FAGOR AUTOMATION

CNC 8025 GP, M, MS

New Features (Ref. 0107 in)

ERRORS FOUND IN THE INSTALLATION MANUAL (REF. 9707)

Appendix "F" page 10. P621(7)

It is wrong, it should say:

P621(7) The M06 function executes the M19 function (0=Yes, 1=No)

Appendix "G" page 20. P621(7)

It is wrong, it should say:

P621(7) The M06 function executes the M19 function (0=Yes, 1=No)

MODIFICATIONS TO THE INSTALLATION MANUAL (REF. 9707)

Comparison table xii. Technical description. Inputs/Outputs.

Feedback inputs for rotary axes, it should say: W (GP), W (M), W (MG), W V (MS)

Comparison table xii. Technical description. Axis control.

	GP	M	MG	MS
Electronic threading. It should say		x	x	x

Comparison table xii. Technical description. Others. Add fields

	GP	M	MG	MS
Open loop motors without servodrives................................	x			
Laser machines...		x	x	x
JIG Grinders..		x	x	x

Section 3.3.3 (chapter 3 page 15). P612(6). Another example:

Having a Fagor electronic handwheel (25 lines/turn) set as follows:

P612(3) = 0 Millimeters P612(4) = 0 and P612(5) = 0 Resolution: 0.001 mm.

P612(6) = 0 Multiplication factor x4

Depending on the position of the MFO switch (Manual Feedrate Override) the selected axis will move:

Position	1	1 x 25 x 4 =	0.100 mm per turn
Position	10	10 x 25 x 4 =	1.000 mm per turn
Position	100	100 x 25 x 4 =	10.000 mm per turn

MODIFICATIONS TO THE PROGRAMMING MANUAL (REF. 9701)

Section 6.30.4 (page 128). G76 Automatic block generation

If the new program to be created is to be sent to a PC (G76 N), the DNC communication must be enabled and, at the PC, the "program management" "Digitizing Reception" option must be selected.

If it is not, the CNC will issue error 56.

1. EXPANSION OF THE INTEGRATED PLC RESOURCES

1.1 INPUTS

1.1.1 AXIS BEING HOMED (REFERENCED)

Input I88 indicates whether a home search is taking place and inputs I100, I101, I102, I103 and I104 indicates which axis is being homed.

I88	Indicates whether any axis is being homed (0=No / 1=Yes)
I100	Indicates whether the X axis is being homed (0=No / 1=Yes)
I101	Indicates whether the Y axis is being homed (0=No / 1=Yes)
I102	Indicates whether the Z axis is being homed (0=No / 1=Yes)
I103	Indicates whether the W axis is being homed (0=No / 1=Yes)
I104	Indicates whether the V axis is being homed (0=No / 1=Yes)

1.1.2 AXIS MOVING DIRECTION

Inputs I42, I43, I44, I45 and I46 will show, at all times, the moving direction of each axis.

I42	Indicates the moving direction of the X axis (0=Positive / 1=negative)
I43	Indicates the moving direction of the Y axis (0=Positive / 1=negative)
I44	Indicates the moving direction of the Z axis (0=Positive / 1=negative)
I45	Indicates the moving direction of the W axis (0=Positive / 1=negative)
I46	Indicates the moving direction of the V axis (0=Positive / 1=negative)

1.2 OUTPUTS

1.2.1 ENABLING THE CYCLE-START KEY VIA PLCI

With this feature it is possible to set the treatment of the [CYCLE START] of the CNC via PLCI. Machine parameter "P627(7)" indicates whether this feature is available or not.

P627(7) = 0 This feature **is not** available.
P627(7) = 1 This feature **is** available.

When using this feature, the way the CNC handles the [CYCLE START] key depends on the status of PLCI output O25 (CYCLE START ENABLE).

O25 = 0	The CNC **ignores** both the [CYCLE-START] key and the external [CYCLE-START] signal.
O25 = 1	The CNC takes into account both the [CYCLE-START] key and the external [CYCLE-START] signal.

1.2.2 TRAVEL LIMITS SET VIA PLCI

With this feature, the travel limits of the axes may be set via PLCI. Machine parameter "P627(7)" indicates whether this feature is available or not.

P627(7) = 0 This feature **is not** available.
P627(7) = 1 This feature **is** available.

To set the travel limits for each axis, use the following outputs:

O52 / O53	Positive / negative X axis limits
O54 / O55	Positive / negative Y axis limits
O56 / O57	Positive / negative Z axis limits
O58 / O59	Positive / negative W axis limits
O60 / O61	Positive / negative V axis limits

When the PLCI activates one of this outputs while the axis is moving in the same direction, the CNC stops the axes and the spindle and it displays an axis-travel-limit-overrun error.

1.2.3 DENYING ACCESS TO THE EDITOR MODE VIA PLCI

Machine parameter"P627(7)" indicates whether this feature is available or not.

P627(7) = 0 This feature **is not** available.
P627(7) = 1 This feature **is** available.

When using this feature, access to the editor mode at the CNC depends on the status of PLCI output O26, as well as on the current conditions (protected memory, number of the program to be locked).

O26 = 0 Free access to the editor mode (it is protected by current conditions).
O26 = 1 Denied access to the editor mode.

1.2.4 SPINDLE CONTROLLED VIA CNC OR VIA PLCI

From this version on, the spindle analog output may be set either by the CNC or by the PLCI. Machine parameter "P627(7)" indicates whether this feature is available or not.

P627(7) = 0 This feature **is not** available
P627(7) = 1 This feature **is** available

Setting the spindle analog output via PLCI

To do this, use the combination: M1956 - R156.

Register R156 sets the spindle analog output in units of 2.442 mV. (10 / 4095)

R156 = 0000 1111 1111 1111	(R1256=4095)	= 10V.
R156 = 0001 1111 1111 1111		= -10V.
R156 = 0000 0000 0000 0001	(R1256=1)	= 2.5 mV.
R156 = 0001 0000 0000 0001		= -2.5 mV.

In order for the CNC to assume the value allocated to register R156, one must activate mark M1956 as described in the PLCI Manual (section 5.5.2. Writing internal CNC variables).

Spindle controlled either by the CNC or by the PLCI

The CNC may have two internal spindle analog outputs, that of the CNC itself and the one set by the PLCI.

Use PLCI output O27 to "tell" the CNC which one of them to output.

O27 = 0 Spindle analog output set by the CNC itself.
O27 = 1 Spindle analog output set by the PLCI (combination: M1956-R156).

1.3 READING INTERNAL CNC VARIABLES

From this version on, the PLCI and the PLC64 have access to more internal CNC information.

With the PLCI, there is no need to activate a mark to access this information. The CNC itself updates this information at the beginning of each PLCI cycle scan.

With the PLC 64, the corresponding mark must be consulted every time a CNC variable is to be checked.

The CNC information now accessible is:

Real S in rpm (REG119 at the PLCI, M1919 at the PLC64)

Not to be mistaken with R112 which indicates the programmed Spindle speed (S).

It is given in rpm and in hexadecimal format. Example: S 2487 R119= 967

Number of the block in execution (REG120 at the PLCI, M1920 at the PLC64)

It is given in hexadecimal format. Example: N120 R120= 78

Code of the last key pressed (B0-7 REG121 at the PLCI, Not available at the PLC64)

Not to be mistaken with register R118 which also indicates the code corresponding to the last key pressed **but**--

When pressing a key, both registers have the same value; but **the data in R121 is only kept there for one cycle scan** whereas **R118 keeps its value until another key is pressed**

When pressing the same key several times, (for example: 1111):
R121 will show code "1" four times (once per cycle scan).
R118 will always show the same value, thus not being able to tell whether the "1" key has been pressed once or more times.

The key codes are listed in the appendix of the PLCI manual.

Operating mode selected at the CNC (B8-11 REG121 at the PLCI, Not available at the PLC64)

B8	B9	B10	B11	
0	0	0	0	Automatic
0	0	0	1	Single block
0	0	1	0	Play-Back
0	0	1	1	Teach-in
0	1	0	0	Dry-Run
0	1	0	1	JOG
0	1	1	0	Editor
0	1	1	1	Peripherals
1	0	0	0	Tool Table and G functions
1	0	0	1	Special modes

Status of the miscellaneous "M" functions (REG122 at the PLCI, Not available at the PLC64)

The status of each one of these functions is given by a bit and will appear as a "1" when active and "0" when inactive.

B15	B14	B13	B12	B11	B10	B9	B8	B7	B6	B5	B4	B3	B2	B1	B0
							M19	M1	M30	M6	M5	M4	M3	M2	M0

2. RETRACE FUNCTION.

This feature is available on the following models:

CNC-8025M CNC-8025MG CNC-8025MS
CNC-8025MI CNC-8025MGI CNC-8025MSI

Machine parameter "P627(6)" indicates whether this feature is available or not.

P627(6)=0 This feature **is not** available
P627(6)=1 This feature **is** available

This function may be selected by the operator. To do this, activate:

On models without PLCI: pin 7 of connector A5.
On models **with** PLCI: PLCI output O47

Operation:

As the CNC executes motion blocks, it always stores the last 10 blocks already executed

Whenever it executes a block containing an M,S,T type function, the machining conditions change and the CNC deletes those previously stored motion blocks.

When the retrace function is activated, the block currently in execution is interrupted and the retrace process begins.
First to the starting point of the current block and, then, to that of the previously stored program blocks.

If all the stored blocks are executed, the CNC stops the machine until the retrace function is canceled.

When this function is canceled, the CNC interrupts the current movement (if any) and it executes all the retraced blocks again. Once the interruption point is reached, the CNC resumes the execution of the program.

3. OPERATION WITH TWO MOTORS AND 3 AXES.

Machine parameter "P627(8)" indicates whether this feature is available or not.

P627(8)=0 This feature **is not** available.
P627(8)=1 This feature **is** available.

Operation:

The CNC permits using 2 motors to move 3 axes with the following conditions:

One of the axes shared by a motor must be the Z axis and the other one must be either the X or the Y axis.

Only interpolations between the X and Y axes are possible. The Z axis cannot be interpolated with any other axis. It must be moved alone.

Example: To move the tool from "X0 Y0 Z0" to "X20 Y20 Z20", The CNC will make this move in two steps. First, it will move the X and Y axes to X20 Y20 and, then, the Z axis to Z20.

4. SPINDLE FOLLOWING ERROR DISPLAY WHILE IN M19

From this version on, when operating with spindle orient (M19), the CNC also shows the spindle following error on the screen corresponding to the following error in Automatic and Single block modes.

The Following Error screen shows, in large characters, the amount of axis lag and, under it, the following information line.

F 00000.0000 & 100 S 0000 % 100 T 00.00 **S 0000.000**

The last value of this line **"S 0000.000"** shows the amount of following error (lag) of the spindle when it operates in spindle orient mode (M19).

5. GANTRY AXES NOT MECHANICALLY SLAVED

From this version on, depending on the setting of machine parameter "P629(8)", it is possible to work with two different types of Gantry axes.

"P629(8)=0" Mechanically slaved Gantry Axes. As until now.
When being homed, both axes behave as a single axis. The CNC takes into account only the parameter settings and feedback pulses of the main axis, the slaved one being just a follower of the main axis,

"P629(8)=1" Not-mechanically slaved Gantry axes.
When being homed, the two axes behave as separate independent axes. First the main axis is homed and, then, the slaved one.

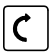
6. SHEETMETAL FORMING MACHINES

This feature is available on GP models.

To enable it, set machine parameter "P626(7)=1". The CNC enables functions M98 and M99 to control the X axis positioning loop.

Function M98 opens the X axis loop and M99 closes it.
When the CNC executes an M30, it also closes the X axis positioning loop.

When operating in jog mode, the CNC enables he following keys to control the X axis positioning loop:

 Executes an M98 opening the X axis loop.

 Executes an M98 opening the X axis loop.

STOP Executes an M98 closing the X axis loop.

Version 7.2 (April 1997)

1. SCREEN SAVER

The screen saver function works as follows:

After 5 minutes without pressing a key or without the CNC refreshing the screen, the screen goes blank. Press any key to restore the display.

Machine parameter "P626(5)" indicates whether this feature is to be used or not.

P626(5) = 0 This feature is not being used.
P626(5) = 1 This feature is being used.

2. JOGGING FEEDRATE

If while in JOG mode, the conditional input (block skip), pin 18 of connector I/O1, the CNC does not allow entering a new F value. Only the feedrate override (%) may be varied by means of the MFO switch.

3. PARAMETRIC PROGRAMMING NEW FUNCTION: F34

Function F34 returns the number of the tool being dealt with.

P27=F34 Parameter P27 takes the value of the new tool being dealt with.

This function must be used when working with a subroutine associated with the tool change.
When using it outside that subroutine, function F34 returns the value of "100".

Version 7.3 (March 1998)

1. PLCI. Input I87

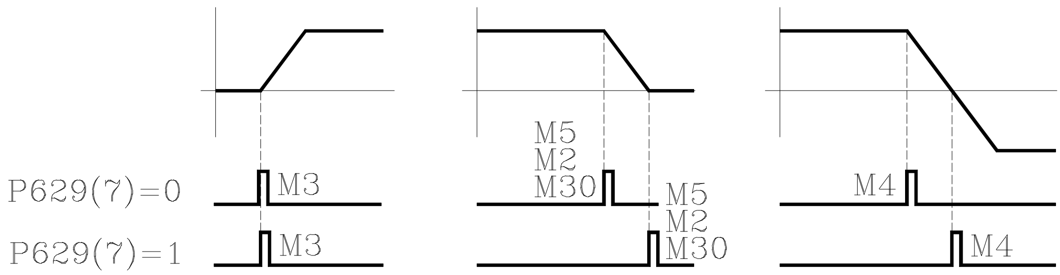
While the CNC is threading (G84), PLCI input I87 is set to "1".

Note: Input I97 indicates rigid tapping.

Version 7.4 (May 1999)

1. NEW MACHINE PARAMETER ASSOCIATED WITH THE M FUNCTIONS

Machine parameter "P629(7)" indicates when the M3, M4, M5 functions are sent out while accelerating or decelerating the spindle.



2. CANCEL TOOL OFFSET DURING A TOOL CHANGE

From this version on, it is possible to execute a "T.0" type block inside the subroutine associated with the tool to cancel the tool offset. This lets move to a particular position without the need for cumbersome calculations.

Only the tool offset may be canceled (T.0) or modified (T.xx). The tool cannot be changed (Txx.xx) inside the subroutine associated with the tool.

3. DIVIDING FACTOR FOR FEEDBACK SIGNALS

Parameters P631(8), P631(7), P631(6), P631(5) and P631(4) are used together with P604(8), P604(7), P604(6), P604(5) and P616(8) which indicate the multiplying factor to be applied to the feedback signals of the X, Y, Z, W, V axes respectively.

X axis	Y axis	Z axis	W axis	V axis
P604(8)	P604(7)	P604(6)	P604(5)	P616(8)
P631(8)	P631(7)	P631(6)	P631(5)	P631(4)

Indicate whether the feedback signals are divided (=1) or not (=0).

P631(8)=0, P631(7)=0, P631(6)=0, P631(5)=0 and P631(4)=0	They are not divided
P631(8)=1, P631(7)=1, P631(6)=1, P631(5)=1 and P631(4)=1	They are divided by two.

Example:
We wish to obtain a resolution of 0.01 mm with a squarewave encoders mounted on the X axis with 5mm pitch ballscrew

Nr of pulses = ballscrew pitch / (Multiplying factor x Resolution)

With P604(8)=0 & P631(8)=0	x4 multiplying factor	Nr of pulses = 125
With P604(8)=1 & P631(8)=0	x2 multiplying factor	Nr of pulses = 250
With P604(8)=0 & P631(8)=1	x2 multiplying factor	Nr of pulses = 250
With P604(8)=1 & P631(8)=1	x1 multiplying factor	Nr of pulses = 500

Version 7.6 (July 2001)

1. G75 AFFECTED BY FEEDRATE OVERRIDE

From this version on, there is a new machine parameter indicating whether G75 is affected by the feedrate override or not.

P631(1) = 0 Not affected. It is always at 100%, like in previous versions.
P631(1) = 1 It is affected by the Feedrate override.

2. FEEDBACK FACTOR.

From this version on, there is a new machine parameter to set the resolution of an axis having an encoder and a leadscrew.

P819 Feedback factor for the X axis P820 Feedback factor for the Y axis P821 Feedback factor for the Z axis
P822 Feedback factor for the W axis P823 Feedback factor for the V axis

Values between 0 and 65534. The “0” value indicates that this feature is not being used.

Use the following formula to calculate the “Feedback Factor” :

Feedback factor = (Gear Ratio x Leadscrew pitch / Number of Encoder pulses) x 8.192

Examples:						
	Gear Ratio	1	1	2	1	
	Leadscrew pitch	5000	6000	6000	8000	(microns)
	Encoder	2500	2500	2500	2500	(pulses/turn)
	Feedback factor	16384	19660.8	39321.6	26214.4	

The machine parameters only admit integer values and sometimes the “Feedback Factor” has decimals. In those cases, assign the integer part to the machine parameter and use the leadscrew compensation table to make up for the decimal part.

The values to be entered in the table are calculated with the following formula:

Leadscrew position = Leadscrew Error (microns) x Integer of feedback factor / decimals of the feedback factor

For example: Gear ratio = 1 Leadscrew pitch = 6000 Encoder = 2500
Feedback factor = 19660.8 Machine parameter = 19660
For a leadscrew error of 20 microns Leadscrew position = 20 x 19660 / 0.8 = 491520
Going on with the calculation, we come up with the following table.

Leadscrew position		Leadscrew error	
P0 =	-1966.000	P1 =	-0.080
P2 =	-1474.500	P3 =	-0.060
P4 =	-983.000	P5 =	-0.040
P6 =	-491.500	P7 =	-0.020
P8 =	0	P9 =	0
P10 =	491.500	P11 =	0.020
P12 =	983.000	P13 =	0.040
P14 =	1472.500	P15 =	0.060
P16 =	1966.000	P17 =	0.080

3. NEW MODEL

From this version on, the new model MLI is now available.
It offers the same features as the MGI model and it is sold together with the motors and ACS drives..

Headquarters (SPAIN): **Fagor Automation S. Coop.**
Bº San Andrés s/n, Apdo. 144
20500 Arrasate - Mondragón
Tel: +34-943-719200
Fax: +34- 943-791712
+34-943-771118 (Service Dept.)
www.fagorautomation.com
E-mail: info@fagorautomation.es

FAGOR 8025/8030 CNC

Models: M, MG, MS, GP

OPERATING MANUAL

Ref. 9701 (in)

ABOUT THE INFORMATION IN THIS MANUAL

This manual is addressed to the machine operator. It describes how to operate with this 8025 CNC.

It includes the necessary information for new users as well as advanced subjects for those who are already familiar with this CNC product.

It may not be necessary to read this whole manual. Consult the list of "New Features and Modifications" which will indicate to you the chapters and sections describing them.

Consult the Comparison Table in order to find the specific features offered by your particular CNC model.

There is also an appendix on error codes which indicates some of the probable reasons which could cause each one of them.

Notes: 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.

INDEX

Section | Page

Comparison table for Mill Model FAGOR 8025/8030 CNCs ix
New features and modifications xv

INTRODUCTION
Safety Conditions .... Intr. 3
Material Returning Terms .... Intr. 5
Fagor Documentation for the 8025/30 M CNC .... Intr. 6
Manual Contents .... Intr. 7

1. Overview 1

2. Front panel 8025/30 CNC 2
2.1. Monitor/keyboard for the 8030 CNC 2
2.2. Control panel for the 8030 CNC 4
2.3. Monitor/keyboard/control panel for the 8025 CNC 5
2.4. Selection of colors 7
2.5. Cancellation of monitor display 7
2.6. Function keys (soft keys) 7

3. OPERATING MODES 8
3.1. 0 mode: AUTOMATIC (Continuous cycle) / 1 mode: SINGLE BLOCK 10
3.1.1. Execution of a program 10
3.1.1.1. Selection of the Automatic (0) Single Block (1) operating modes 10
3.1.1.2. Selection of the program to be executed 10
3.1.1.3. Selection of the first block to be executed 11
3.1.1.4. Display of the contents of the blocks 11
3.1.1.5. Cycle Start 12
3.1.1.6. Cycle Stop 12
3.1.1.7. Changing the operating mode 13
3.1.2. Display modes 13
3.1.2.1. Selection of the display mode 13
3.1.2.2. Standard display mode 14
3.1.2.3. Current position display mode 15
3.1.2.4. Following error display mode 15
3.1.2.5. Arithmetic parameters display mode 15
3.1.2.6. Subroutine status, clock and parts counter display mode 16
3.1.2.7. Graphics display mode 17
3.1.3. Programming while running a program. Background 18
3.1.4. PLC/LAN mode 18
3.1.5. Verification and modification of the values of the tool offset table
without stopping the cycle 19
3.1.6. Tool inspection 19
3.1.7. CNC reset 21
3.1.8. Display and deletion of the Messages sent by the FAGOR PLC 64. 21
3.2. Mode 2: PLAY-BACK 22
3.2.1 Selection of the operating mode PLAY-BACK 22

Section		Page

3.2.2.	Locking/Unlocking of memory	22
3.2.3.	Deletion of a complete program	22
3.2.4.	Change of program number	22
3.2.5.	Display and search of memorized subroutines	22
3.2.6.	Selection of a program	22
3.2.7.	Creating a program	23
3.2.8.	Deletion of a block	23
3.2.9.	Copy a program	23
3.3.	MODE 3: TEACH-IN	24
3.3.1.	Selection of the operating mode TEACH-IN	24
3.3.2.	Locking/Unlocking of memory	24
3.3.3.	Deletion of a complete program	24
3.3.4.	Change of program number	24
3.3.5.	Display and search of memorized subroutines	24
3.3.6.	Selection of a program	24
3.3.7.	Program creation	25
3.3.8.	Deletion of a block	25
3.3.9.	Copy a program	25
3.4.	Mode 4: DRY RUN	26
3.4.1.	Execution of a program	26
3.4.1.1.	Selection of the operating mode DRY RUN (4)	26
3.4.1.1.1.	Selection of execution mode	28
3.4.1.2.	Selection of the program to be executed	29
3.4.1.3.	Selection of starting block	29
3.4.1.4.	Display of the contents of the blocks	29
3.4.1.5.	Cycle Start	29
3.4.1.6.	Cycle Stop	29
3.4.1.7.	Change of operating mode	29
3.4.1.8.	Tool inspection	30
3.4.2.	Display modes	30
3.5.	Mode 5: JOG	31
3.5.1.	Selection of the JOG operating mode	31
3.5.2.	Search for machine reference axis by axis	32
3.5.3.	Presetting a coordinate value	32
3.5.4.	Jogging the axes	33
3.5.4.1.	Continuous movement	33
3.5.4.2.	Incremental movement	34
3.5.5.	Entering F,S and M	34
3.5.5.1.	Entering an F value	34
3.5.5.2.	Entering an S value	35
3.5.5.3.	Entering an M value	35
3.5.6.	Operation of the CNC as a readout	35
3.5.7.	Change of measurement units	36
3.5.8.	Handwheel operation	36
3.5.9.	Display/Modification of RANDOM table	37
3.5.10.	Measuring and loading of tool offsets with a probe	40
3.5.11.	Spindle operating keys	41
3.6.	Mode 6: EDITING	42
3.6.1.	Selection of the operating mode EDITING(6)	42
3.6.2.	Locking/Unlocking of memory and formatting of 512 Kb memory	42
3.6.3.	Part-program directory	43
3.6.3.1.	Deletion of a complete program	43
3.6.4.	Change of program number	44
3.6.5.	Display and search of subroutines stored in memory	44
3.6.6.	Selection of a program	45
3.6.7.	Creating a program	45
3.6.7.1	Displaying the block contents	45
3.6.7.2.	Unassisted programming	46
3.6.7.3.	Modification and deletion of a block	47
3.6.7.4.	Assisted programming	48

3.6.7.5. Save a program being edited (only on models with 512 Kb of memory) 49
3.6.7.6. Copying a program 49
3.7. Mode 7: PERIPHERALS 50
3.7.1. Selection of the operating mode PERIPHERALS (7) 50
3.7.2. Entering a program from the FAGOR cassette/recorder (0) 51
3.7.2.1. Transmission errors 53
3.7.3. Transferring a program to the FAGOR cassette recorder (1) 53
3.7.3.1. Transmission errors 54
3.7.4. Entering a program from a peripheral other than the FAGOR cassette/recorder (2) 55
3.7.5. Transferring a program to a peripheral other than the FAGOR cassette/recorder (3) 55
3.7.6. FAGOR cassette's directory (4) 56
3.7.7. Deletion of a FAGOR cassette program (5) 56
3.7.8. Interruption of the transmission process 57
3.7.9. DNC. Communication with a computer 57
3.8. Mode 8: TOOL OFFSETS AND ZERO OFFSETS G53/G59 58
3.8.1. Selection of the operating mode TOOL OFFSET (8) 58
3.8.2. Read-out of tool table 58
3.8.3. Entering the dimensions of the tools 59
3.8.4. Modification of tool dimensions 59
3.8.5. Change of measurement units 60
3.8.6. Zero offsets 61
3.8.6.1. Displaying the zero offset table 61
3.8.6.2. Entering zero offset values 61
3.8.6.3. Modification of zero offset values 62
3.8.6.4. Change of measuring units 62
3.8.7. Return to the tool offset table 62
3.8.8. Complete deletion of tool offsets or zero table 62
3.9. Mode 9: SPECIAL MODES 62
3.10. Graphics 63
3.10.1. Display area definition 64
3.10.2. Zooming (windowing) 65
3.10.3. Redefinition of the display area by the Zoom function 66
3.10.4. Deletion of graphics 66
3.10.5. Graphic representation in color 66

ERROR CODES

COMPARISON TABLE
FOR MILL MODEL
FAGOR 8025/8030 CNCs

<u>8025/8030 MILL MODEL CNCS</u>

Fagor offers the 8025 and 8030 mill type CNCs.

