I/O board. The fuse may be blown.

Solution Power the outputs of the corresponding I/O board (24v). If the fuse is blown, replace it.

2016 'The PLC is not ready.'

Detection During execution.

Cause The PLC program is not running. These may be the probable causes:

- The PLC program is missing.
- WATCHDOG error.
- The program has been interrupted from monitoring.

Solution Start the PLC program. (Restart the PLC).

2017 'CNC RAM memory error.'

Detection While starting the CNC or during diagnoses.
Cause A defect has been found in the CNC's RAM memory.
Solution Replace the CPU board. Contact the Service Department.

2018 'CNC's EPROM memory error.'

Detection While starting the CNC or during diagnoses.
Cause A defect has been found in the CNC's EPROM memory.
Solution Replace the EPROM. Contact the Service Department.

2019 'PLC's RAM memory error.'

Detection While starting the CNC or during diagnoses.
Cause A defect has been found in the PLC's RAM memory.
Solution Replace the PLC board. Contact the Service Department.

2020 'PLC's EPROM memory error.'

Detection While starting the CNC or during diagnoses.
Cause A defect has been found in the PLC's EPROM memory.
Solution Replace the EPROM. Contact the Service Department.

2021 'CNC's user RAM memory error.' Press any key.'

Detection While starting the CNC or during diagnoses.
Cause A defect has been found in the CNC's user RAM memory.
Solution Contact the Service Department.

2022 'CNC's system RAM memory error.' Press any key.'

Detection While starting the CNC or during diagnoses.
Cause A defect has been found in the CNC's system RAM memory.
Solution Contact the Service Department.

2023 'PLC's RAM memory error.' Press any key.'

Detection While starting the CNC or during diagnoses.
Cause A defect has been found in the PLC's RAM memory.
Solution Contact the Service Department.

2024 'There is no voltage at the tracing board'

Detection During execution.
Cause 24V are missing at the output supply of the tracing board. The fuse may be blown.
Solution Power the outputs of the tracing board. If the fuse is blown, replace it.

2025 'Probe feedback error'

Detection During execution.
Cause The tracing probe is not connected or any of its cables is connected wrong.
Solution Check that the probe is properly connected.

2026 'Probe's maximum travel limit overrun.'

Detection During execution.
Cause The probe has exceeded the maximum deflection allowed by machine parameter.
Solution Decrease the feedrate and check that the probe has not been damaged.

2027 'SERCOS chip RAM memory error.' Press any key.'

Detection While starting the CNC or during diagnoses.
Cause A defect has been found in the SERCOS chip RAM memory.
Solution Replace the SERCOS board. Contact the Service Department.

2028 'SERCOS chip version error.' Press any key.'

Detection During CNC start-up.

Cause The SERCOS chip version is old.

Solution Replace the SERCOS chip. Contact the Service Department.

PLC ERRORS

3001 ' (PLC_ERR without description) '

Detection During execution.

Cause Marks ERR1 to ERR64 have been set to "1".

Solution Check at the PLC why these marks are set to "1" and act accordingly.

3002 'WATCHDOG in the main module (PRG).'

Detection Any time.

Cause The various causes might be:

1. The execution of the PLC's main program has exceeded the time set in PLC parameter WAGPRG(P0).
2. The program is in an endless loop.

Solution Increase the time of PLC parameter WAGPRG(P0) or increase the PLC speed.

- Insert CPU TURBO.
- Change PLC parameter CPUTIME(P26) or general parameter LOOPTIME(P72).

3003 'WATCHDOG in the periodic module (PE).'

Detection Any time.

Cause The various causes might be:

1. The execution of the PLC's periodic program has exceeded the time set in PLC parameter WAGPER(P1).
2. The program is in an endless loop.

Solution Increase the time of PLC parameter WAGPER(P1) or increase the PLC speed.

- Insert CPU TURBO.
- Change PLC parameter CPUTIME(P26) or general parameter LOOPTIME(P72).

3004 'Division by zero at the PLC'

Detection Any time.

Cause In the PLC program, there is a line whose execution implies a division by zero.

Solution When working with registers, that register may have already acquired a zero value. Check that the register does not reach the operation with that value.

3005 'PLC error -> '

Detection Any time.

Cause An error has been detected on the PLC board.

Solution Replace the PLC board. Contact the Service Department.

SERVO ERRORS

4000 'Sercos ring error'

Detection During execution.

Cause SERCOS communication has been interrupted. It may be caused by an interruption in the connection ring (optical fiber disconnected or broken) or by a wrong configuration.

1. The identifying wheel does not match the sercosid.
2. Parameter P120 (SERSPD) does not match the transmission speed.
3. The drive version is incompatible with the CNC.
4. There is an error on the SERCOS board.
5. Different transmission speed (baudrate) at the drive and at the CNC.

A drive has been turned off and back on due to a power supply failure. When starting up again, it displays the error ***4027 'The drive has started up again'***

An attempt has been made to read or write an non-existent variable or too many variables in a drive through the fast channel.

Solution To check that the connection ring is not interrupted, check that the light goes through the optical fiber. If it is due to a wrong configuration, contact the Service Department.