Both types operate the same way and offer similar characteristics. Their basic difference is that the former is compact and the latter is modular.

Both CNC types offer basic models. Although the differences between the basic models are detailed later on, each model may be defined as follows:

8025/8030 GP	Oriented to General Purpose machines
8025/8030 M	Oriented to Milling machines of up to 4 axes.
8025/8030 MG	Same as the M model, but with dynamic graphics.
8025/8030 MS	Oriented to Machining Centers (up to 5 axes).

When the CNC has an Integrated Programmable Logic Controller (PLCI), the letter "I" is added to the CNC model denomination: GPI, MI, MGI, MSI.

Also, When the CNC has 512Kb of part-program memory, the letter "K" is added to the CNC model denomination: GPK, MK, MGK, MSK, GPIK, MIK, MGIK, MSIK.

	Basic	**With PLCI**	**Basic With 512Kb**	**With PLCI and 512Kb**
General Purpose	**GP**	**GPI**	**GPK**	**GPKI**
Mills up to 4 axes	**M**	**MI**	**MK**	**MIK**
Up to 4 axes with graphics	**MG**	**MGI**	**MGK**	**MGIK**
Machining Centers	**MS**	**MSI**	**MSK**	**MSIK**

TECHNICAL DESCRIPTION

	GP	M	MG	MS
INPUTS/OUTPUTS				
Feedback inputs.	**6**	**6**	**6**	**6**
Linear axes	**4**	**4**	**4**	**5**
Rotary axes	**2**	**2**	**2**	**2**
Spindle encoder	**1**	**1**	**1**	**1**
Electronic handwheels	**1**	**1**	**1**	**1**
Probe input	**x**	**x**	**x**	**x**
Square-wave feedback signal multiplying factor, x2/x4	**x**	**x**	**x**	**x**
Sine-wave feedback signal multiplying factor, x2/x4/10/x20	**x**	**x**	**x**	**x**
Maximum counting resolution 0.001mm/0.001°/0.0001inch	**x**	**x**	**x**	**x**
Analog outputs (±10V) for axis servo drives	**4**	**4**	**4**	**5**
Spindle analog output (±10V)	**1**	**1**	**1**	**1**
AXIS CONTROL				
Axes involved in linear interpolations	**3**	**3**	**3**	**3**
Axes involved in circular interpolations	**2**	**2**	**2**	**2**
Helical interpolation	**x**	**x**	**x**	**x**
Electronic threading	**x**	**x**	**x**	
Spindle control	**x**	**x**	**x**	**x**
Software travel limits	**x**	**x**	**x**	**x**
Spindle orientation	**x**	**x**	**x**	**x**
Management of non-servo-controlled Open-Loop motor	x			
PROGRAMMING				
Part Zero preset by user	**x**	**x**	**x**	**x**
Absolute/incremental programming	**x**	**x**	**x**	**x**
Programming in cartesian coordinates	**x**	**x**	**x**	**x**
Programming in polar coordinates	**x**	**x**	**x**	**x**
Programming in cylindrical coordinates (radius, angle, axis)	**x**	**x**	**x**	**x**
Programming by angle and cartesian coordinate	**x**	**x**	**x**	**x**
COMPENSATION				
Tool radius compensation		**x**	**x**	**x**
Tool length compensation	**x**	**x**	**x**	**x**
Leadscrew backlash compensation	**x**	**x**	**x**	**x**
Leadscrew error compensation	**x**	**x**	**x**	**x**
Cross compensation (beam sag)	**x**	**x**	**x**	**x**
DISPLAY				
CNC text in Spanish, English, French, German and Italian	**x**	**x**	**x**	**x**
Display of execution time	**x**	**x**	**x**	**x**
Piece counter	**x**	**x**	**x**	**x**
Graphic movement display and part simulation			**x**	**x**
Tool base position display	**x**	**x**	**x**	**x**
Tool tip position display	**x**	**x**	**x**	**x**
Geometric programming aide	**x**	**x**	**x**	**x**
COMMUNICATION WITH OTHER DEVICES				
Communication vía RS232C	**x**	**x**	**x**	**x**
Communication via DNC	**x**	**x**	**x**	**x**
Communication via RS485 (FAGOR LAN)	**x**	**x**	**x**	**x**
ISO program loading from peripherals	**x**	**x**	**x**	**x**
OTHERS				
Parametric programming	**x**	**x**	**x**	**x**
Model digitizing	**x**	**x**	**x**	**x**
Possibility of an integrated PLC	**x**	**x**	**x**	**x**
Sheetmetal tracing on LASER machines				**x**
Jig Grinder				**x**

PREPARATORY FUNCTIONS

	GP	M	MG	MS
AXES AND COORDINATE SYSTEMS				
XY (G17) plane selection	x	x	x	x
XZ and YZ plane selection (G18,G19)	x	x	x	x
Part measuring units. Millimeters or inches (G70,G71)	x	x	x	x
Absolute/incremental programming (G90,G91)	x	x	x	x
Independent axis (G65)	x	x	x	x
REFERENCE SYSTEMS				
Machine reference (home) search (G74)	x	x	x	x
Coordinate preset (G92)	x	x	x	x
Zero offsets (G53...G59)	x	x	x	x
Polar origin preset (G93)	x	x	x	x
Store current part zero (G31)	x	x	x	x
Recover stored part zero (G32)	x	x	x	x
PREPARATORY FUNCTIONS				
Feedrate F	x	x	x	x
Feedrate in mm/min. or inches/minute (G94)	x	x	x	x
Feedrate in mm/revolution or inches/revolution (G95)	x	x	x	x
Constant surface speed (G96)	x	x	x	x
Constant tool center speed (G97)	x	x	x	x
Programmable feedrate override (G49)	x	x	x	x
Spindle speed (S)	x	x	x	x
S value limit (G92)	x	x	x	x
Tool and tool offset selection (T)	x	x	x	x
AUXILIARY FUNCTIONS				
Program stop (M00)	x	x	x	x
Conditional program stop (M01)	x	x	x	x
End of program (M02)	x	x	x	x
End of program with return to first block (M30)	x	x	x	x
Clockwise spindle start (M03)	x	x	x	x
Counter-clockwise spindle start (M04)	x	x	x	x
Spindle stop (M05)	x	x	x	x
Tool change in machining centers (M06)	x	x	x	x
Spindle orientation (M19)	x	x	x	x
Spindle speed range change (M41, M42, M43, M44)	x	x	x	x
Functions associated with pallets (M22, M23, M24, M25)	x	x	x	
PATH CONTROL				
Rapid traverse (G00) x	x	x	x	
Linear interpolation (G01) x	x	x	x	
Circular interpolation (G02,G03) x	x	x	x	
Circular interpolation with absolute center coordinates (G06) x	x	x	x	
Circular path tangent to previous path (G08) x	x	x	x	
Arc defined by three points (G09) x	x	x	x	
Tangential entry at beginning of a machining operation (G37) x	x	x	x	
Tangential exit at the end of a machining operation (G38) x	x	x	x	
Controlled radius blend (G36) x	x	x	x	
Chamfer (G39) x	x	x	x	
Electronic threading (G33)	x	x	x	
ADDITIONAL PREPARATORY FUNCTIONS				
Dwell (G04 K) x	x	x	x	
Round and square corner (G05, G07) x	x	x	x	
Mirror image (G10,G11,G12) x	x	x	x	
Mirror image along the Z axis (G13) x	x	x	x	
Scaling factor (G72) x	x	x	x	
Pattern rotation (G73) x	x	x	x	
Slaving/unslaving of axes (G77, G78) x	x	x	x	
Single block treatment (G47, G48) x	x	x	x	
User error display (G30) x	x	x	x	
Automatic block generation (G76)			x	
Communication with FAGOR Local Area Network (G52) x	x	x	x	

	GP	M	MG	MS
COMPENSATION				
Tool radius compensation (G40,G41,G42)		x	x	x
Tool length compensation (G43,G44)	x	x	x	x
Loading of tool dimensions into internal tool table (G50)	x	x	x	x
CANNED CYCLES				
Multiple arc-pattern machining (G64)		x	x	x
User defined canned cycle (G79)	x	x	x	x
Drilling cycle (G81)		x	x	x
Drilling cycle with dwell (G82)		x	x	x
Deep hole drilling cycle (G83)		x	x	x
Tapping cycle (G84)		x	x	x
Rigid tapping cycle (G84R)		x	x	x
Reaming cycle (G85)		x	x	x
Boring cycle with withdrawal in G00 (G86)		x	x	x
Rectangular pocket milling cycle (G87)		x	x	x
Circular pocket milling cycle (G88)		x	x	x
Boring cycle with withdrawal in G01 (G89)		x	x	x
Canned cycle cancellation (G80)	x	x	x	x
Return to starting point (G98)		x	x	x
Return to reference plane (G99)		x	x	x
PROBING				
Probing (G75)	x	x	x	x
Tool length calibration canned cycle (G75N0)				x
Probe calibration canned cycle (G75N1)				x
Surface measuring canned cycle (G75N2)				x
Surface measuring canned cycle with tool offset (G75N3)				x
Outside edge measuring canned cycle (G75N4)				x
Inside edge measuring canned cycle (G75N5)				x
Angle measuring canned cycle (G75N6)				x
Outside edge and angle measuring canned cycle (G75N7)				x
Hole centering canned cycle (G75N8)				x
Boss centering canned cycle (G75N9)				x
Hole measuring canned cycle (G75N10)				x
Boss measuring canned cycle (G75N11)				x
SUBROUTINES				
Number of standard subroutines	99	99	99	99
Definition of standard subroutine (G22)	x	x	x	x
Call to a standard subroutine (G20)	x	x	x	x
Number of parametric subroutines	99	99	99	99
Definition of parametric subroutine (G23)	x	x	x	x
Call to a parametric subroutine (G21)	x	x	x	x
End of standard or parametric subroutine (G24)	x	x	x	x
JUMP OR CALL FUNCTIONS				
Unconditional jump/call (G25)	x	x	x	x
Jump or call if zero (G26)	x	x	x	x
Jump or call if not zero (G27)	x	x	x	x
Jump or call if smaller (G28)	x	x	x	x
Jump or call if equal or greater (G29)	x	x	x	x

NEW FEATURES AND MODIFICATIONS

Date: **February 1991** **Software version:** **2.1 and newer**

FEATURE	MODIFIED MANUAL AND SECTION	
Error 65 is not issued while probing (G75)	Installation Manual	Section 3.3.4
It is possible to select the home searching direction for each axis	Installation Manual	Section 4.6
New 1, 2, 5, 10 resolution values for sine-wave feedback signals of each axis	Installation Manual	Section 4.1
PLCI register access from the CNC	Programming Manual	G52
Sheetmetal tracing on laser machines	Applications Manual	
Jig Grinder	Applications Manual	

Date: **June 1991** **Software version:** **3.1 and newer**

FEATURE	MODIFIED MANUAL AND SECTION	
Repetitive emergency subroutine	Installation Manual	Section 3.3.8
New function F29. It takes the value of the selected tool	Programming Manual	Chapter 13
Function M06 does not execute M19	Installation Manual	Section 3.3.5
Greater speed when executing several parametric blocks in a row.		

Date: March 1992 **Software version: 4.1 and newer**

FEATURE	**MODIFIED MANUAL AND SECTION**	
Bell-shape acceleration/deceleration control	Installation Manual	Section 4.7
Expansion of cross compensation	Installation Manual	Section 4.10
Rigid Tapping G84 R	Programming Manual	G84
Possibility to enter the sign of the leadscrew backlash for each axis	Installation Manual	Section 4.9
Independent execution of an axis	Programming Manual	G65

Date: July 1993 **Software version: 5.1 and newer**

FEATURE	**MODIFIED MANUAL AND SECTION**	
Double cross compensation	Installation Manual	Section 4.10
Linear and bell-shaped acc./dec. ramp combination for the axes	Installation Manual	Section 4.7
Acceleration/deceleration control for the the spindle	Installation Manual	Section 5.
Multiple arc pattern machining	Programming Manual	G64
Tool tip position display	Installation Manual	Section 3.3.5
The associated subroutine is executed before the T function	Installation Manual	Section 3.3.5
The additional circular sections of a compensated path are executed in G05 or G07	Installation Manual	Section 3.3.8
VGA monitor 8030 CNC.	Installation Manual	Chapter 1

Date: **March 1995** **Software version:** **5.3 and newer**

FEATURE	**MODIFIED MANUAL AND SECTION**	
Management of feedback with coded Io	Installation Manual	Section 4.6 & 6.5
Spindle inhibit by PLC	Installation Manual	Section 3.3.9
Handwheel management by PLC	Installation Manual	Section 3.3.3
Rapid (JOG) key simulation via PLC	PLCI Manual	
Non-servo-controlled open-loop motors	Applications Manual	
Function G64, multiple machining in an arc. To be selected by machine parameter.	Installation Manual	Section 3.3.9
Initialization of machine parameters after memory loss.		

Date: **September 1995** **Software version:** **6.0 and newer**

FEATURE	**MODIFIED MANUAL AND SECTION**	
512 Kb of part-program memory	Operating Manual	Section 3.6
When conditional input (block skip) active while in JOG mode, the key is ignored	Installation Manual	Section 1.3.6

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 not 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)

Protections of the unit itself

Central Unit

It carries two fast fuses of 3.15 Amp./ 250V. to protect the mains AC input.

All the digital inputs and outputs are protected by an external fast fuse (F) of 3.15 Amp./ 250V. against over voltage and reverse connection of the power supply.

Monitor

The type of fuse depends on the type of monitor. See the identification label of the unit.

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.

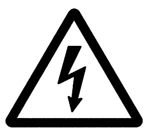
Safety symbols

Symbols which may appear on the manual

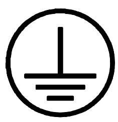
WARNING. symbol
It has an associated text indicating those actions or operations may hurt people or damage products.

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 CNC, 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 8025/30 M CNC

8025M CNC OEM Manual	Is directed to the machine builder or person in charge of installing and starting up the CNC.
	It contains 2 manuals: Installation Manual — describing how to isntall and set-up the CNC. LAN Manual — describing how to instal the CNC in the Local Area Network.
	Sometimes, it may contain an additional manual describing New Software Features recently implemented.
8025M CNC USER Manual	Is directed to the end user or CNC operator.
	It contains 3 manuals: Operating Manual — describing how to operate the CNC. Programming Manual — describing how to program the CNC. Applications Manual — describing other applications for this CNC non-specific of Milling machines
	Sometimes, it may contain an additional manual describing New Software Features recently implemented.
DNC 25/30 Software Manual	Is directed to people using the optional DNC communications software.
DNC 25/30 Protocol Manual	Is directed to people wishing to design their own DNC communications software to communicate with the 800 without using the DNC25/30 software..
PLCI Manual	To be used when the CNC has an integrated PLC.
	Is directed to the machine builder or person in charge of installing and starting up the PLCI.
DNC-PLC Manual	Is directed to people using the optional communications software: DNC-PLC.
FLOPPY DISK Manual	Is directed to people using the Fagor Floppy Disk Unit and it shows how to use it.

MANUAL CONTENTS

The operating manual consists of the following chapters:

Index

Comparison table of FAGOR models: 8025 M CNCs

New Features and modifications.

Introduction
- Safety conditions.
- Material returning conditions.
- FAGOR documentation for the 8025 M CNC.
- Manual contents.

Overview

Front panel of the 8025 M CNC

Operating modes

0- Automatic
1- Single block
2- Play-back
3- Teach-in
4- Dry-run
5- Jog
6- Editor
7- Peripheral
8- Tool table and zero offset table
9- Special modes

Error codes

1. OVERVIEW

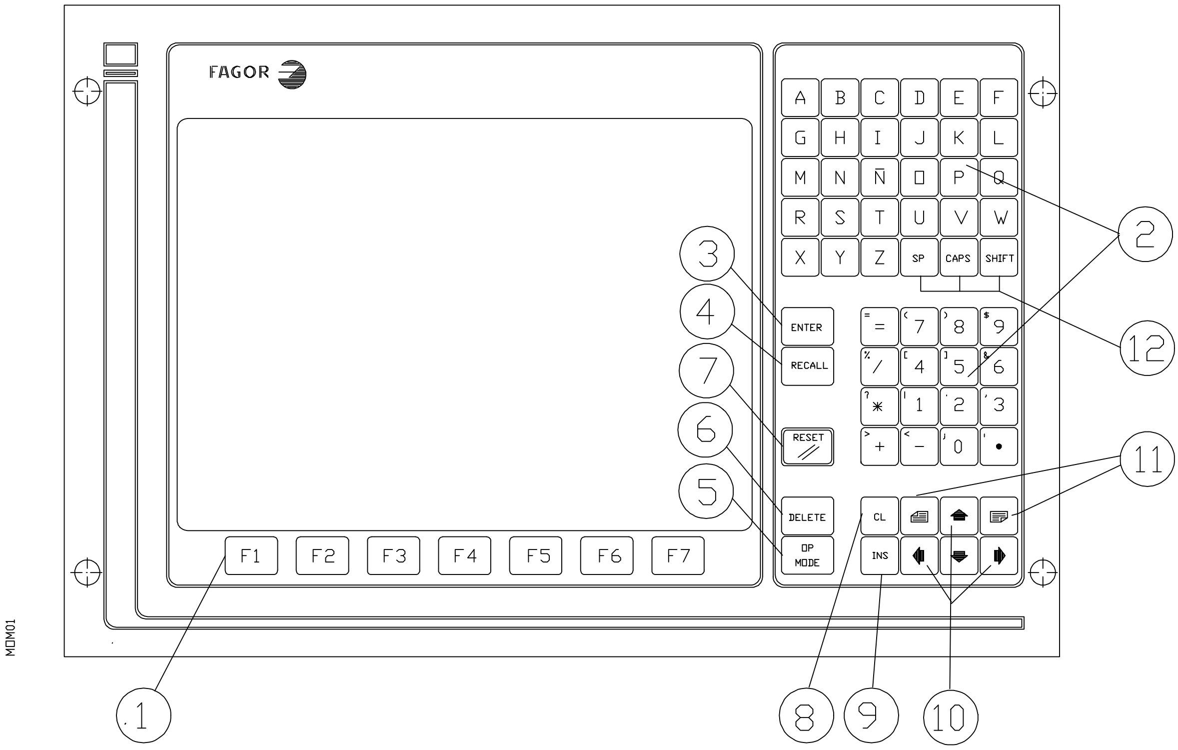
This manual contains the information required for the proper operation of the CNC.

It describes the controls fitted on both the keyboard and the front panel.

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

2. FRONT PANEL 8025/30 CNC

2.1. MONITOR/KEYBOARD FOR THE 8030 CNC



1. Function keys (SOFT-KEYS)

2. Alphanumeric keyboard for editing programs.

3. **ENTER**. Allows information to be entered in the CNC memory, etc.

4. **RECALL**. To access a program, a block within a program,etc.

5. **OP MODE**. Allows a list of operating modes to be displayed on the screen. It is a previous step to accessing any of them.

6. **DELETE**. It allows deletion of a complete program or a block of the programme. Deletion of the graphic representation, etc.

7. **RESET**. To revert the CNC to the initial conditions and recognise new machine parameter values, decoded M functions, etc.

8. **CL**. To delete characters one by one during the editing process, etc.

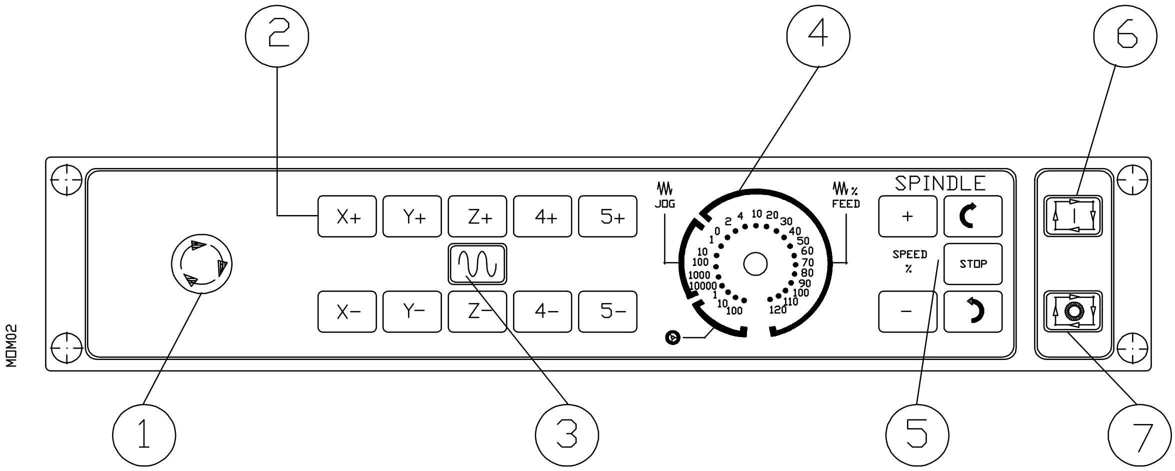
9. **INS**. Key which allows characters to be inserted during the edition of a program block.

10. Arrow keys for moving cursor.

11. Page **up** and page **down** keys.

12. **SP**. Reserves a space between characters of a comment.
CAPS. Allows characters to be edited in capitals.
SHIFT. Allows characters to be edited which are found on keys with double meaning.

2.2. CONTROL PANEL FOR THE 8030 CNC



1. Emergency Button or Electronic Handwheel (optional)

2. **JOG** keys for manual displacement of the axes.

3. **RAPID FEED** button.

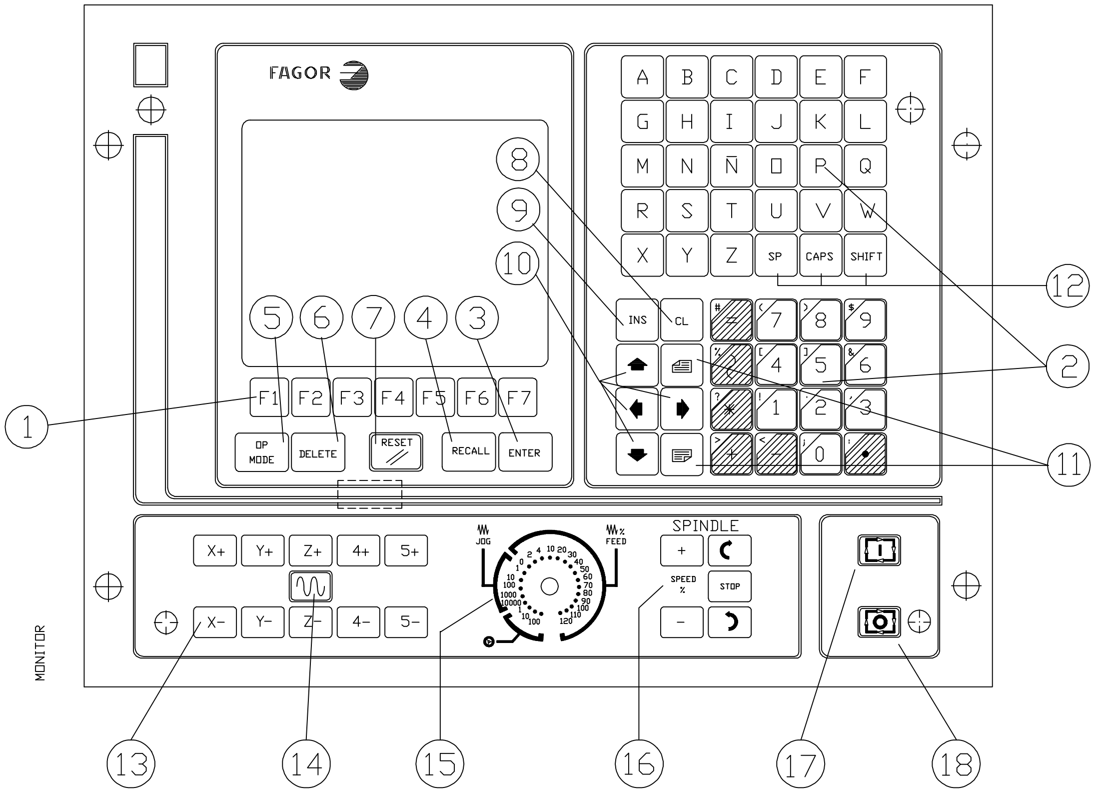
4. Switch (M.F.O.), which allows a % variation of the programmed feedrate and to choose the different ways of working in the **JOG MODE** (continuous, incremental, electronic handwheel).

5. Spindle operating keys. Allow the spindle to be put into OPERATION and to **STOP** it, in the JOG operating mode. The + and – keys allow a % variation of the programmed turning speed of the spindle during operation.

6. **START**. Cycle START key.

7. **STOP**. Cycle STOP key.

2.3. MONITOR/KEYBOARD/CONTROL PANEL FOR THE 8025 CNC



1. Function keys (SOFT-KEYS)

2. Alphanumeric keyboard for editing programs.

3. **ENTER**. Allows information to be entered in the CNC memory, etc.

4. RECALL. To access a program, a block within a program,etc.

5. OP MODE. Allows a list of operating modes to be displayed on the screen. It is a previous step to accessing any of them.

6. DELETE. It allows deletion of a complete program or a block of the programme. Deletion of the graphic representation, etc.

7. RESET. To revert the CNC to the initial conditions and recognise new machine parameter values, decoded **M** functions, etc.

8. **CL**. To delete characters one by one during the editing process, etc.

9. **INS**. Key which allows characters to be inserted during the edition of a program block.

10. Arrow keys for moving the cursor.

11. Page **up** and page **down** keys.

12. **SP**. Reserves a space between characters of a comment.
CAPS. Allows characters to be edited in capitals.
SHIFT. Allows characters to be edited which are found on keys with double meaning.

13. **JOG** keys for manual displacement of the axes.

14. **RAPID FEED** button.

15. Switch (M.F.O.), which allows a % variation of the programmed feed and to choose the different ways of working in the **JOG MODE** (continuous, incremental, electronic handwheel).

16. Spindle operating keys. Allow the spindle to be put into **OPERATION** and to **STOP** it, in the **JOG** operating mode. The and [+] keys allow a % variation of the programmed turning speed of the spindle during operation.

17. **START**. Cycle START key.

18. **STOP.** Cycle STOP key.

2.4. SELECTION OF COLORS

Whenever the CNC is fitted with a COLOR MONITOR, it is possible to choose the set of colors one wishes to appear on the screen.

Colors are selected by means of the designation of values to the Machine Parameter P619 bits (2) and (1).

P619 (2)	P619(1)	Monitor
0	0	Monitor
0	1	Combination 1
1	0	Combination 2

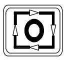
Each of the combinations, 1 and 2, are a group of 3 different colors to distinguish the characters displayed.

2.5. Cancellation of the MONITOR DISPLAY

In any of the Modes of Operation of the CNC, it is possible to blank the MONITOR out.

First of all, it is necessary to press the key SHIFT and then the key CL

To restore the display just press any key.

In this case, the STOP key  , in addition to recovering the last display, stops the possible running of the CNC.

The display is also recovered when a message is received from the PLC64 or from the PLCI.

2.6. FUNCTION KEYS (SOFT KEYS)

The CNC has 7 function keys (F1/F7), placed under the screen, which allow the user to operate with the CNC comfortably and quickly.

Their meaning will be displayed on the screen just above the corresponding function keys and will be different in each of the situations and modes of operation.

Throughout the manual the meaning of the F1/F7 keys which must be pressed in each case, will be indicated in square brackets [].

3. OPERATING MODES

The CNC has 10 different operating modes:

0. AUTOMATIC : Execution of programs in a continuous cycle.

1. SINGLE BLOCK : Execution of part programs block by block.

2. PLAY-BACK : Creation of a program in memory while the machine is being operated manually.

3. TEACH-IN :

- Creation and execution of a block without entering it into memory.
- Creation, execution and entering of a block into memory; thus a program is created while being executed block by block.

4. DRY RUN : To check programs before actual execution of the first part.

5. JOG/HOME SEARCH :

- Manual movement of the machine.
- Machine-reference. (Home search).
- Presetting of any value and zero-setting the axes.
- Entering and executing of F,S,M.
- Setting initial conditions of the tool magazine.
- Handwheel operation.

6. EDITING

Creation, modification and checking of blocks, programs and subroutines.

7. INPUT-OUTPUT

Transferring programs or machine-parameters from/to peripherals.

8. TOOL OFFSETS/ G53-G59

Input, modification and checking of the dimensions (radius and length) of up to 100 tools and of zero offsets (G53-G59).

9. SPECIAL MODES

- General testing of the CNC.
- Verification of inputs and outputs.
- Setting of decoded M functions.
- Setting of machine-parameters.
- Input of values for leadscrew error compensation.
- Operate with the PLC.

By means of these operating modes it is possible to program the CNC, produce parts in a continuous run, work block by block and work manually.

Sequence for obtaining these operating modes:

- Press **OP MODE**: The list of 10 modes will appear on the screen.
- Press the number of the desired operating mode.

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

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

3.1.1. Execution of a program

The execution of a program requires the following steps:

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

- Press **OP MODE** : The list of 10 operating modes appears on the screen.
- Press **0/1** key : The standard display corresponding to this operating mode appears; i.e. in the upper left-hand section of the screen the message **AUTOMAT/SINGLE BLOCK** followed by the number of the program P —— and the number of the first block to be executed N ——.

3.1.1.2. Selection of the program to be executed

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

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

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

N*

3.1.1.3. Selection of the first block to be executed

Once a program has been selected, the number of the first block to be executed appears to the right of the program number.

If you wish to begin with a different block, the following procedure should be followed:

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

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

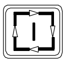
3.1.1.4. Display of the contents of the blocks

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

- Press ↑ : The previous blocks are displayed
- Press ↓ : The next blocks are displayed

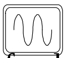
> **Atention:**
>
> The program always starts with the block whose number appears to the right of the program number, regardless of which ones are displayed on the screen.

3.1.1.5. Cycle Start

- Press 

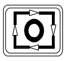
. Once the program and block number have been selected, just press this key to execute the program in **AUTOMATIC** or the block in **SINGLE BLOCK**.

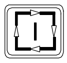
. If the program contains any **conditional block** it will be executed when the relevant input is activated (see INSTALLATION AND START-UP MANUAL). If it is not activated, the CNC will disregard such block.

. During the time that the fast travel button  is pressed carrying out a movement in G01, G02, or G03, the percentage of the feedrate will be 200% of the programmed feedrate, whenever the machine parameter P606(2) has a value equal to zero. This will happen when the external input **START** is activated, if parameter P609(7) = 1.

. In the **SINGLE BLOCK** mode all those blocks which are programmed with parameters will be executed by the FAGOR CNC as if they were a single **BLOCK**, whenever these are in canned cycles.

3.1.1.6. Cycle stop

- Press 

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

The cycle is also stopped by means of:

- Codes M00,M02,M30,M06 [M06 depending on parameter P601(8)].

- Code M01 when the relevant input is activated.

- The external signal $\overline{\textbf{FEED HOLD}}$ (the cycle continues when the signal disappears)

- The external signal $\overline{\textbf{EMERGENCY STOP}}$ (in this case the program must be restarted, since the CNC is reset to initial state).

- The external $\overline{\textbf{STOP}}$ signal

If machine parameter P727 has a value between 1 and 99, when the external **STOP** input is activated while running a program, the CNC will interrupt the program and jump to execute the standard subroutine corresponding to the number assigned to P727.

3.1.1.7. Changing the operating mode

It is possible, at any time during the execution of a cycle in **AUTOMATIC** mode, to switch to **SINGLE BLOCK** mode or vice versa. To do so:

- Press **OP MODE**. The listing of operating modes will appear on the screen.

- Press **1/0** (depending on the execution mode).

If any number other than **1/0** is pressed, the CNC returns to the previous position.

3.1.2. Display Modes

The display modes in **AUTOMATIC** or in **SINGLE BLOCK** are:

. STANDARD
. CURRENT POSITION
. FOLLOWING ERROR
. ARITHMETICAL PARAMETERS
. SUBROUTINE STATUS
. GRAPHICS
. EDITOR (BACKGROUND)
. PLC/LAN
. TOOL COMPENSATION
. TOOL INSPECTION
. PLC MESSAGES

3.1.2.1. Selection of display mode.

By pressing the function keys (**F1/F7**), placed under the screen, the user can select the desired mode which appears displayed just above the corresponding function key.

By means of the [**ETC**] key, other function keys which are not displayed can be accessed.

3.1.2.2. STANDARD display mode.

This mode is automatically imposed on selecting the **AUTOMATIC** or **SINGLE BLOCK** mode of operation.

Information displayed on screen.

. Upper part. The message **AUTOMATIC** or **SINGLE BLOCK** and then the number of the program, of the first block to run or the one which is being run. Underneath, the contents of the first block of the programme or of the block being run and the following (2 or 3).

. Central part. Under the titles **COMMAND**, **ACTUAL** and **TO GO** appear the axis arrival dimensions, the current position and those still to travel, respectively.

. Lower part. The programmed values of **F** and **S** appear and their **%**, as well as the list of activated **G**, **T** and **M** functions.

This part of the screen also displays messages sent to the CNC from the PLC, programmed comments, as well as the meaning of the function keys.

3.1.2.3. ACTUAL POSITION display mode.

The position of the axes is displayed with large characters. The number of the programme, the block, the status of the **G, M, T, S** and **F** functions, as well as PLC messages, if any, comments and the meaning of the function keys, are also displayed.

3.1.2.4. FOLLOWING ERROR display mode.

The axis following error is displayed, as well as the programme number, the block number, the status of the **G, M, T, S** and **F** functions, as well as PLC messages, if any, comments and the meaning of the function keys, are also displayed.

3.1.2.5. ARITHMETIC PARAMETERS display mode.

If the [**PARAMETERS**] function key is pressed, on the upper part of the screen a list of parameters will appear with their corresponding value at that moment. By pressing either of the keys ⬆ and ⬇ the remaining parameters will appear with their values.

For example:

P46 = -1724.9281
P47 = -.10842021 E2

E-2 means ten to the power of minus two.

3.1.2.6. SUBROUTINE STATUS, CLOCK AND PARTS COUNTER display mode.

Identical to the **STANDARD** display mode, except that instead of the following blocks to be run, the subroutines which are active at that moment appear with the following format:

Standard subroutines : **N2.2**

Subroutine number | Number of times still to be run

Parametric subroutines : **P2.2**

Subroutine number | Number of times still to be run

Repetition of subprograms (G25):

G25.2

Indicates that it is a repetition of a subprogram by means of a G25, G26, G27,G28 or G29 function. | Number of times still to be run

Should there be any active canned cycle it is also displayed with the following format:

G2.2

Canned cycle code | Number of times still to be run

The following also appears on the screen in this display mode:

The **CLOCK** which indicates in hours, minutes and seconds the operation time of the CNC in the **AUTOMATIC, SINGLE BLOCK, TEACH IN** and **DRY RUN** modes.

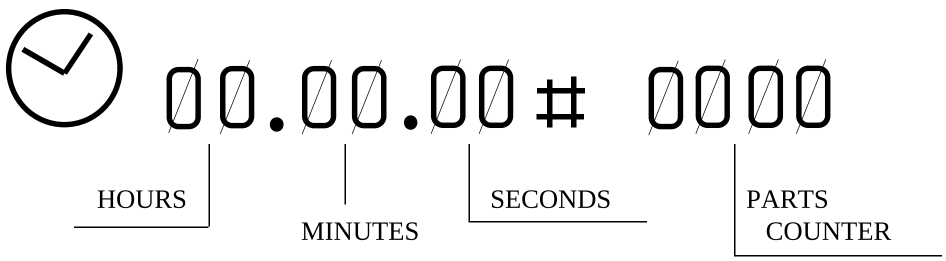
When the running of a program is interrupted or finished, the counting of the clock is also interrupted.

To reset the clock, push the **DELETE** button and then the function key [**TIME**], this clock being displayed on the screen.

On the right, the clock appears with 4 digits **THE PARTS NO. COUNTER**.

This counter increments one unit every time the CNC runs the **M30** function or the **M02** function.

To reset the parts no. counter the **DELETE** key must be pressed and then the function key [**PART COUNT**], this counter being displayed on the screen.



3.1.2.7. GRAPHICS display mode.

This mode is used for the graphic representation of the program and an explanation of it appears in paragraph 3.10 of this MANUAL.

3.1.3. Programming while running a program. BACKGROUND.

The CNC allows the edition of a new program while it is running a cycle in **AUTOMATIC** mode or in **SINGLE BLOCK** mode. For this:

Press the function key [**BACKGROUND EDIT**]

The **P -----** program number which appears corresponds to the number of the last program which was edited.

If the **OP MODE** key is pressed, we return to the Standard Display Mode.

The remaining operations are identical to those in the EDITOR (6).

Atention:

It is not possible to work (edit, correct, etc.) with the program which is being run. It is recommended to give programs numbers which have not been previously stored in the memory, as if the programme which is being run contains calls to subroutines of other programs, there could be problems. Specifically the 001 error may be generated. During an editing operation, the **AUTOMATIC** mode controls and keys or those of the **SINGLE BLOCK** mode remain active.

3.1.4. PLC/LAN mode.

When the **[PLC]** function key is pressed, access is gained to the main menu of the PLC and the LOCAL AREA NETWORK without any need for stopping the execution of the program.

(See the FAGOR PLC 64/INTEGRATED manual).

If the **OP MODE** key is pressed, we return to the **STANDARD** Display Mode.

3.1.5. Verification and modification of the values of the tool offset table without stopping the cycle.

- Press the function key **[TOOL OFFSETS]**
- Key in the number of the offset desired (00-99).
- Press **RECALL**.

The values of the offset which has been called will appear on the screen.

Underneath and to the right, the letter **I** will appear.

If it is wished to modify the value of the **I** on the table, the amount which it is wished to add or subtract is keyed in.

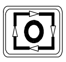
The value keyed in appears on the right of the **I**.

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

Once the values of the tool offset table have been introduced, press the key **[END]** to return to the standard display.

3.1.6. Tool inspection.

If during the running of a program it is wished to inspect or change a tool, the procedure to follow is indicated below:

a) Press 

The programme being run will be interrupted and on the upper right-hand side of the screen the message **INTERRUPTED** shall appear.

b) Press the function key **[TOOL INSPEC].**

At this time, **M05** is run.

On the screen, there will appear:

JOG KEYS AVAILABLE
EXIT

c) By means of the **JOG** keys, the axes can be moved to the desired point.

The **TOOL INSPECTION** sequence allows the spindle to start and stop during the removal of the tool, by means of the spindle operating keys situated on the Control panel.

d) Once the tool has been inspected or changed:

Press **[CONTINUE]** (According to the situation when **[TOOL INSPEC]** is pressed, M03 or M04 are executed).

The screen will show:

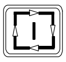
RETURN
AXES NOT POSITIONED

(Axes which have been moved manually).

By means of the **JOG** keys the axes are taken to the position in which the cycle was interrupted. The CNC will not allow this position to be passed.

When the axes are in position, on the screen there will appear:

RETURN
AXES NOT POSITIONED
NONE

e) Press 

The cycle will continue normally.

3.1.7. CNC reset

In **AUTOMATIC** and **SINGLE BLOCK** operating modes, when the **RESET** key is pressed twice, the CNC is reset to switch-on conditions.

When the **RESET** key is pressed the first time, the blinking message **RESET?** will appear on the screen. If **RESET** is not desired, press **CL** to cancel it.

3.1.8. Display and deletion of messages sent by the FAGOR PLC 64

The CNC operates with the **FAGOR PLC** and the latter sends messages for display on the CNC, it is possible to access to a table of messages which are active at that moment.

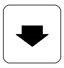
The CNC always displays the message with most priority, if there is more than one active message, the **+** sign will be highlighted (displayed in reverse video).

To display the table, it is necessary to press the **[PLC MESSAGES]** function key.

If there is such a number of messages that they occupy more than one screen, by pressing keys  and  it is possible to display these.

One of the messages will appear highlighted indicating that it can be deleted from the table by pressing the **DELETE** key.

When a deletion is made in this way, the CNC will deactivate the **MARK** corresponding to the PLC which sent the message.

To select the message to delete the  and  keys must be used.

3.2. MODE 2: PLAY-BACK

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

It allows the machine to be operated manually and the coordinate values reached to be entered as program coordinates. The execution of a program requires the following steps:

3.2.1. Selection of the operating mode PLAY-BACK

- Press **OP MODE**
- Press key **2**

The meaning of the function keys to operate in this mode will appear on the screen.

3.2.2. Locking/Unlocking of memory

Same as section 3.6.2. in EDITING mode.

3.2.3. Deletion of a complete program

Same as section 3.6.3. in EDITING mode.

3.2.4. Change of program number

Same as section 3.6.4. in EDITING mode.

3.2.5. Display and search of memorized subroutines

Same as section 3.6.5. in EDITING mode.

3.2.6. Selection of a program

Same as section 3.6.6. in EDITING mode.

3.2.7. Creating a program

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

In a block which only contains the coordinates of one point, after using the **JOG** keys to move the axes, press **ENTER** and the coordinates of the point will be stored in the memory. Every time the **ENTER** key is pressed, the coordinates of the point according to the 3 active axes at that moment will be stored in memory.

In order to activate an axis which is not active at that time, the key of the corresponding axis (X,Y,Z,W,V) must be pressed.

If in addition to the coordinates of a point it is desired to write into the block further information such as **G,S,M,T** functions etc., each time the key of the corresponding axis is pressed the CNC will take as the value of the axis the coordinate at which the machine is at that moment.This method of editing is highly practical when creating a program for copying a part using functions G08 and G09.

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

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

Atention:

If the value **1** is entered in the machine-parameter P610(3), the **START** input is equivalent to **ENTER** key in **PLAY-BACK** operating mode.

3.2.8. Deletion of a block

Same as in EDITING mode (6).

3.2.9. Copy a program

Same as in EDITING mode (6).

3.3. MODE 3: TEACH-IN

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

The execution of a program requires the following steps:

3.3.1. Selection of the operating mode TEACH-IN (3)

- Press **OP MODE**
- Press key **3**

The meaning of the function keys to operate in this mode will appear on the screen.

3.3.2. Locking/Unlocking of memory

Same as section 3.6.2. in EDITING mode.

3.3.3. Deletion of a complete program

Same as section 3.6.3. in EDITING mode.

3.3.4. Change of program number

Same as section 3.6.4. in EDITING mode.

3.3.5. Display and search of memorized subroutines

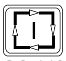
Same as section 3.6.5. in EDITING mode.

3.3.6. Selection of a program

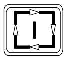
Same as section 3.6.6. in EDITING mode.

3.3.7. Creation of a program

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

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

<u>Atention:</u>

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

By pressing **ENTER** or **DELETE** the display returns to the **TEACH- IN** display mode.

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

Radius compensation cannot be performed in this mode.

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

In machining centers, when **M06** is programmed, the CNC will execute all the movements associated with tool change.

3.3.8. Deletion of a block

Same as in EDITING mode (6).