If the error is due to the fast channel

- Check that all the variables to be read or written through the fast channel actually exist
- Save the SERCOS LOG into a file and see which axis causes the error.
- Set PLC machine parameters "SRD700 and SWR800" of that drive to "0".
- Reset the CNC and verify that no errors come up.
- Set the parameters one by one to the desired value until the failure occurs.
- When locating the parameter, look that variable up in the drive manual to verify that it exists in that version and it may be accessed. If so, the error may come up because it tries read or write too many variables in that drive.

4001 'Undefined class 1 error'

Detection During execution.

Cause The drive has detected an error, but it cannot identify it.

Solution Contact the Service Department.

4002 'Overload (201...203)'
4003 'Overtemperature at the drive (107)'
4004 'Overtemperature at the motor (108)'
4005 'Overtemperature at the heatsink (106)'
4006 'Voltage control error (100...105)'
4007 'Feedback error (600...606)'
4008 'Error at the power bus (213...215)'
4009 'Overcurrent (212)'
4010 'Overvoltage at the power bus (304/306)'
4011 'Undervoltage at the power bus (307)'

Detection During execution.

Cause An error occurred at the drive. The number in brackets indicates the standard error number of the drive. Refer to the drive manual for further information.

Solution These types of error come with the messages 4019, 4021, 4022 or 4023 that indicate in which axis or spindle drive the error came up. Refer to the drive manual to check the error (number in brackets) and act accordingly.

4012 'Drive error'
4013 'Position deviation too high'
4014 'Communications error'
4015 'Travel limit overrun'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4016 ‘Undefined class 1 error’

Detection During execution.

Cause The drive has detected an error, but it cannot identify it.

Solution Contact the Service Department.

4017 ‘Drive error’

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4018 ‘Error accessing a SERCOS variable’

Detection During execution.

Cause An attempt has been made to read (or write) a SERCOS variable from the CNC and:

1. That variable does not exist.
2. The maximum/minimum values have been exceeded.
3. The SERCOS variable has a variable length.
4. An attempt has been made to write a read-only variable.

Solution Check that the variable to be associated with an action is of the right type.

4019 ‘Drive error : Axis ‘

Detection During execution.

Cause These messages come with errors 4002 – 4011. When one of the mentioned errors occurs, they indicate in which axis it came up.

4020 ‘SERCOSID parameter value error’

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4021 ‘Spindle drive error :
4022 ‘Spindle-2 drive error :
4023 ‘Auxiliary spindle drive error’

Detection During execution.

Cause These messages come with errors 4002 – 4011. When one of the mentioned errors occurs, they indicate in which spindle it came up.

4024 ‘SERCOS error when searching home’

Detection During execution.

Cause The home search command of SERCOS has been executed incorrectly.

4025 ‘SERCOS loop time exceeded: Increase P72 (looptime)‘

Detection During execution.

Cause The time it takes to calculate the feedrate of the axis is greater than the cycle time established for transmission to the drive.

Solution Increase the value of general machine parameter LOOPTIME (P72). If the error persists, contact the Service Department.

4026 ‘Error in SERCOS chip RAM memory’

Detection During execution.

Solution Contact the service department to replace the SERCOS board.

4027 ‘The drive has started up again’

Detection During execution.

Cause A drive has been turned off and back on due to a power supply failure.

4028 ‘The light does not reach the CNC through the optic fiber’

Detection On power-up.

Cause The signal sent by the CNC through the optical fiber does not return to the CNC.

Solution Check the condition and installation of the fiber optic cables. Check that the light going "OUT" of the CNC is going through the drives and comes "IN"to the CNC.

If the cables are OK, remove the drives from the ring until the error no longer comes up.

4029 'Communication with the drive cannot be established. No response'

Detection On power-up.

Cause A drive is not responding to the signal sent by the CNC due to one of these causes:

- The drive does not recognize the sercos board.
- The drive is locked up
- The switch number has not been properly read.
- The SERCOS transmission speed has been set differently at the drives and at the CNC. General parameter SERSPD at the CNC and QP11 at the drives.

Solution Save the SERCOS LOG into a file.

See the value of axis machine parameter SERCOSID of the axis causing the error.

Check that the ring contains a drive with the switch in that position.

Reset the drive because the drive only reads the switch on power-up.

Check that the CNC and the drives have the same transmission speed. General parameter SERSPD at the CNC and QP11 at the drives.

Check that the drive does not issue sercos board. To do that look at the display of the drive. If it shows hardware errors, change the drive's sercos board.

If there are no errors at that drive, set the switch of the drive to "1", reset it, set the CNC with a single Sercos axis and connect to the CNC. If it still issues the error, change the drive.

4030 'SERCON register writing error'

Detection During execution.

Solution Contact the Service Department.

4050 'ERROR 1: Internal (Fatal error): Internal RAM test failed'
4051 'ERROR 2 : Internal (Fatal error): internal program malfunction'
4052 'ERROR 3 : Power bus drop: No torque. '
4053 'ERROR 4 : The emergency stop cannot stop the motor in the established time frame'
4054 'ERROR 5 : Program code checksum error'
4055 'ERROR 6 : Sercos board error'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4056 'ERROR 100 : Internal +5 V out of range'
4057 'ERROR 101 : Internal -5 V out of range'
4058 'ERROR 102 : Internal +8 V out of range'
4059 'ERROR 103 : Internal -8 V out of range'
4060 'ERROR 104 : Internal +18 V out of range'
4061 'ERROR 105 : Internal -18 V out of range'
4062 'ERROR 106 : Heat-sink overheating'
4063 'ERROR 107 : VeCon card overheating'
4064 'ERROR 108 : Motor overheating'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4065 'ERROR 200 : Overspeed'
4066 'ERROR 201 : Motor overload'
4067 'ERROR 202 : Driver overload'
4068 'ERROR 211 : Internal (Fatal error): DSP program execution error'
4069 'ERROR 212 : Overcurrent'
4070 'ERROR 213 : Undervoltage at the IGBT power driver'
4071 'ERROR 214: Shortcircuit'
4072 'ERROR 215 : Overvoltage at the power bus(Hard) '