3.3.9. Copy a program.

Same as in EDITING mode (6).

3.4. MODE 4: DRY RUN

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

3.4.1. Execution of a program

The execution of a program requires the following steps:

3.4.1.1. Selection of the operating mode DRY RUN (4)

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

DRY RUN

0 - G FUNCTIONS
1 - G,S,T,M FUNCTIONS
2 - MAIN PLANE MOVE
3 - RAPID MOVE
4 - THEORETICAL PATH

0 - G FUNCTIONS

The CNC will only execute the programmed G functions.

1 - G,S,T,M FUNCTIONS

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

2 - MOVEMENT ON THE MAIN PLANE

The CNC will execute the **G,S,T,M** functions plus the movements on the main plane.

Three-axis machine

XY plane (G17)
XZ plane (G18)
YZ plane (G19)

Four (five) -axis machine

a) If **W** (**V**) is incompatible with **X**

XY or WY (VY) plane (G17)
YZ or WZ (VZ) plane (G18)
YZ plane (G19)

b) If **W** (**V**) is incompatible with **Y**

XY or XW (VX) plane (G17)
XZ plane (G18)
YZ or WZ (VZ) plane (G19)

c) If **W** (**V**) is incompatible with **Z**

XY plane (G17)
XZ or XW (XV) plane (G18)
YZ or YW (YV) plane (G19)

- The movements are carried out to maximum programmable feedrate (F0), whatever the programmed F's may be.

- The % Feedrate may be varied with the Feedrate Override (M.F.O.) switch.

3 - RAPID TRAVERSE

The CNC will execute the program completely. The movements are executed at max. programmable Feedrate (F0) regardless of the F's programmed. The Feedrate Override allows the % feed to be varied.

It should be borne in mind that if machine parameters P721, P722, P723, P728 are activated the Acceleration/Deceleration control will also be applied in F0, avoiding the generation of following errors.

4 - THEORETICAL PATH

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

3.4.1.1.1. Selection of execution mode

- Key-in the desired number.
- The selected line will appear on the screen completed.

FINAL BLOCK:

N

Will be displayed at the bottom of the screen.

There are two possibilities:

a) If it is desired to run the entire program selected.

- Press ENTER

b) If it is desired to run the program as far as a specific block:

- Key-in the number of the last block whose execution in Dry Run mode is desired including the execution of this block. If this block includes the definition in a canned cycle, it will only be executed until it is positioned at the starting point in the cycle.

- Press **ENTER**

- The letter **P** will appear on the screen.

- Enter the number of the program where the final block is located and then press **ENTER**. If the number of the program is the one already selected, just press **ENTER**.

- The symbol **#** will be displayed.

- After this symbol, enter the number of times that the previous block must be repeated. (Maximum value 9999.)

- Finally press **ENTER**.

In both cases, a) and b), the screen will display the same information as in **AUTOMATIC** and **SINGLE BLOCK**.

3.4.1.2. Selection of the program to be executed

Same as section 3.1.1.2.

3.4.1.3. Selection of starting block

Same as section 3.1.1.3.

3.4.1.4. Display of the contents of the blocks

Same as section 3.1.1.4.

3.4.1.5. Cycle start

Same as section 3.1.1.5.

3.4.1.6. Cycle stop

Same as section 3.1.1.6.

3.4.1.7. Change of operation mode

At any time during the execution of a cycle in the **DRY RUN** operating mode, it can be switched to the operating modes **AUTOMATIC** or **SINGLE BLOCK**. To do this:

- Press **OP MODE**: The operating mode list will appear.
- Press **0** or **1**.

If any number other than **0** or **1** is pressed, the CNC will return to the **DRY RUN** mode.

3.4.1.8. Tool inspection

Same as section 3.1.6.

3.4.2. Display modes

Same as section 3.1.2. except **BACKGROUND EDITING** which is not available.

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

If during the execution of a program in **DRY RUN** mode we change to **AUTOMATIC** or **SINGLE BLOCK** mode, one more block is executed in **DRY RUN** mode before changing over to the mode selected, recovering in the first block of this new mode the position corresponding to the point in the program in which the machine finds itself.

In **DRY RUN** operating mode, by pressing the **RESET** key twice, the CNC is reset to power-on conditions. The first time **RESET** is pressed, the message **RESET?** flashes at the top righthand side of the screen; if it is not desired to carry out **RESET**, press the **CL** key.

3.5. MODE 5: JOG

This operating mode is used for:

- Jogging the axes.

- Searching for the machine-reference points of the axes

- Presetting values on the axes

- Entering or executing **F,S,M**

- Operating as a readout

- Displaying/changing the **RANDOM** table

- **RESET**ting the CNC (return to initial conditions).

- Handwheel operation.

- Measure and load the length of tools in the tool offset table, using a touch probe.

- Starting and stopping the spindle.

3.5.1. Selection of the JOG operating mode (5)

- Press **OP MODE**
- Press key **5**

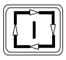
The coordinates of the axes will appear on the screen in large characters.

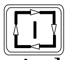
In 5 axis machines, to display the axis which is not active, the corresponding key, i.e., **W** or **V**, must be pressed.

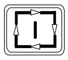
3.5.2. Search for machine reference axis by axis

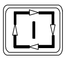
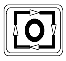
- Once the **JOG** operating mode is displayed, press the key corresponding to the axis to be referenced. In the lower lefthand side of the screen **X,Y,Z,W,** or **V** will appear according to the key pressed.

- Press **[HOME] (ZERO)**. To the right of the axis letter will appear **HOME SEARCH?**.

- Press  The axis will move at a feedrate selected by means of machine-parameter toward the machine-reference point. On pressing the reference microswitch, it will change to a feedrate of 100 mm/min. On receiving the machine-reference pulse from the feedback system, it will stop, setting the counter to the value set as machine-parameter (P119, P219, P319, P419, P519).

If the reference microswitch was pressed when pressing Cycle Start , the axis will withdraw until the microswitch is released. Then the search will be carried out normally.

To cancel the machine reference search before pressing Cycle Start  the **CL** key must be pressed.

To cancel the search after pressing Cycle Start , Cycle Stop  must be pressed.

3.5.3. Presetting a coordinate value

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

- Key in the required value.

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

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

3.5.4. Jogging the axes

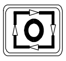
3.5.4.1. Continuous movement

- Front panel (M.F.O.) switch in any position of the % FEEDRATE zone.

- According to the axis and the direction in which it is desired to move, the JOG key corresponding to this axis must be pressed:

- As established by means of the machine-parameter:

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

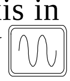
- Press  to stop the movement.

or.

- Press another **JOG** key.
To reverse or transfer the movement of one axis to another.

<u>Atention:</u>

On selecting the **JOG** operating mode the feedrate F0 (defined by parameter P830) remains selected. If this parameter is 0, then P110,P210,P310,P410 and P510, corresponding to axes **X,Y,Z**,(**W**),(**V**), respectively, will determine the max. feedrate for each axis in the **JOG** mode.

Rapid feed of an axis in **JOG** mode can be obtained while pressing the **RAPID FEED** key  .

3.5.4.2. Incremental movement

- Front panel M.F.O. switch in the **JOG** zone.
- Press any of the following keys:

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

Atention:

a) On selecting the **JOG** operating mode the feedrate F0 (defined by parameter P830) remains selected. If this parameter is 0, then P110,P210,P310,P410 and P510, corresponding to axes **X,Y,Z**,(**W**),(**V**), respectively, will determine the max. feedrate for each axis in the **JOG** mode.

Rapid feed of an axis in **JOG** mode can be obtained while pressing the **RAPID FEED** key .

b) The positions of the knob are 1,10,100,1000 and 10000, and indicate the value of the movement in microns or in 0.0001 inches. This value can be limited to 1000 microns or 10000 (of 0.1 or 1 inches) by P609(6).

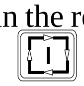
3.5.5. Entering F,S and M

The required values of **F,S** and **M** may be entered in this operating mode.

3.5.5.1. Entering an F value

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

3.5.5.2. Entering an S value

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

3.5.5.3. Entering an M value

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

Atention:

Except for **M41,M42,M43** and **M44** which are automatically generated by the CNC when an **S** value requiring range change is programmed.

3.5.6. Operation of the CNC as a readout

Once the **JOG** operating mode is selected, if the external **MANUAL** input is activated, the CNC acts as a readout.

In this case, the machine has to be moved by means of external controls and the analog signals must be generated outside the CNC.The **S** and **M** functions may be entered in this form of operation.

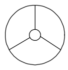
If when operating in this mode, the **software** travel limits (set via machine-parameters) are overrun, the CNC will send the relevant error code and will only allow the machine to be moved to bring it back to the permitted zone.

3.5.7. Change of measurement units

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

3.5.8. Handwheel operation

When an **electronic handwheel** is fitted, with this option the axes can, one at a time, be moved manually. For this:

- Select the **JOG** operation mode.
- Turn the front knob to one of the  positions.
- Press any of the two **JOG** keys which correspond to the axis to be moved by the Handwheel. If a FAGOR Handwheel (mod 100 P) is used, the axis can also be selected by means of the built-in selector button; the relevant axis will be displayed in reverse video on the CRT.
- Turn the Handwheel, the axis will move according to the setting of the relevant machine-parameter multiplied by the factor selected with the switch (X1,X10,X100). It should be borne in mind that if we wish to move an axis at a speed of over G00 corresponding to this axis, the CNC will assume this as maximum, ignoring additional pulses. In this way the generation of following errors is avoided.

To change the axis being jogged:

- Press any of the two JOG keys of the new axis or the axis selector button if a FAGOR Handwheel (mod 100 P) is used.
- Turn the Handwheel.

To end the Handwheel operation.

- Turn the M.F.O. switch to any other position or press the **STOP** key or keep the axis selector button pressed until the **CRT** stops blinking the selected axis, if a FAGOR Handwheel (mod 100 P) is used.

3.5.9. Display/Modification of RANDOM table

I) Display of tool table

It is possible to display at any time the situation of the tool in the magazine. To do so, first select the **JOG** operating mode and then:

- Press **T**. It will appear at the bottom of the screen.
- Key in the number of the tool to be displayed.
- Press **RECALL**. Pxx will appear to the right of the tool number keyed in. The xx(00-99) indicates the position which the tool occupies in the magazine.
- Once a tool has been displayed, the keys ↓ ↑ can be used to display those preceding or following it.

Atention:

If P00 appears, it means that the tool is actually in the spindle.

If P99 appears, it means that the tool is in the tool changer or **M06** has not been executed yet.

If T00 is keyed in, the CNC seeks out the free position in the magazine.

II) Modification of the table

First select the **JOG** operating mode and then:

- Press **T**. It will appear at the bottom of the screen.
- Key in the number of the tool to be modified.
- Press **P**. Key in the store position number which is to be assigned to the preselected tool.
- Press **ENTER**.

Atention:

When it is a **NON-RANDOM** magazine, the only changes allowed on the table are:

- Txx Pxx (assigns position Pxx to tool Txx)
- Txx P0 (assigns the position of the spindle to tool Txx)
- Txx P99 (assigns the position of the Tool changer to tool Txx)

When trying to key any other sequence, the CNC will respond with **?** indicating that such sequence is not possible. Press **CL** to continue.

. If P00 is keyed in, it means that the tool goes to the spindle.

. If P99 is keyed in, it indicates that the tool is in the tool changer. When it is confirmed that a tool is in the spindle (P00), the indication that the tool is in the tool changer (P99) is cancelled. Therefore, if we wish to confirm both positions to the CNC, it must be told which tool occupies the spindle position before "telling" which tool occupies the tool changer.

. If T00 is keyed in, the free position is assigned to it.

. If once the replacing tool has been selected by means of Txx.xx and before executing **M06** the **EMERGENCY** error or power failure occurs, it is possible to indicate which tool is in the tool changer by carrying out the following sequence:

- Select the **JOG** operating mode.

- Key in **T** and the number of the tool in the tool changer.

- Key in **P99**.

- Press **ENTER**.

III) Special tools

Also, In machines with automatic tool changer some tools may occupy more than one position in the magazine. Follow this sequence in order to indicate the CNC which tools are special:

a) Initialize the tool magazine by executing the following sequence in **TEACH-IN** mode.

T99.xx
CYCLE START

b) Go into **JOG** mode and enter the special tools by keying in:

Txx (number of the tool)
S
ENTER

This way, when the position of a special tool is displayed, it will appear:

Txx Pxx S

c) The CNC will automatically assign the two positions next to the one entered (the one before and the one after). So, if by doing this, another “normal” tool has been cancelled,it must be reentered by keying:

Txx (Tool number)
Pxx (Position number)
ENTER

To redefine a special tool as “normal”:

Txx
N
ENTER

<u>Atention:</u>

If an improper programming of tool change originates error code 053 the process to resume the operation is the following:

- Select **JOG** operation mode.
- Key in the number of the tool located in the spindle.
- Key-in **P00**.
- Press **ENTER**.

This way, the number of the tool presently engaged in the spindle has been confirmed to the CNC.

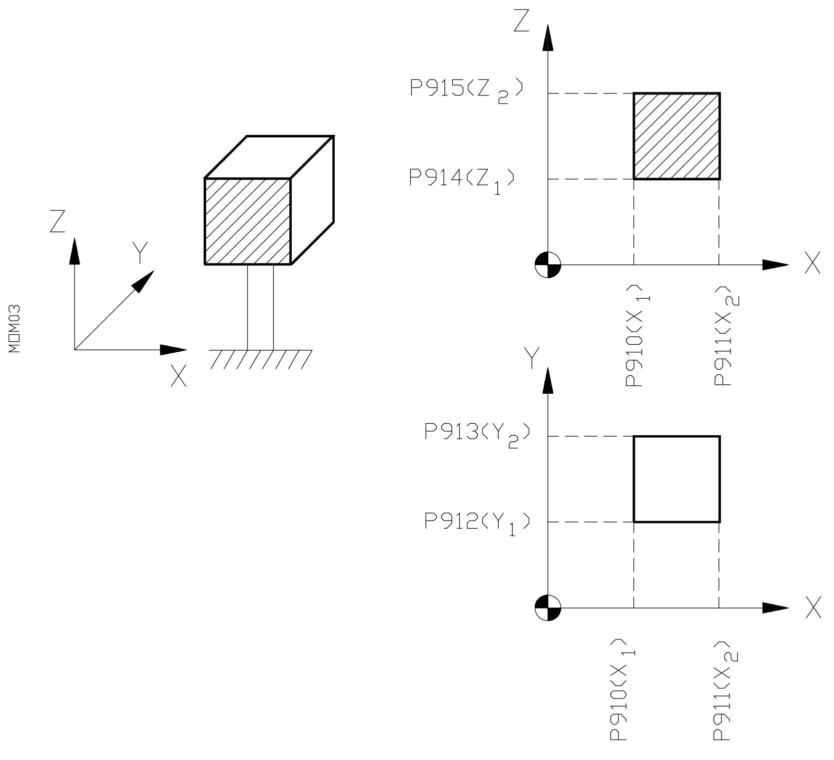
In **JOG** operating mode, pressing the **RESET** key returns the CNC to power-on conditions.

3.5.10. Measuring and loading of tool offsets with a probe.

With this CNC, in the **JOG** mode the tool dimensions can be quickly measured and loaded with a probe. To do this, a tool measuring probe must be installed with its sides parallel to the axes and in an established position on the machine.

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

P910 minimum (X1) value along the X axis
P911 maximum (X2) value along the X axis
P912 minimum (Y1) value along the Y axis
P913 maximum (Y2) value along the Y axis
P914 minimum (Z1) value along the Z axis
P915 maximum (Z2) value along the Z axis



The sequence to be followed is:

1- Press the **[TOOL MEASUREMENT]** key.

2- Place the tool to be measured in the tool holder.

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

4- Select the tool offset number by keying in: **Txx START**

5- Press the **JOG** key that indicates in which direction the axis must be moved to carry out the probing movement. The feedrate is established by **P804**.

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

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

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

To go back to the **JOG** mode, press the **[TOOL MEASUREMENT]** key.

3.5.11. Spindle operating keys.

By means of these keys on the front panel, the spindle can be started in both directions as well as stopping the spindle from turning, as long as the corresponding **S** has been programmed, without need for executing M3,M4 or M5.

By means of the [+] and [−] keys it is possible to vary the **S** turning speed % programmed.

3.6. MODE 6: EDITING

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

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

3.6.1. Selection of the EDITOR (6) operating mode:

- Press **OP MODE**
- Press key **6**

The meaning of the function keys to operate in the **MODE** will appear on the screen.

3.6.2. Locking/Unlocking of memory

- Press **[LOCK/UNLOCK MEMORY]. CODE** appears on the screen:

- Key in: **MKJIY** to lock the memory.
  MKJIN to unlock the memory.

- Press **ENTER**.

<u>Atention:</u>

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

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

3.6.3. Part-program directory

- Press **[PROG**ram **DIR**ectory**]**. The CNC shows a list of up to 7 part-programs with their sizes (in characters) as well as the total free memory available.

  Also, if the first block of each program has a comment, it will appear next to the program size.

  Example:

PROG.	**CHAR.**	
00001	42	PART 1
00002	115	PART 2

28513 free characters

Atention:

When having more than 7 programs in memory, use the ↓ ↑ keys to scroll them up and down.

3.6.3.1. Delete a complete program

- Press **[PROG DIR]**. T he screen will show: **DELETE PROGRAM**.
- Press **DELETE**. The message **DELETE PROGRAM** appears on the screen.
- Key in the number of the program to be deleted. Check the number. If the number is correct, press **ENTER**.

  If the number is not correct:

- Press the **CL** key. We cancel the number with this key.
- Key-in the correct number.
- Press **ENTER**.

Atention:

If the **[CONTINUE]** key is pressed during this sequence, access is obtained to the original display of this MODE.

DELETION OF ALL PROGRAMS

If all the programs in the memory must be deleted, key-in 99999 when **DELETE PROGRAM** is displayed, and then press **ENTER**; if the key **Y** is pressed immediately afterwards, all the programs except the one identified by parameter **P802** will be deleted.

3.6.4. Change of program number

- Press **[PROG RENAME]**. The screen will display:

OLD:P

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

- Press **ENTER**. The screen will then display:

NEW: P

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

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

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

PROGRAM NUMBER: P ——-
DOES NOT EXIST IN MEMORY

- If there is a program already with that number, the screen will display:

ALREADY EXISTS IN MEMORY.

Atention:

During this sequence if the **[CONTINUE]** key is pressed, access is obtained to the original display of this **MODE**.

3.6.5. Display and search of subroutines stored in memory

- By pressing **[STANDARD SUBROUTINE DIRECTORY]** or **[PARAMETER SUBROUTINE DIRECTORY]** all the subroutines, standard or parametric, recorded in the CNC memory are displayed.

- To find out which programs contain the subroutines displayed on the screen, key in the subroutine number and press **RECALL**.

  The number of the program where this subroutine is found will appear on screen.

  To repeat the process for another subroutine, press **DELETE** or the **[SUBRTS]** key and repeat the previous sequence.

Atention:

During this sequence, if the **[CONTINUE]** key is pressed, the access is gained to the original display of this **MODE**.

3.6.6. Selection of a program

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

- If a different program is wanted :

  - Press the **[PROGRAM SELECTION]** key.
  - Key in the program number.
  - Press **[CONTINUE]**. The program selected will appear on the screen.

3.6.7. Creating a program

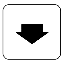
If there is a program in the CNC's memory with the same number as the one to be recorded, there are two methods for recording the new program:

- Completely erase the existing program.

- Not to erase it and write it block by block (as described further on) over the existing program, taking care to assign the same numbering as the previously recorded blocks to the blocks being written.If there is no other program in memory with the same number, proceed as follows:

3.6.7.1 Displaying the block contents

To scroll the blocks being displayed up and down:

- Press  : The display scrolls up one line (block).
- Press  : The display scrolls down one line (block).

On models with 512 Kb of part-program memory (models: MK, MGK, MSK, GPK, MIK, MGIK, MSIK, GPIK), the following keys and softkeys are also available:

- Press  : The display scrolls up 5 lines (blocks).

- Press  : The display scrolls down 5 lines (blocks).

- Press **[BEGIN]** : The display shows the first blocks.

- Press **[END]** : The display shows the last blocks.

3.6.7.2. Unassisted programming

Format of a block

(dimensions in millimeters)

N4 G2 (V)+/-4.3(W)+/-4.3 X+/-4.3 Y+/-4.3 Z+/-4.3 F5.4 S4 T2.2 M3 (in this order)

(dimensions in inches)

N4 G2 (V)+/-3.4(W)+/-3.4 X+/-3.4 Y+/-3.4 Z+/-3.4 F5.5 S4 T2.2 M3 (in this order)

Programming in the same block of the fourth axis **W**, of the fifth axis **V** and the one associated to both which is indicated in the machine parameter P11, are incompatible.

Programming:

The CNC automatically numbers the blocks:

In multiples of 10 on non-"K" model CNCs.
In multiples of 5 on "K" model CNCs

If a different block number is desired, press **CL** and then:

- Key in the block number. It will appear on the lower left-hand side of the screen. The blocks may not be correlative.

- If a normal conditional block is desired, after keying in the block number, press [•] (decimal point) and if a special conditional block is required press [•] again.

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

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

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

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

- A comment can be written which must be placed within brackets.

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

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

Refer to the **PROGRAMMING MANUAL** for incompatibilities when programming various functions.

3.6.7.3. Modification and deletion of a block

I) During the writing process

a) Modification of characters

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

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

b) Insertion of characters

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

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

II) Block already entered in memory

a) Modification and insertion of characters

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

b) Deletion of the block

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

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

3.6.7.4. Assisted programming

Access to assisted programming is available in any of the program editing modes, i.e. **PLAY BACK (2)**, **TEACH-IN (3)** or **EDITING (6)**. For this, if, during the writing of a block the **[HELP]** key is pressed, the cursor which is found in the block to be written will disappear and the screen will display:

PROGRAMMING GUIDE

1 - MOVEMENT PROGRAMMING
2 - CANNED CYCLES
3 - SUBROUTINES/JUMPS
4 - GEOMETRIC AIDE
5 - ARITHMETICAL FUNCTIONS
6 - G FUNCTIONS
7 - M FUNCTIONS

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

Supposing, for example, that when editing a program it is desired to program in a block the canned cycle for rectangular pocket milling, the sequence will be:

Press **[HELP]**

Press **2**

Press ⬇

Press **4**

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

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

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

a) When nothing is written in the block, press **RECALL** if the cursor is displayed (if it is not, press **[HELP]).**

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

SPECIAL ASSISTED PROGRAMMING

During the edition of a canned cycle, whenever the corresponding preparatory function key has been pressed, when the **[HELP]** key is pushed, the information corresponding to this canned cycle will appear directly on the screen highlighting the parameter to be introduced.

Once the value has been introduced and in order to be able to continue with the edition of new parameters, it is necessary to press the **ENTER** key.

If it is not required to program any parameter, as long as it is not obligatory to do so, the **DELETE** key must be pressed.

As in the case of normal programming, the **CL** key deletes one character at a time and the **DELETE** key deletes the whole value given to the current parameter.

At any time during this type of programming, if the **[HELP]** function key is pressed, the control changes over to the normal assisted programming.

3.6.7.5. Save a program being edited (only on models with 512 Kb of memory).

Models having 512 Kb of part-program memory (MK, MGK, MSK, GPK, MIK, MGIK, MSIK, GPIK) have an additional RAM memory for program editing and modification.

The program, or portion of it, being edited is entered again in memory when exiting the editing mode.

If, for any reason, the CNC loses power while editing a program, all the data stored in the RAM memory will be lost. In other words, all the changes made to the program will be lost but not the program itself.

To avoid this problem, press **[SAVE]** once in a while so the CNC stores the current changes in the user memory.

3.6.7.6. Copying a program.

This feature allows an existing program to be copied in the CNC memory, by designating it a number which is different from the original.

To do this, press first **[PROG**ram **DIR**ectory**]** and then the **[COPY]** key.

The CNC will ask which number is that of the source program and which is the one for the new program. After keying in each of them the **ENTER** key must be pressed.

Should there be no number keyed in as the original program, as there is another program with the same number in the memory and the one keyed in as being new, or if there is not sufficient memory in the CNC, a message will be displayed indicating the cause.

3.7. MODE 7: PERIPHERALS

This is used for transferring part programs or machine- parameters from/to peripherals. The method of working in this operating mode is as follows:

3.7.1. Selection of the operating mode PERIPHERALS (7)

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

PERIPHERALS

0 . RECEIVE FROM CASSETTE
1 . SEND TO CASSETTE
2 . RECEIVE FROM GENERAL DEVICE
3 . SEND TO GENERAL DEVICE
4 . CASSETTE DIRECTORY
5 . DELETE CASSETTE PROGRAM
6 . DNC ON/OFF

Atention:

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

The CNC must be **OFF** when connecting/disconnecting peripheral units.

When using the FAGOR cassette recorder, parameter P607(4) must be set to 0.

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

- Press the **0** key. The screen will display:

  PROGRAM NUMBER: P

- Key in the number of the program to be read in. If 99999 is entered, the CNC gets ready to accept machine-parameters, the decoded **M**'s functions table and the table of leadscrew compensation parameters.
  Should a PLC I be fitted, the PLC user program will be kept together with the above.

- Press **ENTER**. Four possibilities:

a) A program exists in the control's memory with the same number. The screen will display:

   ALREADY EXISTS IN MEMORY
   DELETE?

   If deletion is not wanted:

   - Press any key other than **Y**. Return to the state in section 3.7.1.

   If deletion is wanted:

   - Press **Y**. The screen will display:

   PROGRAM NUMBER: P —— DELETED

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

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

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

DOES NOT EXIST IN THE CASSETTE

- Press **[CONTINUE]**. It returns to the status of section 3.7.1. or,

- Press **OP MODE**. The operating mode menu will appear.

c) The program selected exists on the tape and not in the CNC memory.

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

- If in the program being read there is any erroneous block number (example, Nxxxxx) the screen will display:

PROGRAM NUMBER: P —— RECEIVED
INCORRECT DATA RECEIVED
N xxxxx

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

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

PROGRAM NUMBER: P —— RECEIVED

That means that the CNC carries out a syntactic test of the program. If there is any programming error the relevant error code and the affected block are displayed and the program is loaded completely.

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

3.7.2.1. Transmission errors

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

- If during transmission **INCORRECT DATA RECEIVED** appears on the screen.

This indicates that there is an incorrect character on the tape, or a non permitted block number has been written.

Atention:

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

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

- Press key **1**. The screen will display:

  PROGRAM NUMBER: P ——-

- Key-in the number of the program to be transferred.

  If P99999 is entered, the CNC gets ready to transmit machine- parameters, **M** functions decoded table and the leadscrew error compensation table.

- Press **ENTER**.

Three possibilities:

a) The selected program does not exist in the CNC memory. The screen will display:

DOES NOT EXIST IN MEMORY

- Press **[CONTINUE]**. We return to the status of section 3.7.1. or,

- Press **OPERATE MODE**. The operating mode menu will appear:

b) There is a program with the same number on the tape. When pressing **ENTER** the screen will display:

ALREADY EXISTS IN THE CASSETTE
DELETE?

If deletion is not wanted:

- Press any key other than Y. This returns to the status of section 3.7.1.

If deletion is wanted:

- Press **Y**. The screen will display:

PROGRAM NUMBER: P —— DELETED

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

c) The selected program exists at the CNC but not on the tape.

The transfer takes place normally. During this process the screen display:

SENDING

On completion the following text will appear:

PROGRAM NUMBER: P —— SENT

3.7.3.1. Transmission errors

Same as section 3.7.2.1.

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

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

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

<u>Atention:</u>

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

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

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

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

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

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

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

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

3.7.6. FAGOR cassette directory (4)

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

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

- Pressing **[CONTINUE]** returns to the status of section 3.7.1.

3.7.7. Deletion of a FAGOR cassette program (5)

- Press the **5** key. The screen will display:

PROGRAM NUMBER: P

- Key in the number of the selected program.

- Press **ENTER**.

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

PROGRAM NUMBER: P —— DELETED

- Press **[CONTINUE]**. The status of section 3.7.1. returns or,

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

3.7.8. Interruption of the transmission process

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

The screen will display:

PROCESS ABORTED

3.7.9. DNC. Communication with a computer

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

. Directory and program deletion commands.

. Transfer of programs and tables.

. Execution of infinite programs.

. Machine's remote control.

. Advanced **DNC** system's status report.

To activate the **DNC** feature, P607(3) must be **1**. Also, **PERIPHERALS** (**DNC ON/OFF**) mode **6** must show the highlighted message **ON**. Otherwise, press **6**. See DNC manual for more detailed information.

In **PERIPHERALS** operating mode (7), every time **RESET** is pressed, the CNC returns to power-on conditions.

3.8. MODE 8: TOOL OFFSET AND ZERO OFFSETS G53/G59

This is used to enter into the memory the dimensions (length and radius) of up to 100 tools and the values of up to **7** zero offsets (G53-G59).

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

3.8.1. Selection of the operating mode TOOL OFFSET (8)

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

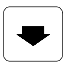
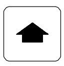
TOOL OFFSET/G53-G59

T00	R —— . —--	L ——	. —--
	I -—- . —--	K ----	. —---
T01	R —— . —--	L ——	. —--
	I -—- . —--	K —--	. —---
T02	R —— . —--	L ——	. —--
	I —-- . —--	K —--	. —---

3.8.2. Read-out of tool table

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

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

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

3.8.3. Entering the dimensions of the tools

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

- Press **R**.

- Key in the value of the radius of the tool. Max. value: +/- 999.999 mm or +/-39.3700 inch.

- Press **L**.

- Key in the value of the length of the tool. Max. value:+/-999.999 mm or +/-39.3700 inch.

- Press **I**. Key in its value. Maximum value +/-32.766 mm or +/-1.2900 inches.

- Press **K**. Key in its value Maximum value +/-32.766 mm or +/-1.2900 inches.

- Press **ENTER**. (If what is written is correct). The values are entered into memory.

3.8.4. Modification of tool dimensions

I) During the writing process

a) Modification of characters

If during the writing of the dimensions of a tool already written has to be modified (R,L,I,K or any number):

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

b) Insertion of characters

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

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

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

- Key in the new characters required.

- Press **INS**. The blinking stops.

II) Dimensions already entered in memory

- Key in the tool number concerned.

- Press **RECALL**.

- Proceed as in the previous item.

- Press **ENTER**. The modified dimensions are entered into the memory.

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

- A block that has been written can be completely erased by pressing **DELETE**, if the cursor is situated at the beginning of the block.

3.8.5. Change of measuring units

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

3.8.6. Zero offsets G53/G59

In the same operation mode (8) if the key G is pressed the screen will display:

TOOL OFFSETS/G53-G59

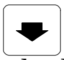
```
G53 V ---- . --  W ——— . —  X —-- . —-
        Y ——— . —  Z  ——— . —-
G54 V  ---- . -- W ——— . —  X ——— . -—
        Y ——— . —  Z  ——— . —-
G55 V ----  . -- W ——— . —  X ——— . -—
        Y ——— . —  Z  ———    .       —-
```

3.8.6.1. Read-out of zero offset table

If a readout is wanted of the values of a zero offset which does not appear on the screen, there are two methods:

a) Key in the number of the zero offsets (G53/G59)

Press **RECALL**

b) Press  or  to move the zero offset displayed back and forth until the required one is reached.

3.8.6.2. Entering zero offsets values

- Key in the number of the zero offset (G53-G59).
- Write the desired values for **W**,**X**,**Y**,**Z**.
- Press **ENTER**.

Atention:

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

3.8.6.3. Modification of zero offset values

Same as 3.8.4.

3.8.6.4. Change of measurement units

Same as 3.8.5.

3.8.7. Return to the tool offset table

When the zero offset table is being displayed, the tool table can be recovered by pressing **T**.

3.8.8. Complete deletion of tool offsets or zero table

- Key in **K**,**J**,**I**.
- Press **ENTER**.

The displayed table is completely erased.

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

3.9. MODE 9: SPECIAL MODES

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

3.10. GRAPHICS

CNC 8030 model **MS** or **MG** have **GRAPHIC REPRESENTATION** and by means of this feature the tool path can be displayed on the CRT, as the program is being executed.

This feature can be applied in one of the following modes: **AUTOMATIC**, **SINGLE BLOCK**, **TEACH IN**, **DRY RUN**.

In **DRY RUN** mode, if **THEORETICAL PATH (4)** is selected, the system checks the program and displays the theoretical tool- center’s path in solid lines, ignoring its dimensions.

Nevertheless, if mode **0** or **1** is selected in the same operating mode (**DRY RUN)**, the tool center’s path will be displayed in dotted lines.

If, when executing a program in **DRY RUN** operation in modes **0**,**1** or **4**, there is a block involving movement plus the function (Tx.x) the relevant path will not be displayed unless the machine is a machining center.

In the remaining modes, the tool’s real path is displayed in dotted lines. The distance between dots varies according to the value of **F**.

3.10.1. Display area definition

Prior to the representation of graphics on the CRT, the display area must be defined before the program is run. To do this, after selecting the desired operation mode.

- Press the **[GRAPHICS]** key.
- Press the **[DEFINE AREA-G]** key.

At this time the CNC asks which are the views required for viewing. Press **Y**/**N** to identify desired/not desired views. The CNC displays then the four possible views:

- XY plane
- XZ plane
- YZ plane
- Three-dimensional

Key in the coordinate values (**X**,**Y** and **Z**) of the point desired to be at the center of the screen, and the width of the image. Press **ENTER** after every value.

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

To display the desired view (max. **3** of the possible **4**) press:

[XY] for X-Y plane
[XZ] for X-Z plane
[YZ] for Y-Z plane
[3D] for three-dimensional

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

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

When a program is being run in the **DRY RUN** operating mode, it is possible to vary the speed the diagram is drawn on the screen, by means of the **FEED RATE** switch.

3.10.2. Zooming (windowing)

The CNC has a **ZOOM** function by which entire graphics or parts of them can be enlarged or reduced by this feature. To use this **ZOOM** function the program must be either interrupted or completed.

Press the key which corresponds to the view in which the zooming is desired. Then press [**ZOOM**] and a rectangle identifying the window will be displayed over the existing graphic.

Its dimensions can be altered pressing [+] or [−] on the front panel and its position by using cursor moving keys.

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

If **[EXECUTE]** is pressed, the windowed area will fill the CRT.

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

To repeat the whole **ZOOM** sequence, start by pressing **[ZOOM]** as before.

To exit the **ZOOM** mode and continue, press **[END]**.

3.10.3. Redefinition of the display area by the ZOOM function

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

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

Atention:

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

When the **ZOOM** function is used, it is necessary to bear in mind that the CNC will keep information on approximately the last 500 blocks with movement which have been executed, therefore, if the programme has more blocks with movement, only those retained will appear in the new diagram.

3.10.4. Deletion of graphics

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

3.10.5 Graphic representation in colour (CNC 8030 MS)

Whenever only one of the **4** views possible have been selected, every time the Tool (T2) is changed, the path will be drawn in a different color (3 colors).

ERROR CODES

001 This error occurs in the following cases:

> When the first character of the block to be executed is not an "N".

> When while BACKGROUND editing, the program in execution calls a subroutine located in the program being edited or in a later program.

The order in which the part-programs are stored in memory are shown in the part-program directory. If during the execution of a program, a new one is edited, this new one will be placed at the end of the list.

002 Too many digits when defining a function in general.

003 A negative value has been assigned to a function which does not accept the (-) sign or an incorrect value has been given to a canned cycle parameter.

004 A canned cycle has been defined while function G02, G03 or G33 was active.

005 Parametric block programmed wrong.

006 There are more than 10 parameters affected in a block.

007 Division by zero.

008 Square root of a negative number.

009 Parameter value too large

010 M41, M42, M43 or M44 has been programmed.

011 More than 7 "M" functions in a block.

012 This error occurs in the following cases:

> Function G50 is programmed wrong

> Tool dimension values too large.

> Zero offset values (G53/G59) too large.

013 Cycle defined incorrectly.

014 A block has been programmed which is incorrect either by itself or in relation with the program history up to that instant.

015 Functions G20, G21, G22, G23, G24, G25, G26, G27, G28, G29, G30, G31, G32, G50, G52, G53, G54, G55, G56, G57, G58, G59, G72, G73, G74, G92 and G93 must be programmed alone in a block.

016 The called subroutine or block does not exist or the block searched by means of special function F17 does not exist.

017 This error is issued in the following cases:

> Negative or too large thread pitch value.

> Function G95 or M19 has been used with machine parameter "P800=0".

018 Error in blocks where the points are defined by means of angle-angle or angle-coordinate.

019 This error is issued in the following cases:

> After defining G20, G21, G22 or G23, the number of the subroutine it refers to is missing.

> The "N" character has not been programmed after function G25, G26, G27, G28 or G29.

> Too many nesting levels.

020 The axes of the circular interpolation are not programmed correctly.

021 There is no block at the address defined by the parameter assigned to F18, F19, F20, F21, F22.

022 An axis is repeated when programming G74.

023 K has not been programmed after G04.

024	The decimal point is missing when programming T2.2 or N2.2.

025	Error in a definition block or subroutine call, or when defining either conditional or unconditional jumps.

026	This error is issued in the following cases:

> Memory overflow.

> Not enough free tape or CNC memory to store the part-program.

027	I/J/K has not been defined for a circular interpolation or thread.

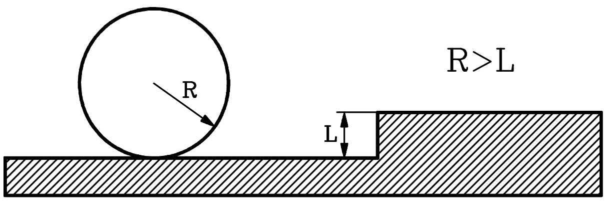
028	An attempt has been made to select a tool offset at the tool table or a non-existent external tool (the number of tools is set by machine parameter).

029	Too large a value assigned to a function.

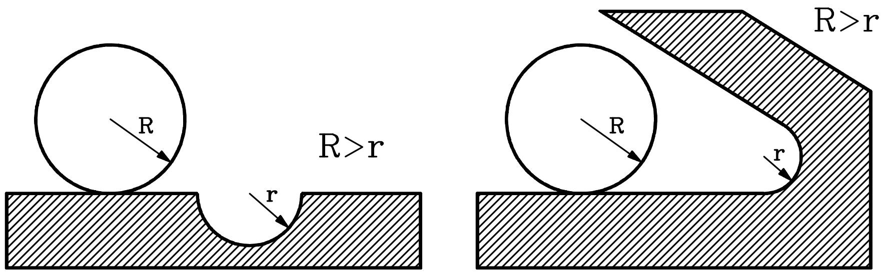
This error is often issued when programming an F value in mm/min (inch/min) and, then, switching to work in mm/rev (inch/rev) without changing the F value.

030	The programmed G function does not exist.

031	Tool radius value too large.



032	Tool radius value too large.



033	A movement of over 8388 mm or 330.26 inches has been programmed.

Example: Being the X axis position X-5000, if we want to move it to point X5000, the CNC will issue error 33 when programming the block N10 X5000 since the programmed move will be: 5000 - (-5000) = 10000 mm.

In order to make this move without issuing this error, it must be carried out in two stages as indicated below:

```
N10 X0        ; 5000 mm move
N10 X5000     ; 5000 mm move
```

034	S or F value too large.

035	Not enough information for corner rounding, chamfering or compensation.

036	Repeated subroutine.

037	Function M19 programmed incorrectly.

038 Function G72 or G73 programmed incorrectly.

It must be borne in mind that if G72 is applied only to one axis, this axis must be positioned at part zero (0 value) at the time the scaling factor is applied.

039 This error occurs in the following cases:

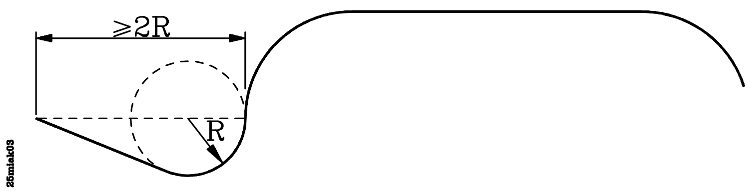
> More than 15 nesting levels when calling subroutines.

> A block has been programmed which contains a jump to itself. Example: N120 G25 N120.

040 The programmed arc does not go through the defined end point (tolerance 0.01mm) or there is no arc that goes through the points defined by G08 or G09.

041 This error is issued when programming a tangential entry as in the following cases:

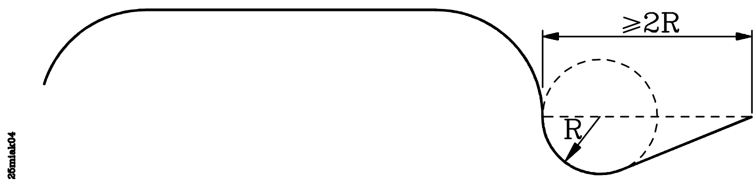
> There is no room to perform the tangential entry. A clearance of twice the rounding radius or greater is required.



> If the tangential entry is to be applied to an arc (G02, G03), The tangential entry must be defined in a linear block.

042 This error is issued when programming a tangential exit as in the following cases:

> There is no room to perform the tangential exit. A clearance of twice the rounding radius or greater is required.



> If the tangential exit is to be applied to an arc (G02, G03), The tangential exit must be defined in a linear block.

043 Polar origin coordinates (G93) defined incorrectly.

044 Canned cycle defined wrong.

045 Function G36, G37, G38 or G39 programmed incorrectly.

046 Polar coordinates defined incorrectly.

047 A zero movement has been programmed during radius compensation or corner rounding.

048 W axis programmed wrong.

049 Chamfer programmed incorrectly.

050 Functions M06, M22, M23, M24, M25 must be programmed alone in a block.

051 * A tool or pallet change cannot be performed without being in the change position.

052 * The requested tool is not in the magazine.

053 * This error occurs when having a machining center and 2 different external Ts have been programmed in a row without programming an M06 in between.

054 There is no disk in the FAGOR Floppy Disk Unit, there is no tape in the cassette reader or the reader head cover is open.

055 Parity error when reading or writing a floppy or cassette.

056 This error comes up in the following cases:

> When the memory is locked and an attempt is made to generate a CNC program by means of function G76.

> When trying to generate program P99999 or a protected program by means of function G76.

> If function G76 is followed by function G22 or G23.

> If there are more than 70 characters after G76.

> If function G76 (block content) has been programmed without having programmed G76 P5 or G76 N5 before.

> If in a G76 P5 or G76 N5 type function does not contain the 5 digits of the program number.

> If while a program is being generated (G76 P5 or G76 N5), its program number is changed without cancelling the previous one.

> If while executing a G76 P5 type block, the program referred to is not the one edited. In other words, that another one has been edited later or that a G76 P5 type block is executed while a program is being edited in background.

057 Write-protected floppy disk or cassette.

058 Irregular floppy drive motion or sluggish tape transport.

059 Communication error between the CNC and the FAGOR Floppy Disk Unit or between the CNC and the cassette reader.

060 Internal CNC hardware error. Consult with the Technical Service Department.

061 Battery error.

The memory contents will be kept for 10 more days (with the CNC off) from the moment this error occurs. The whole battery module located on the back must be replaced. Consult with the Technical Service Department.

Due to danger of explosion or combustion: do not try to recharge the battery, do not expose it to temperatures higher than 100°C (232°F) and do not short the battery leads.

064 * External emergency input (pin 14 of connector I/O1) is activated.

065 * This error comes up in the following cases:

> If while probing (G75) the programmed position is reached without receiving the probe signal.

> If while executing a probing canned cycle, the CNC receives the probe signal without actually carrying out the probing move itself (collision).

066 * X axis travel limit overrun.

It is generated either because the machine is beyond limit or because a block has been programmed which would force the machine to go beyond limits.

067 * Y axis travel limit overrun.

It is generated either because the machine is beyond limit or because a block has been programmed which would force the machine to go beyond limits.

068 * Z axis travel limit overrun.

It is generated either because the machine is beyond limit or because a block has been programmed which would force the machine to go beyond limits.

069 * W axis travel limit overrun.

It is generated either because the machine is beyond limit or because a block has been programmed which would force the machine to go beyond limits.

070 ** X axis following error.

071 ** Y axis following error.

072 ** Z axis following error.

073 ** W axis following error.

074 ** Spindle speed value too large.

075 ** X axis feedback error. Connector A1.

076 ** Y axis feedback error. Connector A2.

077 ** Z axis feedback error. Connector A3.

078 ** W axis feedback error. Connector A4.

079 ** Spindle feedback error. Connector A5.

080 ** Handwheel feedback error. Connector A5.

081 ** V axis feedback error. Connector **A**5.

082 ** Parity error in general parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

083 ** Parity error in V axis parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

084 * V axis travel limit overrun.

085 ** V axis following error.

086 Not being used at this time.

087 ** Internal CNC hardware error. Consult with the Technical Service Department.

088 ** Internal CNC hardware error. Consult with the Technical Service Department.

089 * All the axes have not been homed.

This error comes up when it is mandatory to search home on all axes after power-up. This requirement is set by machine parameter.

090 ** Internal CNC hardware error. Consult with the Technical Service Department.

091 ** Internal CNC hardware error. Consult with the Technical Service Department.

092 ** Internal CNC hardware error. Consult with the Technical Service Department.

093 ** Internal CNC hardware error. Consult with the Technical Service Department.

094 Parity error in tool table or zero offset table G53-G59. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

095 ** Parity error in W axis parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

096 ** Parity error in Z axis parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

097 ** Parity error in Y axis parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

098 ** Parity error in X axis parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

099 ** Parity error in M table. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

100 ** Internal CNC hardware error. Consult with the Technical Service Department.

101 ** Internal CNC hardware error. Consult with the Technical Service Department.

105 This error comes up in the following cases:
> A comment has more than 43 characters.
> A program has been defined with more than 5 characters.
> A block number has more than 4 characters.
> Strange characters in memory.