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4073 'ERROR 300 : Power supply module overheating'
4074 'ERROR 301 : Power supply module ballast circuit overheating'
4075 'ERROR 302 : Shortcircuit in the power supply module ballast'
4076 'ERROR 303 : Ballast circuit supply voltage out of range'
4077 'ERROR 304 : Overvoltage at the power bus detected by the power supply module'
4078 'ERROR 305 : Protocol error in the interface between the power supply module and the driver'
4079 'ERROR 306 : Overvoltage at the power bus (Soft, trigger prior to hardware)'
4080 'ERROR 307 : Undervoltage of the power bus'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4081 'ERROR 400 : No SERCOS board is detected'
4082 'ERROR 401 : Internal SERCOS error'
4083 'ERROR 403 : MST failure'
4084 'ERROR 404 : MDT failure'
4085 'ERROR 405 : Wrong phase (> 4)'
4086 'ERROR 406 : Wrong phase increase'
4087 'ERROR 407 : Wrong phase decrease'
4088 'ERROR 408 : Phase change without «ready» aknowledgement'
4089 'ERROR 409 : Change to an unitialized phase'
4090 'ERROR 410 : Two drivers have the same ring address'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4091 'ERROR 500 : Inconsistent parameters'
4092 'ERROR 501 : Parameter checksum error'
4093 'ERROR 502 : Wrong parameter value'
4094 'ERROR 503 : The table for default parameter values for each motor is wrong. '
4095 'ERROR 504 : Wrong parameter in SERCOS phase 2'
4096 'ERROR 505 : Different RAM and Flash parameters'
4097 'ERROR 600 : Communication error with the second feedback'
4098 'ERROR 601 : Communication error with the rotor encoder'
4099 'ERROR 602 : motor feedback B signal saturation'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4100 'ERROR 603 : motor feedback A signal saturation '
4101 'ERROR 604 : Saturation of A and/or B signal values'
4102 'ERROR 605 : Week A and/or B signal values'
4103 'ERROR 606 : Too much dispersion of the rotor sensor signals'
4104 'ERROR 700 : RS232 board error'
4105 'ERROR 701 : Internal : Wrong VeCon board identification'
4106 'ERROR 702 : Expansion board identification error'
4107 'ERROR 703 : I/O board identification error'
4108 'ERROR 704 : Analog board identification error'
4109 'ERROR 705 : Power board identification error'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4110 'ERROR 706 : X3 encoder simulation board identification error'
4111 'ERROR 707 : X4 motor feedback board identification error'
4112 'ERROR 801 : Encoder not detected'
4113 'ERROR 802 : Communication error with the encoder'
4114 'ERROR 803 : Unitialized encoder'
4115 'ERROR 804 : Defective encoder'
4116 'ERROR 805 : No encoder has been detected on the motor'
4117 'ERROR 7 : SERCON clock error'
4118 'ERROR 8 : SERCON data error'
4119 'ERROR 203 : Torque overload error'
4120 'ERROR 411 : telegram reception error'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

CAN ERRORS

‘5003 Application error

Cause Internal CANopen error

Solution Contact the Service Department.

5004 CAN bus error

Cause The error type is indicated with a code:

2	Transmission queue full, the message cannot be sent.
128	Bus Off, the bus has been deactivated due to too many errors.
129	CAN warning, there are more than 96 errors at the bus, step prior to the buss off error.
130	Loss of message received or too many messages received. Usually due to wrong speed for the cable length.
131	The CNC has switched to an inoperative state in the bus (internal).

Solution The solution for each cause is:

2	Check the connection between the CNC and first node.
128	Check cables and connections.
129	Check cables and connections.
130	Check machine parameter IOCANSPE (P88).
131	Check cables and connections.

5005 Presence control error detected by the CNC

Cause The CNC detects that the node has reset itself or is connected wrong.

Solution Check cables and connections.

5006 Error because the node has been reset

Cause The node has been reset due to a power supply failure

Solution Check the power supply voltage at the indicated node, the ground connection and the load of the outputs.

5007 Error message corrected

Cause It is activated when an error situation disappears and shows whether there are any more left. If there is none, it resets the node connections.

5022 Internal software error

Cause Internal node software error.

Solution Access the Status screen \ Can \ Versions and reload the software.

5027 Communications error

Cause Node communication error

Solution Contact the Service Department.

5028 Lost messages

Cause The node has lost messages.

Solution Check cables and connections.

5029 Presence control error detected by the node

Cause The presence control done by the CNC node has failed.

Solution Check cables and connections.

5030 Protocol error

Cause The node has received a message that it cannot interpret

Solution Contact the Service Department.

5031 PDO not processed due to its length.

Cause The node has received a process message whose length does not match

Solution Contact the Service Department.

5032 PDO too long

Cause The node has received a process message longer than the one programmed

Solution Contact the Service Department.

5036 Output over-current

Cause Excessive consumption (over current) has been detected in the outputs of the indicated node. As a precaution, the system deactivates all the outputs of this module setting them to zero volts.

Solution Check the consumption and possible short-circuits at the outputs of the module.

5037 Power supply voltage error

Cause A power supply failure has been detected at the indicated node, it has no power or it is under +24V.

Solution Check the supply voltage at the outputs and the consumption of the module's supply voltage.

TABLE DATA ERRORS

'CHECKSUM ERROR: GENERAL PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR: SPINDLE PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:2nd SPINDLE PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:AUX. SPINDLE PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:SERIAL LINE 1 PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:SERIAL LINE 2 PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:HD/ETHERNET PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:USER PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:OEM PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:PLC PARAMETERS Load CARD A? (ENTER/ESC)'

'CHECKSUM ERROR:ZERO OFFSET TABLE Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:PASSWORD TABLE Load CARD A? (ENTER/ESC)'

'CHECKSUM ERROR:AXIS * PARAMETERS Load CARD A? (ENTER/ESC)'

'CHECKSUM ERROR:TOOL TABLE Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:TOOL OFFSET TABLE Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:MAGAZINE TABLE Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:M FUNCTION TABLE Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:LEADSCREW * TABLE Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:CROSS COMP. TABLE * Load CARD A? (ENTER/ESC)'

Detection During CNC start-up.

Cause Certain table data has been lost (possible RAM error) and there is a table saved in CARD A.

Solution Pressing [ENTER] copies the table saved in CARD A to RAM memory. If the error persists, contact the service department.

'ERROR:GENERAL PARAMETER CHECKSUM Reset? (ENTER/ESC)'
'ERROR:SPINDLE PARAMETER CHECKSUM Reset? (ENTER/ESC)'
'ERROR:SPINDLE-2 PARAMETER CHECKSUM Reset? (ENTER/ESC)'
'ERROR:AUX. SPINDLE PARAMETER CHECKSUM Reset? (ENTER/ESC)'
'ERROR:SERIAL-LINE-1 PARAMETER CHECKSUM Reset? (ENTER/ESC)'
'ERROR:SERIAL-LINE-2 PARAMETER CHECKSUM Reset? (ENTER/ESC)'
'CHECKSUM ERROR:HD/ETHERNET PARAMETERS Initialize? (ENTER/ESC)'
'CHECKSUM ERROR:USER PARAMETERS Initialize? (ENTER/ESC)'
'CHECKSUM ERROR:OEM PARAMETERS Initialize? (ENTER/ESC)'
'ERROR:PLC PARAMETER CHECKSUM Reset? (ENTER/ESC)'

'ERROR:ZERO OFFSET TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR:CODE TABLE CHECKSUM Reset? (ENTER/ESC)'

'ERROR:AXIS PARAMETER CHECKSUM Reset? (ENTER/ESC)'

'ERROR:TOOL TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR:TOOL OFFSET TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR:TOOL MAGAZINE TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR:M FUNCTION TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR:AXIS LEADSCREW TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR:CROSS COMP. TABLE CHECKSUM Reset? (ENTER/ESC)'

Detection During CNC start-up.

Cause Certain table data has been lost (possible RAM error) and there is no table saved in CARD A.

Solution Pressing [ENTER] loads the tables with CNC's default values. If the error persists, contact the Service Department.

'Wrong * leadscrew table. Press key'

Detection During CNC start-up.

Cause There is some erroneous data in the parameters of the leadscrew compensation table.

Solution The definition of the points of the table must meet the following requirements:

- The points of the table must be ordered according to their position on the axis, starting from the most negative or less positive point to be compensated.
- The machine reference point must have no error (zero).
- The error difference between consecutive points cannot be greater than the distance between them.

'Wrong * cross compensation table. Press key'

Detection During CNC start-up.

Cause There is some erroneous data in the parameters of the cross compensation table.

Solution The definition of the points of the table must meet the following requirements:

- The points of the table must be ordered according to their position on the axis, starting from the most negative or less positive point to be compensated.
- The machine reference point must have no error (zero).

'Incorrect cross compensation table parameters'

Detection During CNC start-up.

Cause The parameters indicating the axes involved in the cross compensation are defined wrong.

Solution Maybe a nonexistent axis has been defined or the affected axis (to be compensated) and the one affecting it are the same.

'Wrong axis or spindle parameters sercosid'

Detection During CNC start-up.

Cause The servosid parameters have not been entered correctly.

Solution The rules of sercosid parameters are:

- They must begin with number 1.
- They must be consecutive.
- They cannot be repeated.

9001 'CENTER PUNCHING: F=0'

Detection During execution.
Cause A feedrate "F" has been defined with a wrong value.
Solution Program a positive feedrate "F" other than zero.

9002 'CENTER PUNCHING: S=0'

Detection During execution.
Cause A spindle speed "S" has been defined with a wrong value.
Solution Program a positive spindle speed "S" other than zero.

9003 'CENTER PUNCHING: T=0'

Detection During execution.
Cause The tool number "T" has not been defined.
Solution The tool number "T" must be other than zero.

9004 'CENTER PUNCHING: P=0'

Detection During execution.
Cause The center punching depth "P" has not been defined.
Solution The center punching depth "P" must be other than zero.