106 ** Inside temperature limit exceeded.

107 ** Error in W axis leadscrew error compensation parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

108 ** Error in Z axis leadscrew error compensation parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

109 ** Error in Y axis leadscrew error compensation parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

110 ** Error in X axis leadscrew error compensation parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

111 * FAGOR LAN line error. Hardware installed incorrectly.

112 * FAGOR LAN error. It comes up in the following instances:

> When the configuration of the LAN nodes is incorrect.
> The LAN configuration has been changed. One of the nodes is no longer present (active).

When this error occurs, access the LAN mode, editing or monitoring, before executing a program block.

113 * FAGOR LAN error. A node is not ready to work in the LAN. For example:

> The PLC64 program is not compiled.
>A G52 type block has been sent to an 82CNC while it was in execution.

114 * FAGOR LAN error. An incorrect command has been sent out to a node.

115 * Watch-dog error in the periodic module.

This error occurs when the periodic module takes longer than 5 milliseconds.

116 * Watch-dog error in the main module.

This error occurs when the main module takes longer than half the time indicated in machine parameter "P741".

117 * The internal CNC information requested by activating marks M1901 thru M1949 is not available.

118 * An attempt has been made to modify an unavailable internal CNC variable by means of marks M1950 thru M1964.

119 Error when writing machine parameters, the decoded M function table and the leadscrew error compensation tables into the EEPROM memory.

This error may occur when after locking the machine parameters, the decoded M function table and the leadscrew error compensation tables, one tries to save this information into the EEPROM memory.

120 Checksum error when recovering (restoring) the machine parameters, the decoded M function table and leadscrew error compensation tables from the EEPROM memory.

150 Incoherent data in the 512 Kb memory.

When this error occurs, save as many programs as you can into the Floppy Disk Unit, peripheral or PC.

Then, proceed as follows to format the 512 Kb memory (when doing this, all part-programs stored in this memory will be lost).

Press **[OP MODE] [6]** to select the Editing mode.
Press **[LOCK/UNLOCK]** the screen displays the text: CODE:
Key in: **FM512** and press **[ENTER]**

Once the 512 Kb memory is formatted, recover (restore) the programs you saved into the Floppy Disk Unit, peripheral or PC.

151 Defective 512 Kb memory. Consult with the Technical Service department.

152 Not enough available free space in the 512 Kb memory.

Atention:

The **ERRORS** indicated with "*" behave as follows:

They stop the axis feed and the spindle rotation by cancelling the Enable signals and the analog outputs of the CNC.

They interrupt the execution of the part-program of the CNC if it was being executed.

The **ERRORS** indicated with "**" besides behaving as those with an "*", they activate the INTERNAL EMERGENCY OUTPUT.

FAGOR 8025/8030 CNC

Models: M, MG, MS, GP

PROGRAMMING MANUAL

Ref. 9701 (in)

ABOUT THE INFORMATION IN THIS MANUAL

This manual is addressed to the machine operator. It describes how to operate with this 8025 CNC.

It includes the necessary information for new users as well as advanced subjects for those who are already familiar with this CNC product.

It may not be necessary to read this whole manual. Consult the list of "New Features and Modifications" which will indicate to you the chapters and sections describing them.

Consult the Comparison Table in order to find the specific features offered by your particular CNC model.

There is also an appendix on error codes which indicates some of the probable reasons which could cause each one of them.

Notes: 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.

INDEX

Section | Page

	Comparison table for mill model FAGOR 8025/8030 CNC	ix
	New features and modifications	xv
	INTRODUCTION	
	Safety Conditions	Intr. 3
	Material Returning Terms	Intr. 5
	Fagor Documentation for the 8025/30 M CNC	Intr. 6
	Manual Contents	Intr. 7
1.	Overview	1
1.1.	External programming	1
1.2.	Text programming	2
1.3.	DNC connection	2
1.4.	The FAGORDNC	3
2.	Creating a program	4
3.	Program format	5
3.1.	Parametric programming	6
4.	Program numbering	7
5.	Program blocks	7
5.1.	Block numbering	7
5.2.	Conditional blocks	8
6.	Preparatory functions	9
6.1.	Table of G functions used at the CNC	9
6.2.	Types of movements	12
6.2.1.	G00. Positioning	12
6.2.2.	G01. Linear interpolation	13
6.2.3.	G02/G03. Circular/helical interpolation	14
6.2.3.1.	Circular interpolation	15
6.2.3.2.	Circular interpolation in cartesian coordinates by programming the radius	23
6.2.3.3.	G06. Circular interpolation with absolute center coordinates	24
6.2.3.4.	Helical interpolation	25
6.3.	G04. Dwell	27
6.4.	Transition between blocks	28
6.4.1.	G05. Round corner	28
6.4.2.	G07. Square corner	29
6.5.	G08. Arc tangent to previous path	30
6.6.	G09. Arc programmed by three points	32
6.7.	Mirror image	34
6.8.	Plane selection	36
6.9.	G25. Unconditional jump/call	37
6.10.	G31/G32. Storage and retrieval of part program's zero point	39
6.11.	G33. Threadcutting	41
6.12.	G36. Automatic radius blend	43
6.13.	G37. Tangential approach at the start of machining	45
6.14.	G38. Tangential exit on completion of machining	47
6.15.	G39. Chamfering	49

Section | Page

Section		Page
6.16.	Tool radius compensation	50
6.16.1.	Selection and initiation of tool radius compensation	52
6.16.2.	Operating with tool radius compensation	56
6.16.3.	Cancellation of tool radius compensation	61
6.17.	Tool length compensation	67
6.18.	G47. Single block treatment	
	G48. Cancellation of single block treatment	69
6.19.	G49. Programmable feedrate override	69
6.20.	G50. Loading of the values in the tool offset table	70
6.21.	G52. Communication with the FAGOR Local Area Network	71
6.22.	G53-G59. Zero offsets	73
6.22.1.	G59 As additive zero offset	75
6.23.	G64. Multiple arc pattern machining cycle	76
6.24.	G65. Independent axis execution	79
6.25.	G70/G71. Units of measurement	80
6.26.	G72. Scaling factor	80
6.26.1.	Method a).Scaling factor to affect all axes	80
6.26.2.	Method b).Scaling factor affecting one axis only	82
6.27.	G73. Pattern rotation	84
6.28.	G74. Machine reference search	86
6.29.	Probes	87
6.29.1.	Definition	87
6.29.2.	Characteristics	87
6.29.3.	Most common applications	88
6.29.4.	G75. Probing	89
6.29.5.	G75 N2. Probing canned cycles	90
6.30.	Digitizing with the FAGOR 8025/30 MS CNC	121
6.30.1	Digitizing	121
6.30.2.	Characteristics of digitizing with the FAGOR 8025/30 MS CNC	122
6.30.3.	Preparation of a digitizing operation and later execution at the machine	124
6.30.4.	G76. Automatic block generation	128
6.30.5.	Other digitizing examples	134
6.31.	G77. Slaving the 4th axis (W), 5th (V) axis with its associated axis	
6.31.	G78. cancellation of G77	150
6.32.	Machining canned cycles	151
6.32.1.	Zone of influence of the canned cycle	151
6.32.2.	Cancellation of canned cycles	152
6.32.3.	General considerations	152
6.32.4.	Canned cycle definition G79	153
6.32.5.	(G81, G82, G84, G84R, G85, G86, G89) canned cycle definition	154
6.32.5.1.	G81. Drilling canned cycle	156
6.32.5.2.	G82. Drilling canned cycle with dwell	161
6.32.5.3.	G84. Tapping canned cycle	166
6.32.5.4.	G84 R. Rigid tapping canned cycle	170
6.32.5.5.	G85. Reaming canned cycle	172
6.32.5.6.	G86. Boring canned cycle with G00 withdrawal	172
6.32.5.7.	G89. Boring canned cycle with G01 withdrawal	172
6.32.6	Deep hole drilling canned cycle definition. G83	174
6.32.7.	Pocket milling canned cycle definition (G87,G88)	185
6.32.8.	G87. Rectangular pocket milling canned cycle	190
6.32.9.	G88. Circular pocket milling canned cycle	197
6.33.	G90 G91. Absolute and incremental programming	203
6.34.	G92. Coordinate preset	204
6.35.	G93. Polar origin preset	205
6.36.	G94. Feedrate F in mm/minute (inches/minute)	208
6.37.	G95. Feedrate F in mm/revolution (inches/revolution)	208

Section Page

6.38. G96. Constant Surface Speed 209
6.39. G97. Constant tool center speed 209

7. Coordinate programming 210
7.1. Cartesian coordinates 210
7.1.1. Axis coordinates 210
7.1.2. Center coordinates 212
7.1.3. Rotary axis 213
7.2. Polar Coordinates 215
7.3. Cylindrical coordinates 219
7.4. Two angles (A1,A2) 220
7.5. Angle and one cartesian coordinate 221

8. (F) Feedrate programming 223

9. (S) Spindle speed and spindle orientation 225

10. (T) Tool programming 227
10.1. How to use codes: T2.2 / T2 / T.2 228
10.1.1. Machines without automatic Tool Changer 228
10.1.2. Machines with automatic Tool Changer 229

11. (M) Miscellaneous functions 230
11.1. M00. Program stop 230
11.2. M01. Conditional stop of program 231
11.3. M02. End of program 231
11.4. M30. End of program with return to beginning 231
11.5. M03. Clockwise start of the spindle 231
11.6. M04. Counter-clockwise start of the spindle 231
11.7. M05. Spindle stop 231
11.8. M06. Tool change code 232
11.9. M19. Residual analog S output (creep) for tool change and spindle orientation 233
11.10. M22, M23, M24, M25. Operation with pallets 234

12. Standard and parametric subroutines 236
12.1. Identification of a standard subroutine 237
12.2. Calling in a standard subroutine 238
12.3. Identification of a parametric subroutine 238
12.4. Calling in a parametric subroutine 239
12.5. Nesting levels 246
12.6. Emergency subroutine 246

13. Parametric programming. Operations with parameters 247

ERROR CODES

COMPARISON TABLE
FOR MILL MODEL
FAGOR 8025/8030 CNCs

8025/8030 MILL MODEL CNCS

Fagor offers the 8025 and 8030 mill type CNCs.

Both types operate the same way and offer similar characteristics. Their basic difference is that the former is compact and the latter is modular.

Both CNC types offer basic models. Although the differences between the basic models are detailed later on, each model may be defined as follows:

8025/8030 GP	Oriented to General Purpose machines
8025/8030 M	Oriented to Milling machines of up to 4 axes.
8025/8030 MG	Same as the M model, but with dynamic graphics.
8025/8030 MS	Oriented to Machining Centers (up to 5 axes).

When the CNC has an Integrated Programmable Logic Controller (PLCI), the letter "I" is added to the CNC model denomination: GPI, MI, MGI, MSI.

Also, When the CNC has 512Kb of part-program memory, the letter "K" is added to the CNC model denomination: GPK, MK, MGK, MSK, GPIK, MIK, MGIK, MSIK.

	Basic	**With PLCI**	**Basic With 512Kb**	**With PLCI and 512Kb**
General Purpose	**GP**	**GPI**	**GPK**	**GPKI**
Mills up to 4 axes	**M**	**MI**	**MK**	**MIK**
Up to 4 axes with graphics	**MG**	**MGI**	**MGK**	**MGIK**
Machining Centers	**MS**	**MSI**	**MSK**	**MSIK**

TECHNICAL DESCRIPTION

	GP	M	MG	MS
INPUTS/OUTPUTS				
Feedback inputs.	**6**	**6**	**6**	**6**
Linear axes	**4**	**4**	**4**	**5**
Rotary axes	**2**	**2**	**2**	**2**
Spindle encoder	**1**	**1**	**1**	**1**
Electronic handwheels	**1**	**1**	**1**	**1**
Probe input	**x**	**x**	**x**	**x**
Square-wave feedback signal multiplying factor, x2/x4	**x**	**x**	**x**	**x**
Sine-wave feedback signal multiplying factor, x2/x4/10/x20	**x**	**x**	**x**	**x**
Maximum counting resolution 0.001mm/0.001°/0.0001inch	**x**	**x**	**x**	**x**
Analog outputs (±10V) for axis servo drives	**4**	**4**	**4**	**5**
Spindle analog output (±10V)	**1**	**1**	**1**	**1**
AXIS CONTROL				
Axes involved in linear interpolations	**3**	**3**	**3**	**3**
Axes involved in circular interpolations	**2**	**2**	**2**	**2**
Helical interpolation	**x**	**x**	**x**	**x**
Electronic threading	**x**	**x**	**x**	
Spindle control	**x**	**x**	**x**	**x**
Software travel limits	**x**	**x**	**x**	**x**
Spindle orientation	**x**	**x**	**x**	**x**
Management of non-servo-controlled Open-Loop motor	x			
PROGRAMMING				
Part Zero preset by user	**x**	**x**	**x**	**x**
Absolute/incremental programming	**x**	**x**	**x**	**x**
Programming in cartesian coordinates	**x**	**x**	**x**	**x**
Programming in polar coordinates	**x**	**x**	**x**	**x**
Programming in cylindrical coordinates (radius, angle, axis)	**x**	**x**	**x**	**x**
Programming by angle and cartesian coordinate	**x**	**x**	**x**	**x**
COMPENSATION				
Tool radius compensation		**x**	**x**	**x**
Tool length compensation	**x**	**x**	**x**	**x**
Leadscrew backlash compensation	**x**	**x**	**x**	**x**
Leadscrew error compensation	**x**	**x**	**x**	**x**
Cross compensation (beam sag)	**x**	**x**	**x**	**x**
DISPLAY				
CNC text in Spanish, English, French, German and Italian	**x**	**x**	**x**	**x**
Display of execution time	**x**	**x**	**x**	**x**
Piece counter	**x**	**x**	**x**	**x**
Graphic movement display and part simulation			**x**	**x**
Tool base position display	**x**	**x**	**x**	**x**
Tool tip position display	**x**	**x**	**x**	**x**
Geometric programming aide	**x**	**x**	**x**	**x**
COMMUNICATION WITH OTHER DEVICES				
Communication vía RS232C	**x**	**x**	**x**	**x**
Communication via DNC	**x**	**x**	**x**	**x**
Communication via RS485 (FAGOR LAN)	**x**	**x**	**x**	**x**
ISO program loading from peripherals	**x**	**x**	**x**	**x**
OTHERS				
Parametric programming	**x**	**x**	**x**	**x**
Model digitizing	**x**	**x**	**x**	**x**
Possibility of an integrated PLC	**x**	**x**	**x**	**x**
Sheetmetal tracing on LASER machines				**x**
Jig Grinder				**x**

PREPARATORY FUNCTIONS

Function		GP	M	MG	MS
AXES AND COORDINATE SYSTEMS					
XY (G17) plane selection		x	x	x	x
XZ and YZ plane selection (G18,G19)		x	x	x	x
Part measuring units. Millimeters or inches (G70,G71)		x	x	x	x
Absolute/incremental programming (G90,G91)		x	x	x	x
Independent axis (G65)		x	x	x	x
REFERENCE SYSTEMS					
Machine reference (home) search (G74)		x	x	x	x
Coordinate preset (G92)		x	x	x	x
Zero offsets (G53...G59)		x	x	x	x
Polar origin preset (G93)		x	x	x	x
Store current part zero (G31)		x	x	x	x
Recover stored part zero (G32)		x	x	x	x
PREPARATORY FUNCTIONS					
Feedrate F		x	x	x	x
Feedrate in mm/min. or inches/minute (G94)		x	x	x	x
Feedrate in mm/revolution or inches/revolution (G95)		x	x	x	x
Constant surface speed (G96)		x	x	x	x
Constant tool center speed (G97)		x	x	x	x
Programmable feedrate override (G49)		x	x	x	x
Spindle speed (S)		x	x	x	x
S value limit (G92)		x	x	x	x
Tool and tool offset selection (T)		x	x	x	x
AUXILIARY FUNCTIONS					
Program stop (M00)		x	x	x	x
Conditional program stop (M01)		x	x	x	x
End of program (M02)		x	x	x	x
End of program with return to first block (M30)		x	x	x	x
Clockwise spindle start (M03)		x	x	x	x
Counter-clockwise spindle start (M04)		x	x	x	x
Spindle stop (M05)		x	x	x	x
Tool change in machining centers (M06)		x	x	x	x
Spindle orientation (M19)		x	x	x	x
Spindle speed range change (M41, M42, M43, M44)		x	x	x	x
Functions associated with pallets (M22, M23, M24, M25)		x	x	x	
PATH CONTROL					
Rapid traverse (G00)	x	x	x	x	
Linear interpolation (G01)	x	x	x	x	
Circular interpolation (G02,G03)	x	x	x	x	
Circular interpolation with absolute center coordinates (G06)	x	x	x	x	
Circular path tangent to previous path (G08)	x	x	x	x	
Arc defined by three points (G09)	x	x	x	x	
Tangential entry at beginning of a machining operation (G37)	x	x	x	x	
Tangential exit at the end of a machining operation (G38)	x	x	x	x	
Controlled radius blend (G36)	x	x	x	x	
Chamfer (G39)	x	x	x	x	
Electronic threading (G33)		x	x	x	
ADDITIONAL PREPARATORY FUNCTIONS					
Dwell (G04 K)	x	x	x	x	
Round and square corner (G05, G07)	x	x	x	x	
Mirror image (G10,G11,G12)	x	x	x	x	
Mirror image along the Z axis (G13)	x	x	x	x	
Scaling factor (G72)	x	x	x	x	
Pattern rotation (G73)	x	x	x	x	
Slaving/unslaving of axes (G77, G78)	x	x	x	x	
Single block treatment (G47, G48)	x	x	x	x	
User error display (G30)	x	x	x	x	
Automatic block generation (G76)				x	
Communication with FAGOR Local Area Network (G52)	x	x	x	x	

	GP	M	MG	MS
COMPENSATION				
Tool radius compensation (G40,G41,G42)		x	x	x
Tool length compensation (G43,G44)	x	x	x	x
Loading of tool dimensions into internal tool table (G50)	x	x	x	x
CANNED CYCLES				
Multiple arc-pattern machining (G64)		x	x	x
User defined canned cycle (G79)	x	x	x	x
Drilling cycle (G81)		x	x	x
Drilling cycle with dwell (G82)		x	x	x
Deep hole drilling cycle (G83)		x	x	x
Tapping cycle (G84)		x	x	x
Rigid tapping cycle (G84R)		x	x	x
Reaming cycle (G85)		x	x	x
Boring cycle with withdrawal in G00 (G86)		x	x	x
Rectangular pocket milling cycle (G87)		x	x	x
Circular pocket milling cycle (G88)		x	x	x
Boring cycle with withdrawal in G01 (G89)		x	x	x
Canned cycle cancellation (G80)	x	x	x	x
Return to starting point (G98)		x	x	x
Return to reference plane (G99)		x	x	x
PROBING				
Probing (G75)	x	x	x	x
Tool length calibration canned cycle (G75N0)				x
Probe calibration canned cycle (G75N1)				x
Surface measuring canned cycle (G75N2)				x
Surface measuring canned cycle with tool offset (G75N3)				x
Outside edge measuring canned cycle (G75N4)				x
Inside edge measuring canned cycle (G75N5)				x
Angle measuring canned cycle (G75N6)				x
Outside edge and angle measuring canned cycle (G75N7)				x
Hole centering canned cycle (G75N8)				x
Boss centering canned cycle (G75N9)				x
Hole measuring canned cycle (G75N10)				x
Boss measuring canned cycle (G75N11)				x
SUBROUTINES				
Number of standard subroutines	99	99	99	99
Definition of standard subroutine (G22)	x	x	x	x
Call to a standard subroutine (G20)	x	x	x	x
Number of parametric subroutines	99	99	99	99
Definition of parametric subroutine (G23)	x	x	x	x
Call to a parametric subroutine (G21)	x	x	x	x
End of standard or parametric subroutine (G24)	x	x	x	x
JUMP OR CALL FUNCTIONS				
Unconditional jump/call (G25)	x	x	x	x
Jump or call if zero (G26)	x	x	x	x
Jump or call if not zero (G27)	x	x	x	x
Jump or call if smaller (G28)	x	x	x	x
Jump or call if equal or greater (G29)	x	x	x	x

NEW FEATURES AND MODIFICATIONS

Date: **February 1991** **Software version:** **2.1 and newer**

FEATURE	**MODIFIED MANUAL AND SECTION**	
Error 65 is not issued while probing (G75)	Installation Manual	Section 3.3.4
It is possible to select the home searching direction for each axis	Installation Manual	Section 4.6
New 1, 2, 5, 10 resolution values for sine-wave feedback signals of each axis	Installation Manual	Section 4.1
PLCI register access from the CNC	Programming Manual	G52
Sheetmetal tracing on laser machines	Applications Manual	
Jig Grinder	Applications Manual	

Date: **June 1991** **Software version:** **3.1 and newer**

FEATURE	**MODIFIED MANUAL AND SECTION**	
Repetitive emergency subroutine	Installation Manual	Section 3.3.8
New function F29. It takes the value of the selected tool	Programming Manual	Chapter 13
Function M06 does not execute M19	Installation Manual	Section 3.3.5
Greater speed when executing several parametric blocks in a row.		

Date: March 1992 **Software version: 4.1 and newer**

FEATURE	MODIFIED MANUAL AND SECTION	
Bell-shape acceleration/deceleration control	Installation Manual	Section 4.7
Expansion of cross compensation	Installation Manual	Section 4.10
Rigid Tapping G84 R	Programming Manual	G84
Possibility to enter the sign of the leadscrew backlash for each axis	Installation Manual	Section 4.9
Independent execution of an axis	Programming Manual	G65

Date: July 1993 **Software version: 5.1 and newer**

FEATURE	MODIFIED MANUAL AND SECTION	
Double cross compensation	Installation Manual	Section 4.10
Linear and bell-shaped acc./dec. ramp combination for the axes	Installation Manual	Section 4.7
Acceleration/deceleration control for the the spindle	Installation Manual	Section 5.
Multiple arc pattern machining	Programming Manual	G64
Tool tip position display	Installation Manual	Section 3.3.5
The associated subroutine is executed before the T function	Installation Manual	Section 3.3.5
The additional circular sections of a compensated path are executed in G05 or G07	Installation Manual	Section 3.3.8
VGA monitor 8030 CNC.	Installation Manual	Chapter 1

Date: March 1995 **Software version: 5.3 and newer**

FEATURE	**MODIFIED MANUAL AND SECTION**	
Management of feedback with coded Io	Installation Manual	Section 4.6 & 6.5
Spindle inhibit by PLC	Installation Manual	Section 3.3.9
Handwheel management by PLC	Installation Manual	Section 3.3.3
Rapid (JOG) key simulation via PLC	PLCI Manual	
Non-servo-controlled open-loop motors	Applications Manual	
Function G64, multiple machining in an arc. To be selected by machine parameter.	Installation Manual	Section 3.3.9
Initialization of machine parameters after memory loss.		