9005 'CENTER PUNCHING: ø=0'

Detection During execution.
Cause The point diameter «ø» has not been defined.
Solution The point diameter «ø» must be positive and other than zero.

9006 'CENTER PUNCHING: α=0'

Detection During execution.
Cause The angle of the tip of the drill bit has not been «α».
Solution The angle of the tip of the drill bit «α» must be positive and other than zero.

9007 'DRILLING 1: F=0'

Detection During execution.
Cause A feedrate "F" has been defined with a wrong value.
Solution Program a positive feedrate "F" other than zero.

9008 'DRILLING 1: S=0'

Detection During execution.
Cause A spindle speed "S" has been defined with a wrong value.
Solution Program a positive spindle speed "S" other than zero.

9009 'DRILLING 1: T=0'

Detection During execution.
Cause The tool number "T" has not been defined.
Solution The tool number "T" must be other than zero.

9010 'DRILLING 1: P=0'

Detection During execution.
Cause The center drilling depth "P" has not been defined.
Solution The drilling depth "P" must be other than zero.

9011 'DRILLING 2: F=0'

Detection During execution.

Cause A feedrate “F” has been defined with a wrong value.

Solution Program a positive feedrate “F” other than zero.

9012 ‘DRILLING 2: S=0’

Detection During execution.

Cause A spindle speed “S” has been defined with a wrong value.

Solution Program a positive spindle speed “S” other than zero.

9013 ‘DRILLING 2: T=0’

Detection During execution.

Cause The tool number “T” has not been defined.

Solution The tool number “T” must be other than zero.

9014 ‘DRILLING 2: P=0’

Detection During execution.

Cause The center drilling depth “P” has not been defined.

Solution The drilling depth “P” must be other than zero.

9015 ‘DRILLING 2: B=0’

Detection During execution.

Cause The withdrawal distance «B» after each penetration has not been defined.

Solution The distance «B» it withdraws after each penetration must be other than zero.

9016 ‘TAPPING: F=0’

Detection During execution.

Cause A feedrate “F” has been defined with a wrong value.

Solution Program a positive feedrate “F” other than zero.

9017 ‘TAPPING: S=0’

Detection During execution.

Cause A spindle speed “S” has been defined with a wrong value.

Solution Program a positive spindle speed “S” other than zero.

9018 ‘TAPPING: T=0’

Detection During execution.

Cause The tool number “T” has not been defined.

Solution The tool number “T” must be other than zero.

9019 ‘TAPPING: P=0’

Detection During execution.

Cause The tapping depth “P” has not been defined.

Solution The tapping depth “P” must be other than zero.

9020 ‘REAMING: F=0’

Detection During execution.

Cause A feedrate “F” has been defined with a wrong value.

Solution Program a positive feedrate “F” other than zero.

9021 ‘REAMING: S=0’

Detection During execution.

Cause A spindle speed “S” has been defined with a wrong value.

Solution Program a positive spindle speed “S” other than zero.

9022 ‘REAMING: T=0’

Detection During execution.

Cause The tool number “T” has not been defined.

Solution The tool number “T” must be other than zero.

9023 ‘REAMING: P=0’

Detection During execution.
Cause The reaming depth “P” has not been defined.
Solution The reaming depth “P” must be other than zero.

9024 ‘BOARING: F=0’

Detection During execution.
Cause A feedrate “F” has been defined with a wrong value.
Solution Program a positive feedrate “F” other than zero.

9025 ‘BOARING: S=0’

Detection During execution.
Cause A spindle speed “S” has been defined with a wrong value.
Solution Program a positive spindle speed “S” other than zero.

9026 ‘BOARING: T=0’

Detection During execution.
Cause The tool number “T” has not been defined.
Solution The tool number “T” must be other than zero.

9027 ‘BOARING: P=0’

Detection During execution.
Cause The boaring depth “P” has not been defined.
Solution The boring depth “P” must be other than zero.

9028 ‘DRILLING 3: F=0’

Detection During execution.
Cause A feedrate “F” has been defined with a wrong value.
Solution Program a positive feedrate “F” other than zero.

9029 ‘DRILLING 3: S=0’

Detection During execution.
Cause A spindle speed “S” has been defined with a wrong value.
Solution Program a positive spindle speed “S” other than zero.

9030 ‘DRILLING 3: T=0’

Detection During execution.
Cause The tool number “T” has not been defined.
Solution The tool number “T” must be other than zero.

9031 ‘DRILLING 3: P=0’

Detection During execution.
Cause The center drilling depth “P” has not been defined.
Solution The drilling depth “P” must be other than zero.

9032 ‘BOARING 2: F=0’

Detection During execution.
Cause A feedrate “F” has been defined with a wrong value.
Solution Program a positive feedrate “F” other than zero.

9033 ‘BOARING 2: S=0’

Detection During execution.
Cause A spindle speed “S” has been defined with a wrong value.
Solution Program a positive spindle speed “S” other than zero.

9034 'BOARING 2: T=0'

Detection During execution.
Cause The tool number "T" has not been defined.
Solution The tool number "T" must be other than zero.

9035 'BOARING 2: P=0'

Detection During execution.
Cause The boaring depth "P" has not been defined.
Solution The boring depth "P" must be other than zero.

9036 'RECTANGULAR POCKET 1: F=0'

Detection During execution.
Cause A feedrate "F" has been defined with a wrong value.
Solution Program a positive feedrate "F" other than zero.