Date: September 1995 **Software version: 6.0 and newer**

FEATURE	**MODIFIED MANUAL AND SECTION**	
512 Kb of part-program memory	Operating Manual	Section 3.6
When conditional input (block skip) active while in JOG mode, the key is ignored	Installation Manual	Section 1.3.6

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 not 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)

Protections of the unit itself

Central Unit

It carries two fast fuses of 3.15 Amp./ 250V. to protect the mains AC input.

All the digital inputs and outputs are protected by an external fast fuse (F) of 3.15 Amp./ 250V. against over voltage and reverse connection of the power supply.

Monitor

The type of fuse depends on the type of monitor. See the identification label of the unit.

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.

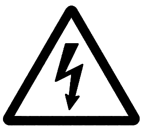
Safety symbols

Symbols which may appear on the manual

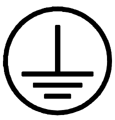
WARNING. symbol
It has an associated text indicating those actions or operations may hurt people or damage products.

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 CNC, 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 8025/30 M CNC

8025M CNC OEM Manual Is directed to the machine builder or person in charge of installing and starting up the CNC.

It contains 2 manuals:

Installation Manual	describing how to isntall and set-up the CNC.
LAN Manual	describing how to instal the CNC in the Local Area Network.

Sometimes, it may contain an additional manual describing New Software Features recently implemented.

8025M CNC USER Manual Is directed to the end user or CNC operator.

It contains 3 manuals:

Operating Manual	describing how to operate the CNC.
Programming Manual	describing how to program the CNC.
Applications Manual	describing other applications for this CNC non-specific of Milling machines

Sometimes, it may contain an additional manual describing New Software Features recently implemented.

DNC 25/30 Software Manual Is directed to people using the optional DNC communications software.

DNC 25/30 Protocol Manual Is directed to people wishing to design their own DNC communications software to communicate with the 800 without using the DNC25/30 software..

PLCI Manual To be used when the CNC has an integrated PLC.

Is directed to the machine builder or person in charge of installing and starting up the PLCI.

DNC-PLC Manual Is directed to people using the optional communications software: DNC-PLC.

FLOPPY DISK Manual Is directed to people using the Fagor Floppy Disk Unit and it shows how to use it.

MANUAL CONTENTS

The Programming manual consists of the following chapters:

Index.

Comparison table of FAGOR models: 8025 M CNCs

New Features and modifications.

Introduction
- Summary of safety conditions.
- Material returning conditions.
- FAGOR documentation for the 8025 M CNC.
- Manual contents

Overview

Writing a program

Program format

Program blocks

Preparatory functions

Coordinate programming

Feedrate programming

Spindle speed and orientation

Tool programming

Auxiliary functions

Subroutines

Parametric programming

Machining canned cycles

Error codes

1. OVERVIEW

The CNC can be programmed both from its front panel and from external peripherals (tape reader, cassette reader/recorder, computer etc.). The memory capacity for part programming is **32K characters** In this CNC the part programs can be entered in four different operating modes:

2 - PLAY BACK
3 - TEACH IN
6 - EDITING
7 - INPUT-OUTPUT

In mode 7, the programs are transferred to the CNC from any external peripheral (RS 232 C). In the other modes, the programs are entered directly from the front panel of the CNC. This means that the programming can be carried out both at the machine and at a remote location, e.g. in a programming office.

In the **PLAY BACK** mode, the axes are shifted manually (Jog) and the coordinates reached are then entered as the program coordinates.

In the **TEACH IN** mode, a block is written and executed and then entered as part of the program.

In the **EDITING** mode, the complete program is recorded and then executed.

1.1. EXTERNAL PROGRAMMING

If the programming is to be carried out by means of an external peripheral, ISO code must be used. % will initiate the program, followed by the program number (five digits, followed by the characters, **RT** or **LF** and the **N** of the first block). **R**e**T**urn or LineFeed must be used after each block prior to the N of the beginning of the following block.

To end the program the characters **ESC**ape (ESC) or **E**nd **O**f **T**ape (EOT) or a series of 20 nul characters (ASCII 00) must be used.

1.2. TEXT PROGRAMMING

Comments to be displayed on the CRT must be written between parenthesis **( )**.

(**43** characters maximum, parenthesis included).

The comment must be written at the end of the block, that is:

N4 G.. X.. F.. M.. (comment).

If the first character in parenthesis is an asterisk (* Comment) the comment will blink on the screen.

An **EMPTY** comment ( ) cancels the display of the previous one.

1.3. DNC CONNECTION

Every CNC offers as a standard feature, the possibility of working with DNC (Distributed numerical control), enabling the communication between the CNC and a computer to carry out the following functions:

. Directory and deletion commands

. Program and table transfers between the CNC and acomputer

. Execution of infinite program

. Machine remote control

. Ability to supervise the status of the advanced DNC systems

1.4. THE FAGORDNC

Communication program Commercialized in a 5.25" or 3.5" flexible diskette is an application for the connection of FAGOR numerical controls to a **PC COMPATIBLE** computer with FAGOR Numerical Controls, using the DNC incorporated in those controls.

Several CNC can be connected to the DNC through the RS 232 lines of these computers.

The operation mode is interactive, with **MENUS** which guide the user and simplify the use of this program.

The computer is used as a part-programs centralized **STORAGE**, avoiding the use of awkward puncher tapes. This simplifies the version upgrading, allows to make safety copies, listing and edition of part programs with inclusion of comments .....

The manual of DNC connection and the **FAGORDNC** communication can be requested at this address.

2. CREATING A PROGRAM

The machining program must be entered in a form acceptable to the CNC. It must include all the geometrical and technological data required for the machine-tool to perform the required functions and movements.

A program is built up in the form of a sequence of blocks.

Each programming block consists of:

N	Block No.
G	Preparatory functions
V,**W**,**X**,**Y**,**Z**	Coordinate values
F	Feedrate
S	Spindle speed
T	Tool No.
M	Miscellaneous functions

This order has to be maintained within each block, although each block does not necessarily contain all of these items.

3. PROGRAM FORMAT

The CNC can be programmed in millimeters or in inches.

Metric format (in mm):

P(%)5 N4 G2 V +/-4.3 W+/-4.3 X+/-4.3 Y+/-4.3 Z+/- 4.3 F5.5 S4 T2.2 M2

Format in inches:

P(%)5 N4 G2 V+/-4.3 W+/-3.4 X+/-3.4 Y+/-3.4 Z+/-3.4 F5.5 S4 T2.2 M2

+/- 4.3 Means that a positive or negative figure with up to four digits to the left of the decimal point and three to the right may be programmed.

+/- 3.4 Means that a positive or negative figure with up to four digits to the left of the decimal point and three to the right may be programmed.

4 Means that only a positive integer (no decimals) of up to four digits may be programmed.

2.2 Means that only a positive value of up to two digits to the left and two to the right of the decimal point may be programmed.

The CNC can control up to **5** axes (V,W,X,Y,Z) depending on the type of machine used.

Programming in the same block of the **5th axis V**, of the **4th axis W** and the one associated with both, which is indicated in the machine parameter P11, is incompatible.

The **4th axis W** can be replaced with the **5th axis V** in the different programming formats which are indicated in the manual.

Throughout this manual the format corresponding to each function will be enumerated, as well as the meaning of the different parameters used.

3.1. PARAMETRIC PROGRAMMING

It is also possible to program in a block any function by parameters, except the program number, the block number, **G** functions, in the same block of another piece of data, such as: G4K..;G22N..;G25N.. etc in such a way that , when executing the block, the function takes the current value of the parameter.

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

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

The CNC has **255** arithmetic parameters (P00/P254). (See chapter 13 of this manual, Parametric Programming,).

4. PROGRAM NUMBERING

Every program must be numbered between **0** and **99998**.

This number must be entered at the beginning of the program, before the first block.

If the program is entered from an external peripheral, the symbol **%** is used, followed by the number required and the pressing of **LF** or **RT** or both followed by the **N** of the first block.

5. PROGRAM BLOCKS

5.1. BLOCK NUMBERING

The block number is used to identify each of the blocks that make up a program.

The block number consists of the letter **N** followed by a figure between 0 and 9999.

This number must be written at the start of each block.

Blocks may be given any number between **0** and **9999** except that no block may be given a lower number than the blocks preceding it in the program.

It is advisable to avoid giving blocks consecutive numbers, so that new blocks can be interposed where required.

If the CNC is programmed from its front panel, blocks are automatically numbered in steps of **10**. The automatic numbering can be manually altered.

5.2. CONDITIONAL BLOCKS

There are two types of conditional blocks:

a) N4 Standard conditional block

If next to the block number **N4 (0-9999)**, a decimal point **(.)** is written, the block is characterized as a normal conditional block. That means that the CNC will execute it, only if the relevant external signal (enabling input for conditional blocks) is activated.

During many program execution, the CNC reads four blocks ahead the one being executed, so the external signal is to be activated, at least during the execution of the fifth block previous to the conditional block, for its execution to be carried out.

b) M4 Special conditional block

If next to the block number **N4**, two decimal points (..) are written, the block is characterized as a special conditional block, in other words, the CNC will execute it, only if the relevant external signal (enabling input for conditional blocks) is activated.

In this case, it is enough to activate the external signal (conditional input), during the execution of the block previous to the special conditional block, for its execution to be carried out.

The **N4..** special conditional block, cancels **G41** or **G42** tool radius compensation.

6. PREPARATORY FUNCTIONS

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

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

6.1. TABLE OF G FUNCTIONS USED AT THE CNC

(Modal)	G00*	: Positioning (Modal)
	G01	: Linear interpolation
(Modal)	G02	: Clockwise circular helical interpolation
(Modal)	G03	: Counter-clockwise circular helical interpolation
	G04	: Dwell, duration programmed by means of **K**
(Modal)	G05	: Round corner
	G06	: Circular Interpolation with absolute center coordinates
(Modal)	G07*	: Square corner
	G08	: Arc tangent to previous path
	G09	: Arc programmed by three points
(Modal)	G10*	: Cancellation of mirror image
(Modal)	G11	: Mirror image on the **X** axis
(Modal)	G12	: Mirror image on the **Y** axis
(Modal)	G13	: Mirror image on the **Z** axis
(Modal)	G17*	: Selection of the **XY** plane
(Modal)	G18	: Selection of the **XZ** plane
(Modal)	G19	: Selection of the **YZ** plane
	G20	: Call for standard subroutine
	G21	: Call for parametric subroutine
	G22	: Definition of standard subroutine
	G23	: Definition of parametric subroutine
	G24	: End of subroutine
	G25	: Unconditional jump/call
	G26	: Conditional jump/call if zero
	G27	: Conditional jump/call if different from zero
	G28	: Conditional jump/call if smaller than zero
	G29	: Conditional jump/call if equal to or greater than zero
	G30	: Display error code defined by **K**
	G31	: Store present program’s datum point
	G32	: Retrieve datum point stored by **G31**
Modal)	G33	: Threadcutting

G36 : Automatic radius blend
G37 : Tangential approach
G38 : Tangential exit
G39 : Chamfering
(Modal) G40* : Cancellation of radius compensation
(Modal) G41 : Left hand radius compensation
(Modal) G42 : Right hand radius compensation
(Modal) G43 : Length compensation
(Modal) G44* : Cancellation of length compensation
(Modal) G47 : Single block treatment
(Modal) G48* : Cancellation of single block treatment
G50 : Loading of the values in the tool offset table
G52 : Communication with **FAGOR LOCAL AREA NETWORK**
(Modal) G53-G59 : Zero offsets
G64 : Multiple arc pattern machining cycle
G65 : Independent axis execution
(Modal) G70 : Programming in inches
(Modal) G71 : Programming in millimeters
(Modal) G72 : Scaling factor
(Modal) G73 : Pattern rotation
G74 : Automatic search for machine reference
G75 : Probing
G75 N2: Probing canned cycles
G76 : Automatic block generation
(Modal) G77 : Coupling of **4th axis W** or **5th axis V** with associated axis
(Modal) G78* : Cancellation of **G77**.
(Modal) G79 : User defined canned cycle
(Modal) G80* : Cancellation of canned cycles
(Modal) G81 : Drilling canned cycle
(Modal) G82 : Drilling canned cycle with dwell
(Modal) G83 : Deep drilling canned cycle
(Modal) G84 : Tapping canned cycle
(Modal) G85 : Reaming canned cycle
(Modal) G86 : Boring canned cycle with **G00** withdrawal
(Modal) G87 : Rectangular pocket canned cycle
(Modal) G88 : Circular pocket canned cycle
(Modal) G89 : Boring canned cycle with **G01** withdrawal
(Modal) G90* : Programming of absolute coordinates
(Modal) G91 : Programming of incremental coordinates
G92 : Preselection of coordinates
G93 : Preselection of polar origin
(Modal) G94* : Feedrate **F** in mm/min. (inches/min.)
(Modal) G95 : Feedrate **F** in mm/rev. (inches/rev.)
(Modal) G96 : Constant surface feed
(Modal) G97* : Constant surface speed at the center of the tool
(Modal) G98* : Tool return to starting plane on completing a canned cycle
(Modal) G99 : Tool return to reference (approach) plane on completing a canned cycle

Functions G75 N2 and G76 are only available on the model 8025/30 MS model.

Modal means that once the **G** functions have been programmed they remain active until cancelled by another **G** which is incompatible or by **M02**,**M30**,**EMERGENCY** or **RESET**.

The **G** functions marked * are those which the CNC assumes on being turned on or after executing **M02** or **M30** or after an **EMERGENCY** or **RESET**. Whether **G05** or G07 is assumed will depend on the value assigned to P613(5).

All the **G**'s required may be programmed in any order in the same block, except **G20**,**G21**,**G22**,**G23**,**G24**,**G25**,**G26**,**G27**,**G28**,**G29**,**G30**,**G31**,**G32**,**G50**,**G52**,**G53**,**G59**,**G72**,**G73**,**G74** and G92 which have to be alone in a block.

If incompatible **G** functions are programmed in the same block, the CNC assumes the one programmed last.

6.2. TYPES OF MOVEMENT

6.2.1. G00. Positioning

The movements programmed following **G00** are executed at rapid feedrate set during the final adjustment of the machine by means of the machine-parameters.

There are two different ways of movement in **G00**, depending on the value applied to P610(2) machine- parameter.

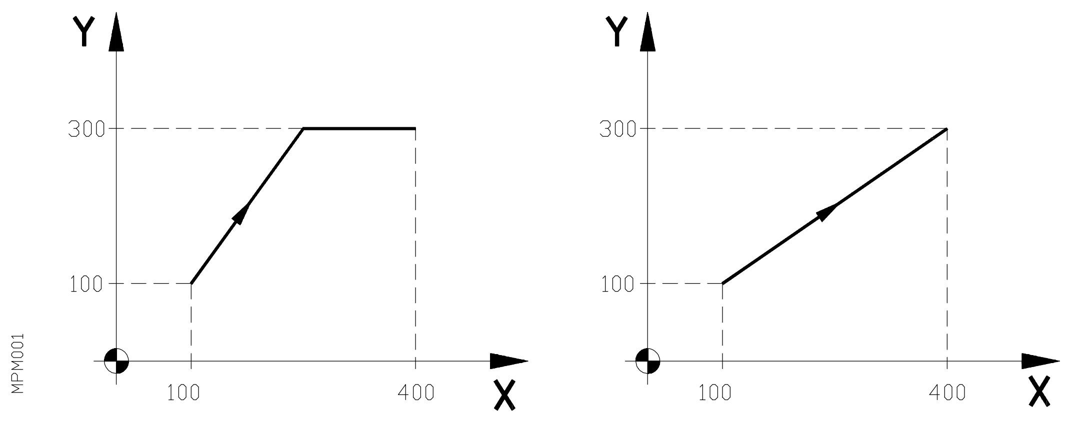
a) Path not controlled. P610(2)=0

The rapid feedrate value is independent for each axis, thus, the path is not controlled when more than one axis move at the same time.

b) Vectored G00. P610(2)=1

In this case the resultant path is always a straight line between the initial and the final point, no matter the number of axes that are moving.

a) P610(2)=0 b) P610(2)=1



Initial point X100 Y100
N4 G00 G90 X400 Y300

In **G00** movements, **P4** machine-parameter can be used to identify whether the feedrate override knob operates between **0%** and **100%** or is frozen at **100%**.

When the CNC is turned on, after executing **M02/M30** or after an **EMERGENCY** or **RESET**, the CNC takes the code **G00** on. The code **G00** is modal and incompatible with **G01**,**G02**,**G03** and **G33**.

G00 function can be programmed with **G**, **G0** or **G00**.

When programming **G00** function, the last **F** programmed is not cancelled, that means that when **G01**,**G02** or G03 is programmed again, the mentioned **F** is recovered.

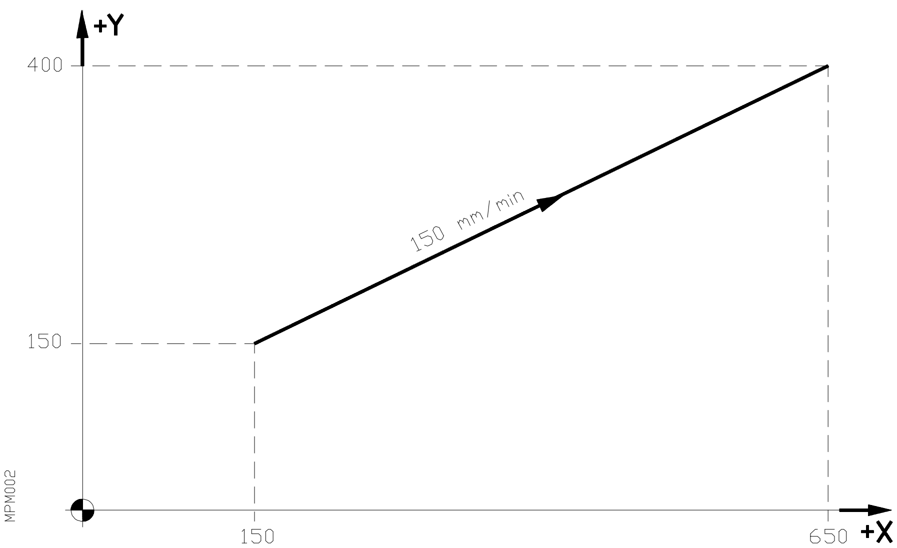
6.2.2. G01. Linear interpolation

The movements programmed after **G01** are performed in a straight line at the feedrate **F** programmed.

When two or three axes move simultaneously, the resulting path is a straight line between the initial point and the final point.

The machine moves along that path at the programmed feedrate **F**. The CNC calculates the feedrates of each axis so that the feedrate of the resulting path is the programmed **F**.

Example:



G01 G90 X650 Y400 F150

The knob on the front panel of the CNC (M.F.O.) can be used to vary the programmed feedrate **F** between 0% and 120% or between 0% and 100%, according to parameter P606(2).

If, during a **G01** movement, the **RAPID FEED** key is pressed, the movement will be performed at twice the programmed feedrate if P606(2) is zero. The same thing will happen when the external **START** input is activated if P609(7) is one. Function **G01** is modal and incompatible with G00,G02,G03 and G33. Function G01 can be programmed as **G1**.

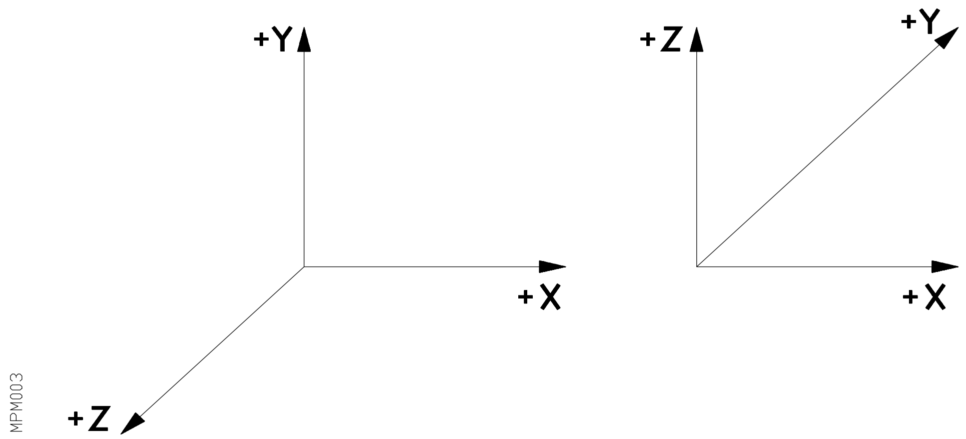
6.2.3. G02/G03. Circular helical interpolation

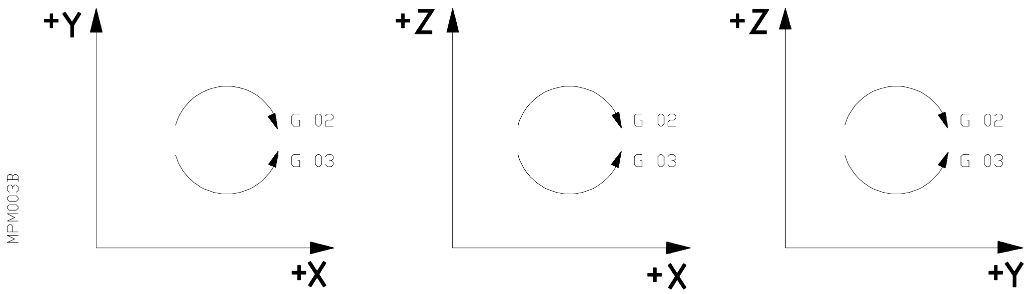
G02: Clockwise circular helical interpolation.
G03: Counter-clockwise circular helical interpolation.

6.2.3.1. Circular interpolation

The movements programmed following G02/G03 are performed in a circular path at the programmed feedrate **F**.

The definitions of clockwise (G02) and counter-clockwise (G03) have been fixed according to the system of coordinates depicted below (right-hand or dextrogyratory system).





This system of coordinates is referred to the movement of the tool over the part.

Atention:

The direction of G02 and G03 on the XZ plane can be changed by means of parameter P605(4).

If the system of left-hand coordinates is used, the directions of G02 and G03 are reversed.

Circular interpolation can only be carried out in the plane. The method of defining circular interpolation is as follows:

Cartesian coordinates

XY plane

G17 G02 (G03) X+/-4.3 Y+/-4.3 I+/-4.3 J+/-4.3 F5.4

XZ plane

G18 G02 (G03) X+/-4.3 Z+/-4.3 I+/-4.3 K+/-4.3 F5.4

YZ plane

G19 G02 (G03) Y+/-4.3 Z+/-4.3 J+/-4.3 K+/-4.3 F5.4

In the case of four-axis machines:

a) If the fourth axis (W) is incompatible with the X axis.

WY plane

G17 G02 (G03) W+/-4.3 Y+/-4.3 I+/-4.3 J+/-4.3 F5.4

WZ plane

G18 G02 (G03) W+/-4.3 Z+/-4.3 I+/-4.3 K+/-4.3 F5.4

b) If the fourth axis (W) is incompatible with the Y axis.