9037 'RECTANGULAR POCKET 1: S=0'

Detection During execution.
Cause A spindle speed "S" has been defined with a wrong value.
Solution Program a positive spindle speed "S" other than zero.

9038 'RECTANGULAR POCKET 1: T=0'

Detection During execution.
Cause The tool number "T" has not been defined.
Solution The tool number "T" must be other than zero.

9039 'RECTANGULAR POCKET 1: P=0'

Detection During execution.
Cause The pocket depth "P" has not been defined.
Solution The pocket depth "P" must be other than zero.

9040 'RECTANGULAR POCKET 1: Tool diameter smaller than Δ'

Detection During execution.
Cause The programmed milling step «Δ» is larger than the tool diameter.
Solution Program a milling step «Δ» smaller than the tool diameter or choose a tool of larger diameter.

9041 'RECTANGULAR POCKET 1: Tool diameter larger than the pocket'

Detection During execution.
Cause The tool diameter is larger than one of the pocket's "H" or "L" dimensions.
Solution Choose a tool of smaller diameter to mill the pocket.

9042 'RECTANGULAR POCKET 1: FINISHING tool diameter smaller than δ'

Detection During execution.
Cause The programmed finishing stock «δ» is larger than the tool diameter.
Solution Program a finishing stock «δ» smaller than the tool diameter or choose a tool of larger diameter.

9043 'RECTANGULAR POCKET 2: F=0'

Detection During execution.
Cause A feedrate "F" has been defined with a wrong value.
Solution Program a positive feedrate "F" other than zero.

9044 'RECTANGULAR POCKET 2: S=0'

Detection During execution.
Cause A spindle speed "S" has been defined with a wrong value.
Solution Program a positive spindle speed "S" other than zero.

9045 'RECTANGULAR POCKET 2: P=0'

Detection During execution.

Cause The pocket depth "P" has not been defined.

Solution The pocket depth "P" must be other than zero.

9046 'RECTANGULAR POCKET 2: Wrong penetration angle'

Detection During execution.

Cause A penetration angle smaller than 0º and greater than 90º has been programmed

Solution Program a penetration angle «β» and «θ» between 0º and 90º.

9047 'RECTANGULAR POCKET 2: Tool diameter smaller than Δ'

Detection During execution.

Cause The programmed milling step «Δ» is larger than the tool diameter.

Solution Program a milling step «Δ» smaller than the tool diameter or choose a tool of larger diameter.

9048 'RECTANGULAR POCKET 2: Tool diameter larger than the pocket'

Detection During execution.

Cause The tool diameter is larger than one of the pocket's "H" or "L" dimensions.

Solution Choose a tool of smaller diameter to mill the pocket.

9049 'RECTANGULAR POCKET 2: FINISHING tool diameter smaller than δ'

Detection During execution.

Cause The programmed finishing stock «δ» is larger than the tool diameter.

Solution Program a finishing stock «δ» smaller than the tool diameter or choose a tool of larger diameter.

9050 'CIRCULAR POCKET 1: F=0'

Detection During execution.

Cause A feedrate "F" has been defined with a wrong value.

Solution Program a positive feedrate "F" other than zero.

9051 'CIRCULAR POCKET 1: S=0'

Detection During execution.

Cause A spindle speed "S" has been defined with a wrong value.

Solution Program a positive spindle speed "S" other than zero.

9052 'CIRCULAR POCKET 1: P=0'

Detection During execution.

Cause The pocket depth "P" has not been defined.

Solution The pocket depth "P" must be other than zero.

9053 'CIRCULAR POCKET 1: Wrong penetration angle'

Detection During execution.

Cause A penetration angle smaller than 0º and greater than 90º has been programmed

Solution Program a penetration angle «β» and «θ» between 0º and 90º.

9054 'CIRCULAR POCKET 1: Tool diameter smaller than Δ'

Detection During execution.

Cause The programmed milling step «Δ» is larger than the tool diameter.

Solution Program a milling step «Δ» smaller than the tool diameter or choose a tool of larger diameter.

9055 'CIRCULAR POCKET 1: Tool diameter larger than the pocket'

Detection During execution.

Cause The tool radius is greater than the pocket radius "R".

Solution Choose a tool of smaller diameter to mill the pocket.

9056 'CIRCULAR POCKET 1: FINISHING tool diameter smaller than δ'

Detection During execution.

Cause The programmed finishing stock «δ» is larger than the tool diameter.

Solution Program a finishing stock «δ» smaller than the tool diameter or choose a tool of larger diameter.

9057 'CIRCULAR POCKET 2: F=0'

Detection During execution.

Cause A feedrate "F" has been defined with a wrong value.

Solution Program a positive feedrate "F" other than zero.

9058 'CIRCULAR POCKET 2: S=0'

Detection During execution.

Cause A spindle speed "S" has been defined with a wrong value.

Solution Program a positive spindle speed "S" other than zero.

9059 'CIRCULAR POCKET 2: P=0'

Detection During execution.

Cause The pocket depth "P" has not been defined.

Solution The pocket depth "P" must be other than zero.

9060 'CIRCULAR POCKET 2: Wrong penetration angle'

Detection During execution.

Cause A penetration angle smaller than 0º and greater than 90º has been programmed

Solution Program a penetration angle «β» and «θ» between 0º and 90º.

9061 'CIRCULAR POCKET 2: Tool radius greater than Ri'

Detection During execution.

Cause A tool has been selected with a radius greater than Ri (inside radius).

Solution Select a tool with a smaller diameter.

9062 'CIRCULAR POCKET 2: Tool diameter smaller than Δ'

Detection During execution.

Cause The programmed milling step «Δ» is larger than the tool diameter.

Solution Program a milling step «Δ» smaller than the tool diameter or choose a tool of larger diameter.

9063 'CIRCULAR POCKET 2: Tool diameter larger than the pocket'

Detection During execution.

Cause The tool radius is greater than the pocket radius "R".

Solution Choose a tool of smaller diameter to mill the pocket.

9064 'CIRCULAR POCKET 2: FINISHING tool diameter smaller than δ'

Detection During execution.

Cause The programmed finishing stock «δ» is larger than the tool diameter.

Solution Program a finishing stock «δ» smaller than the tool diameter or choose a tool of larger diameter.

9065 'CIRCULAR POCKET 2: Ri > Re'

Detection During execution.

Cause An inside radius (Ri) has been programmed greater than the outside (Re).

9066 'RECTANGULAR BOSS: F=0'

Detection During execution.

Cause A feedrate "F" has been defined with a wrong value.

Solution Program a positive feedrate "F" other than zero.

9067 'RECTANGULAR BOSS: S=0'

Detection During execution.

Cause A spindle speed "S" has been defined with a wrong value.

Solution Program a positive spindle speed "S" other than zero.

9068 'RECTANGULAR BOSS: P=0'

Detection During execution.

Cause The boss depth "P" has not been defined.

Solution The boss height "P" must be other than zero.

9069 'RECTANGULAR BOSS: Tool diameter smaller than Δ'

Detection During execution.

Cause The programmed milling step «Δ» is larger than the tool diameter.

Solution Program a milling step «Δ» smaller than the tool diameter or choose a tool of larger diameter.

9070 'RECTANGULAR BOSS: FINISHING tool diameter smaller than δ'

Detection During execution.

Cause The programmed finishing stock «δ» is larger than the tool diameter.

Solution Program a finishing stock «δ» smaller than the tool diameter or choose a tool of larger diameter.

9071 'CIRCULAR BOSS: F=0'

Detection During execution.

Cause A feedrate "F" has been defined with a wrong value.

Solution Program a positive feedrate "F" other than zero.

9072 'CIRCULAR BOSS: S=0'

Detection During execution.

Cause A spindle speed "S" has been defined with a wrong value.

Solution Program a positive spindle speed "S" other than zero.

9073 'CIRCULAR BOSS: P=0'

Detection During execution.

Cause The boss depth "P" has not been defined.

Solution The boss height "P" must be other than zero.

9074 'CIRCULAR BOSS: Tool diameter smaller than Δ'

Detection During execution.

Cause The programmed milling step «Δ» is larger than the tool diameter.

Solution Program a milling step «Δ» smaller than the tool diameter or choose a tool of larger diameter.

9075 'CIRCULAR BOSS: FINISHING tool diameter smaller than δ'

Detection During execution.

Cause The programmed finishing stock «δ» is larger than the tool diameter.

Solution Program a finishing stock «δ» smaller than the tool diameter or choose a tool of larger diameter.

9076 'PROFILE POCKET: F=0'

Detection During execution.

Cause A feedrate "F" has been defined with a wrong value.

Solution Program a positive feedrate "F" other than zero.

9077 'PROFILE POCKET: S=0'

Detection During execution.

Cause A spindle speed "S" has been defined with a wrong value.

Solution Program a positive spindle speed "S" other than zero.

9078 'PROFILE POCKET: P=0'

Detection During execution.

Cause The pocket depth "P" has not been defined.

Solution The pocket depth "P" must be other than zero.

9079 'PROFILE POCKET: Wrong penetration angle'

Detection During execution.

Cause A penetration angle smaller than 0º and greater than 90º has been programmed

Solution Program a penetration angle «β» and «θ» between 0º and 90º.

9080 'PROFILE POCKET: Tool diameter smaller than Δ'

Detection During execution.

Cause The programmed milling step «Δ» is larger than the tool diameter.

Solution Program a milling step «Δ» smaller than the tool diameter or choose a tool of larger diameter.

9081 'PROFILE POCKET: FINISHING tool diameter smaller than δ'

Detection During execution.

Cause The programmed finishing stock «δ» is larger than the tool diameter.

Solution Program a finishing stock «δ» smaller than the tool diameter or choose a tool of larger diameter.

9082 '3D PROFILE POCKET: F=0'

Detection During execution.

Cause A feedrate "F" has been defined with a wrong value.

Solution Program a positive feedrate "F" other than zero.

9083 '3D PROFILE POCKET: S=0'

Detection During execution.

Cause A spindle speed "S" has been defined with a wrong value.

Solution Program a positive spindle speed "S" other than zero.

9084 '3D PROFILE POCKET: P=0'

Detection During execution.

Cause The pocket depth "P" has not been defined.

Solution The pocket depth "P" must be other than zero.

9085 '3D PROFILE POCKET: Wrong penetration angle'

Detection During execution.

Cause A penetration angle smaller than 0º and greater than 90º has been programmed

Solution Program a penetration angle «β» and «θ» between 0º and 90º.

9086 '3D PROFILE POCKET: Tool diameter smaller than Δ'

Detection During execution.

Cause The programmed milling step «Δ» is larger than the tool diameter.

Solution Program a milling step «Δ» smaller than the tool diameter or choose a tool of larger diameter.

9087 '3D PROFILE POCKET: FINISHING tool diameter smaller than δ'

Detection During execution.

Cause The programmed finishing stock «δ» is larger than the tool diameter.

Solution Program a finishing stock «δ» smaller than the tool diameter or choose a tool of larger diameter.

9088 '3D PROFILE POCKET: FINISHING tool radius smaller than R'

Detection During execution.

Cause The radius of the finishing tool is smaller than R (finishing tool tip radius).

Solution Select a tool with a larger diameter.

9089 'SURFACE MILLING: F=0'

Detection During execution.

Cause A feedrate "F" has been defined with a wrong value.

Solution Program a positive feedrate "F" other than zero.

9090 'SURFACE MILLING: S=0'

Detection During execution.

Cause A spindle speed "S" has been defined with a wrong value.

Solution Program a positive spindle speed "S" other than zero.

9091 'SURFACE MILLING: T=0'

Detection During execution.

Cause The tool number "T" has not been defined.

Solution The tool number "T" must be other than zero.

9092 'SURFACE MILLING: P=0'

Detection During execution.

Cause The depth "P" of the surface milling has not been defined.

Solution The surface milling depth "P" must be other than zero.

Errors in the profile milling operation 1.

9093 'PROFILE MILLING 1: F=0'

Detection During execution.

Cause A feedrate "F" has been defined with a wrong value.

Solution Program a positive feedrate "F" other than zero.

9094 'PROFILE MILLING 1: S=0'

Detection During execution.

Cause A spindle speed "S" has been defined with a wrong value.

Solution Program a positive spindle speed "S" other than zero.

9095 'PROFILE MILLING 1: T=0'

Detection During execution.

Cause The tool number "T" has not been defined.

Solution The tool number "T" must be other than zero.

9096 'PROFILE MILLING 1: P=0'

Detection During execution.

Cause The milling depth "P" has not been defined.

Solution The milling depth "P" must be other than zero.

9097 'PROFILE MILLING 1: Null profile'

Detection During execution.

Cause The profile to be machined has not been defined.

Solution The profile must consist of at least two points besides the entry one and the exit one.

9098 ‘PROFILE MILLING 2: T=0’

Detection During execution.

Cause The tool number “T” has not been defined.

Solution The tool number “T” must be other than zero.

9099 ‘PROFILE MILLING 2: F=0’

Detection During execution.

Cause A feedrate “F” has been defined with a wrong value.

Solution Program a positive feedrate “F” other than zero.

9100 ‘PROFILE MILLING 2: S=0’

Detection During execution.

Cause A spindle speed “S” has been defined with a wrong value.

Solution Program a positive spindle speed “S” other than zero.

9101 ‘PROFILE MILLING 2: P=0’

Detection During execution.

Cause The milling depth “P” has not been defined.

Solution The milling depth “P” must be other than zero.

9102 ‘SLOT MILLING: F=0’

Detection During execution.

Cause A feedrate “F” has been defined with a wrong value.

Solution Program a positive feedrate “F” other than zero.

9103 ‘SLOT MILLING: S=0’

Detection During execution.

Cause A spindle speed “S” has been defined with a wrong value.

Solution Program a positive spindle speed “S” other than zero.

9104 ‘SLOT MILLING: P=0’

Detection During execution.

Cause The milling depth “P” has not been defined.

Solution The milling depth “P” must be other than zero.

9105 ‘SLOT MILLING: L=0’

Detection During execution.

Cause The slot length “L” has not been defined.

Solution The pocket depth “P” must be other than zero.

9106 ‘SLOT MILLING: Tool diameter smaller than Δ’

Detection During execution.

Cause The programmed milling step «Δ» is larger than the tool diameter.

Solution Program a milling step «Δ» smaller than the tool diameter or choose a tool of larger diameter.

9107 ‘SLOT MILLING: Tool diameter greater than groove'

Detection During execution.

Cause The diameter of the tool is larger than the programmed slot.

Solution Select a tool with a smaller diameter.

9108 ‘SLOT MILLING: FINISHING tool diameter smaller than δ’

Detection During execution.

Cause The programmed finishing stock «δ» is larger than the tool diameter.

Solution Program a finishing stock «δ» smaller than the tool diameter or choose a tool of larger diameter.

9109 'LINEAR POSITIONING: Wrong I'

Detection During execution.

Cause The «I» distance between positionings has the wrong value and it does not allow an integer number of machining operations.

Solution Check that the data entered is correct.

9110 'ARC POSITIONING: Wrong β'

Detection During execution.

Cause The distance «β» between positionings has the wrong value and it does not allow an integer number of machining operations.

Solution Check that the data entered is correct.

9111 'RECTANGULAR POSITIONING: Wrong Ix/Iy'

Detection During execution.

Cause One of the distances «Ix/Iy» between positionings has the wrong value and it does not allow an integer number of machining operations.

Solution Check that the data entered is correct.

9112 'GRID PATTERN POSITIONING: Wrong Ix/Iy'

Detection During execution.

Cause One of the distances «Ix/Iy» between positionings has the wrong value and it does not allow an integer number of machining operations.

Solution Check that the data entered is correct.