WX plane

G17 G02 (G03) W+/-4.3 X+/-4.3 I+/-4.3 J+/-4.3 F5.4

WZ plane

G19 G02 (G03) W+/-4.3 Z+/-4.3 J+/-4.3 K+/-4.3 F5.4

c) If the fourth axis is incompatible with the Z axis.

WX plane

G18 G02 (G03) W+/-4.3 X+/-4.3 I+/-4.3 K+/-4.3 F5.4

WY plane

G19 G02 (G03) W+/-4.3 Y+/-4.3 J+/-4.3 K+/-4.3 F5.4

Polar coordinates

XY plane

G17 G02 (G03) A+/-3.3 I+/-4.3 J+/-4.3 F5.4

XZ plane

G18 G02 (G03) A+/-3.3 I+/-4.3 K+/-4.3 F5.4

YZ plane

G19 G02 (G03) A+/-3.3 J+/-4.3 K+/-4.3 F5.4

In the case of four-axis machines:

a) If the fourth axis (W) is incompatible with the X axis.

WY plane

G17 G02 (G03) A+/-3.3 I+/-4.3 J+/-4.3 F5.4

WZ plane

G18 G02 (G03) A+/-3.2 I+/-4.3 K+/-4.3 F5.4

b) If the fourth axis (W) is incompatible with the Y axis.

WX plane

G17 G02 (G03) A+/-3.3 I+/-4.3 J+/-4.3 F5.4

WZ plane

G19 G02 (G03) A+/-3.3 J+/-4.3 K+/-4.3 F5.4

c) If the fourth axis is incompatible with the Z axis.

WX plane

G18 G02 (G03) A+/-3.3 I+/-4.3 K+/-4.3 F5.4

WY plane

G19 G02 (G03) A+/-3.3 J+/-4.3 K+/-4.3 F5.4

The **fourth axis (W)** must be linear and therefore P600(1)(2) and (3) must be zero.

Atention:

In **5-axis** machines, programming of the **5th axis V** is equivalent to that described for the **4th axis W**.

Functions G17,G18,G19 define the XY,XZ,YZ interpolation planes.

These functions are modal and incompatible with one another, i.e. once programmed they remain active until another one is programmed.

In the case of four (five)-axis machines:

a. If W (V) is incompatible with X

G17 defines the XY or WY (VY) planes.
G18 defines the XZ or WZ (VZ) planes.

b. If W (V) is incompatible with Y

G17 defines the XY or XW (VX) planes.
G19 defines the YZ or WZ (VZ) planes.

c. If W (V) is incompatible with Z

G18 defines the XZ or XW (VX) planes.
G19 defines the YZ or YW (VY) planes.

Once any of the codes G17,G18,G19 has been programmed, the CNC will move the axes programmed thereafter.

I,J,K define the arc’s center.

I: distance from the starting point to the arc’s center, along X(W)(V) axis.

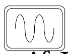
J: distance from the starting point to the arc’s center, along Y(W)(V) axis.

K: distance from the starting point to the arc’s center, along Z(W)(V) axis.

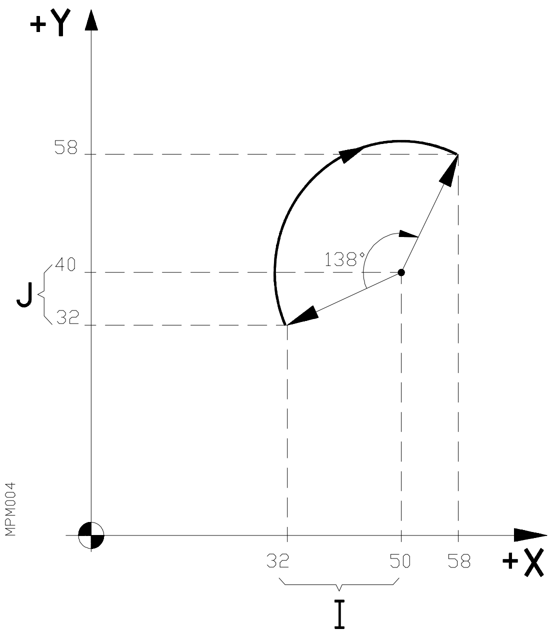
I,J,K must be programmed with sign. They must always be programmed, even when their value is zero.

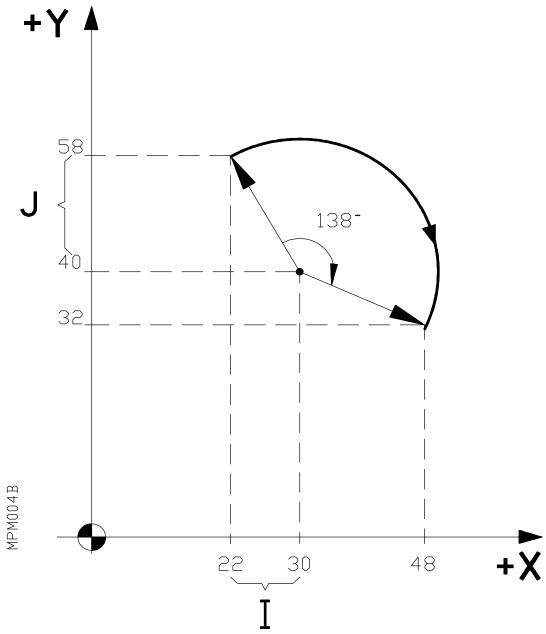
The CNC takes the arc’s center as the new polar origin when carrying out a G02,G03 circular interpolation.

The knob on the front panel of the CNC (M.F.O.) can be used to vary the programmed feedrate **F** between 0% and 120% or between 0% and 100%, according to parameter P606(2).

If during a G02/G03 movement a **Rapid Feed**  key is pressed, the movement will be performed at twice the programmed feedrate if P606(2) is zero. The same thing will happen when the external **START** input is activated if P609(7) is one.

Example:





CARTESIAN COORDINATES

G17 G02 G91 X26 Y26 I18 J8 G17 G02 G91 X26 Y-26 I8 J-18

POLAR COORDINATES

G17 G02 G91 A-138 I18 J8 G17 G02 G91 A- 138 I8 J-18

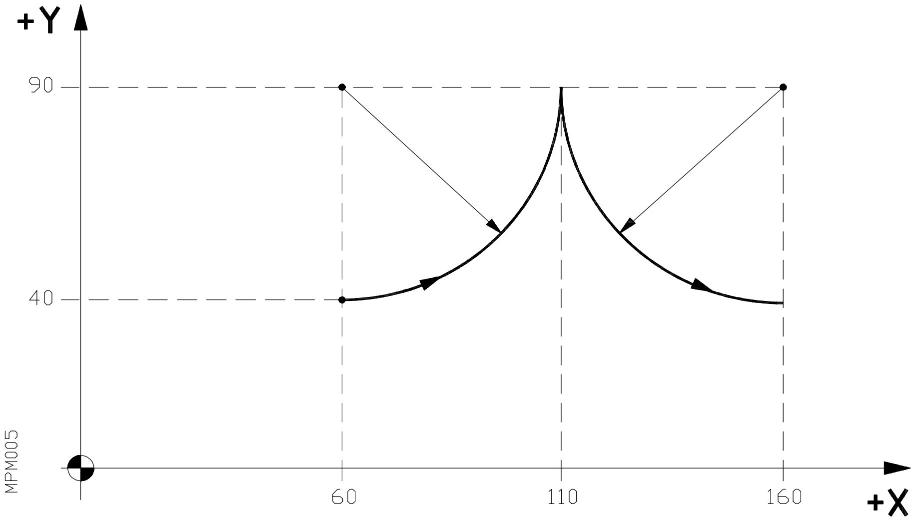
Any arc of up to a value of 360º can be programmed.

Functions G02/G03 are modal and incompatible both with one another and with G00,G01 and G33.

Functions G74,G75,M06 (machining centers) and M22,M23,M24,M25 (machines with pallets) cancel G02/G03 functions.

Functions G02/G03 can be programmed as G2/G3.

Example:



Cartesian coordinate values:

N5 G90 G17 G03 X110 Y90 I0 J50 F150
N10 X160 Y40 I50 J0

Polar coordinate values:

N5 G90 G17 G03 A0 I0 J50 F150
N10 A-90 I50 J0

or,

N5 G91 G17 G03 A90 I0 J50 F150
N10 A90 I50 J0

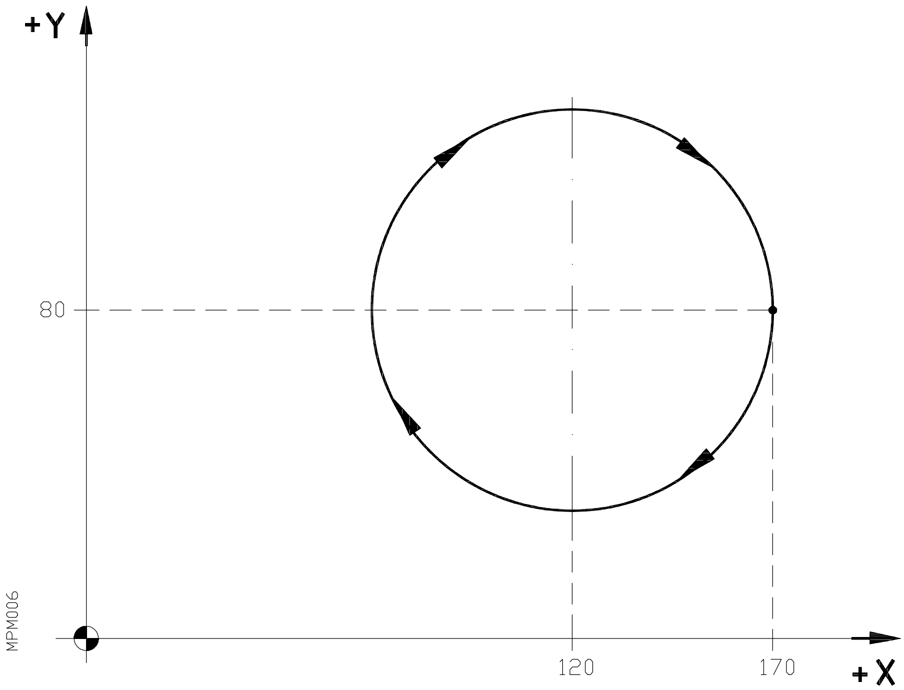
or,

N5 G93 I60 J90
N10 G90 G17 G03 A0 F150
N15 G93   I160 J90
N20   A-90

or,

N5   G93 I60 J90
N10 G91 G17 G03 A90   F150
N15 G93 I160 J90
N20 A90

Example: Single block programming of a full circle.



Assuming that the starting point is X170 Y80

Cartesian coordinate values:

N5 G90 G17 G02 X170 Y80 I-50 J0 F150

Polar coordinate values:

N5 G90 G17 G02 A360 I-50 J0 F150

or,

N5 G93 I120 J80 (Definition of polar centre)
N10 G17 G02 A360

6.2.3.2. Circular interpolation in cartesian coordinates by programming the radius

The programming format is the following:

For the XY

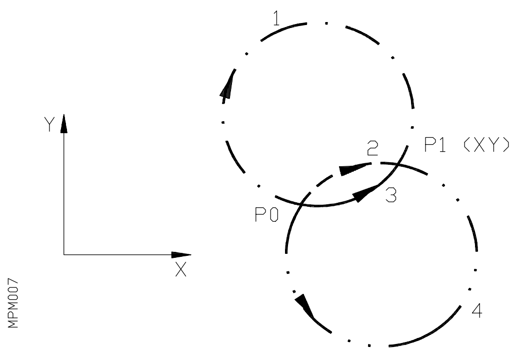
Plane G17 G02 (G03) X+/-4.3 Y+/-4.3 R+/-4.3 F5.4

This means that an arc can be programmed with its final point plus the radius (Center coordinates I,J are not required).

For the XZ Plane the format would be:
G18 G02 (G03) X+/-4.3 Z+/-4.3 R+/-4.3 F5.4

For the YZ Plane the format would be:
G19 G02 (G03) Y+/-4.3 Z+/-4.3 R+/-4.3 F5.4

If a circle is programmed by means of its radius programming, error 47 will be displayed as there are infinite solutions. If the arc is smaller than 180º, the radius will be programmed with positive sign and if it is greater than 180º, the sign will be negative.



If P0 is the initial point and P1 the final point of the arc, there are four different arcs for a given value of R.

By combining the direction (G02/G03) and the sign of R(+/-). The required arc is identified.

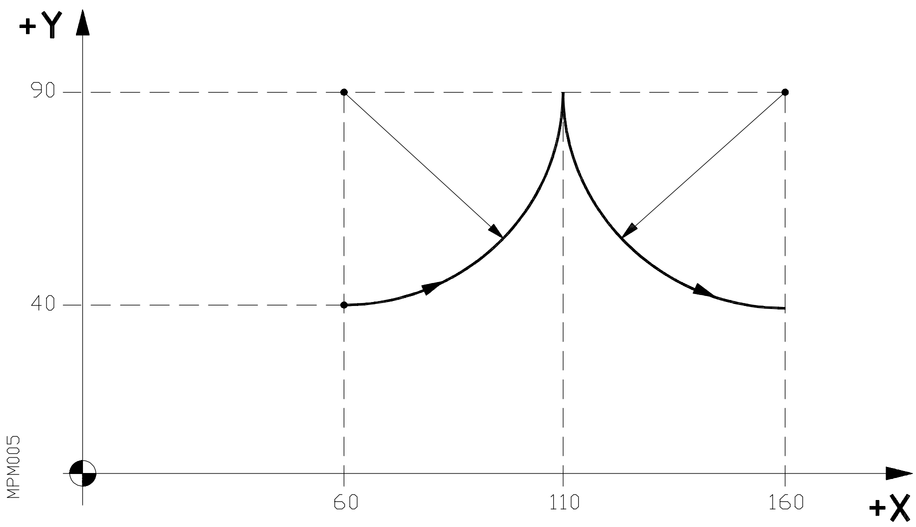
Arc 1 G02 X — Y — R -
Arc 2 G02 X — Y — R +
Arc 3 G03 X — Y — R +
Arc 4 G03 X — Y — R -

6.2.3.3. G06. Circular interpolation with absolute center coordinates

By adding function G06 in a block with circular interpolation, the coordinate values for the center of the arc (I, J, K) can be given in absolute; that is, the distance from the center to the datum point and not to the starting point of the arc.

The G06 function **is not modal**, therefore it must be programmed whenever it is wished to indicate coordinates within the arc, in absolute coordinates.

For example: Starting point X60 Y40



Circular interpolation by programming the radius:

N5 G90 G17 G03 X110 Y90 R50 F150
N10 X160 Y40 R50

Circular interpolation with absolute center coordinates:

N5   G90 G17   G06 G03 X110 Y90 I60 J90 F150
N10 G06 X160 Y40 I160 J90

6.2.3.4. Helical interpolation

Helical interpolations can be programmed by using G02/G03. Helical interpolation is defined as a circular interpolation on the main plane plus a simultaneous synchronized linear movement on the third axis. It is programmed as follows:

Cartesian coordinates In millimeters

XY plane
G02 (G03) X+/-4.3 Y+/-4.3 I+/-4.3 J+/-4.3 Z+/-4.3 K4.3 F5.4

XY. Coordinate values of the arc's final point.
IJ. Center coordinates referred to the arc's initial point.
Z. Final position on the Z axis.
K. Helical pitch on the Z axis.
F. Feedrate of the circular interpolation.

XZ plane
G02 (G03) X+/-4.3 Z+/-4.3 I+/-4.3 K+/-4.3 Y+/-4.3 J4.3 F5.4

YZ plane
G02 (G03) Y+/-4.3 Z+/-4.3 J+/-4.3 K+/-4.3 X+/-4.3 I4.3 F5.4

Polar coordinates

XY Plane
G02 (G03) A+/-3.3 I+/-4.3 J+/-4.3 Z+/-4.3 K4.3 F5.4

XZ plane
G02 (G03) A+/-3.3 I+/-4.3 K+/-4.3 Y+/-4.3 J4.3 5.4

YZ Plane
G02 (G03) A+/-3.3 J+/-4.3 K+/-4.3 X+/-4.3 I4.3 F5.4

In a helical interpolation it is also possible to program a circular interpolation by programming the radius or by using **G08** or **G09**.

Format for the **XY** plane:

G02 (G03) X+/-4.3 Y+/-4.3 R+/-4.3 Z+/-4.3 K4.3
G08 X+/-4.3 Y+/-4.3 Z+/-4.3 K4.3
G09 X+/-4.3 Y+/-4.3 I+/-4.3 J+/-4.3 Z+/-4.3 K4.3

Helical interpolations can also be programmed with the **4th axis (W)** as well as the **5th axis (V)** as long as they are linear.

Example:

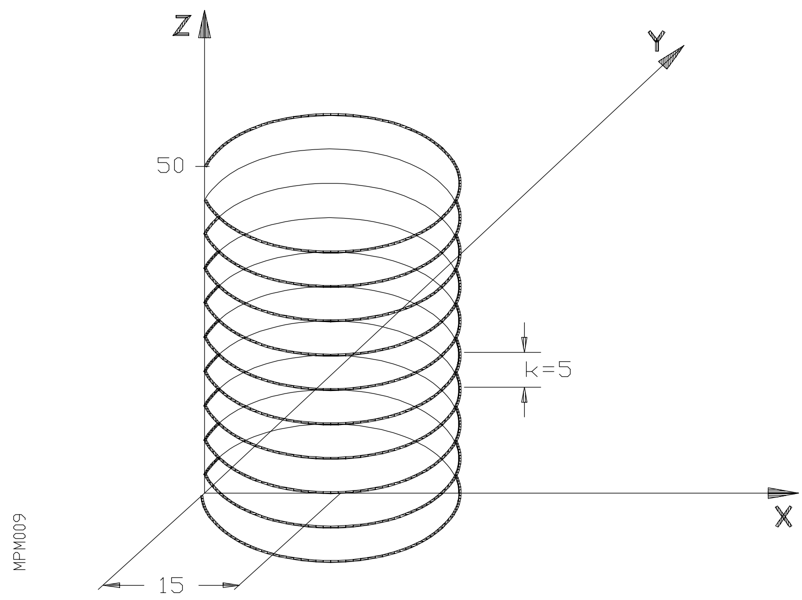
Starting from X0,Y0,Z0. The helical interpolation will be programmed as follows:

Cartesian coordinates

N10 G03 X0 Y0 I15 J0 Z50 K5 F150.

Polar coordinates

N10 G03 A180 I15 J0 Z50 K5 F150



Atention:

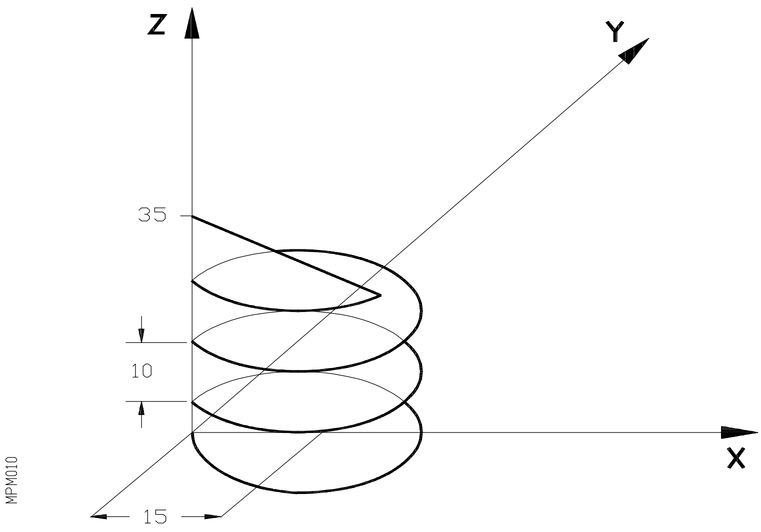
When the program is executed in **DRY RUN** operation mode (4), with no real machine movement, the path of the tool in a helical interpolation, will be displayed neither on the graphic simulation nor when a **ZOOM** function is used.

In helical movements where the final position on the axis perpendicular to the main plane is reached before the circular interpolation on the main plane (Z on the XY plane) is finished, the arc will end at the programmed coordinate value. From there to the programmed final point, the CNC will move the axes describing a straight line on a plane parallel to the main plane.

Example:

Starting from X0 Y0 Z0:

N10 G03 X0 Y0 I15 J0 Z35 K10 F250



Atention:

When a circular (helical) interpolation is programmed with G02,G03, the CNC takes the arc's center as the new polar origin.

6.3. G04. DWELL

Function **G04** can be used to program a period of time between 0,01 and 99,99 seconds. The dwell value is programmed by means of the letter **K**.

Example: G04 K0,05 Dwell of 0,05 seconds
G04 K2,5 Dwell of 2,5 seconds

If **K** is programmed directly, its value must be within 0.00 and 99.99. However, if a parameter **(K P3)** is used, the limits are 0.00 and 655.35.

The dwell is executed at the start of the block in which it is programmed.

Function **G04** can be programmed as **G4**.

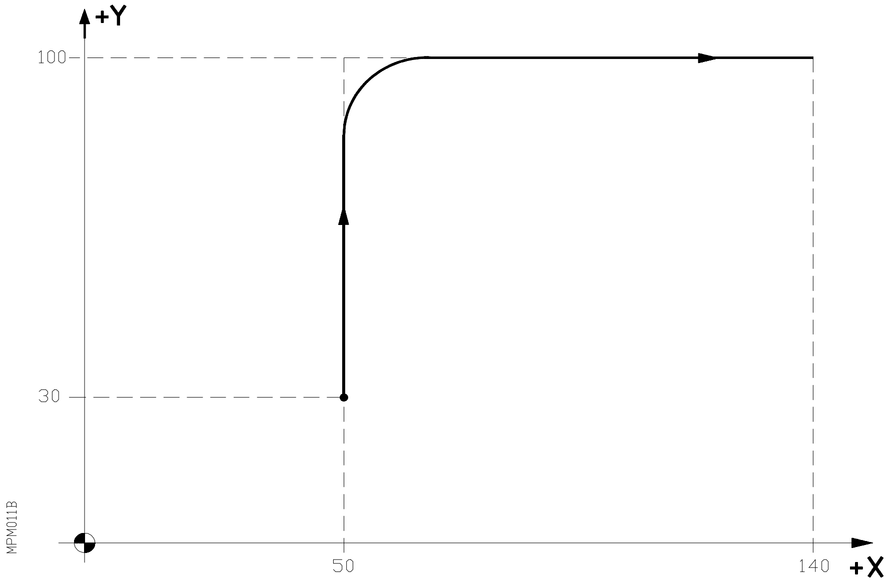
6.4. TRANSITION BETWEEN BLOCKS

6.4.1. G05. Round corner (Does not wait for in position)

When operating on G05, the CNC starts to execute the next block of the program as soon as the deceleration of the axes programmed in the previous block begins (it does not wait for in-position).

In other words, the movements programmed in the next block are executed before the machine has reached the exact position programmed in the previous block.

Example:



N1 G91 G01 G05 Y70
F100 N10 X90

As can be seen in the example, the edges would remain rounded in the case of two mutually perpendicular movements.

The difference between the theoretical and actual profiles is a function of the feedrate value.

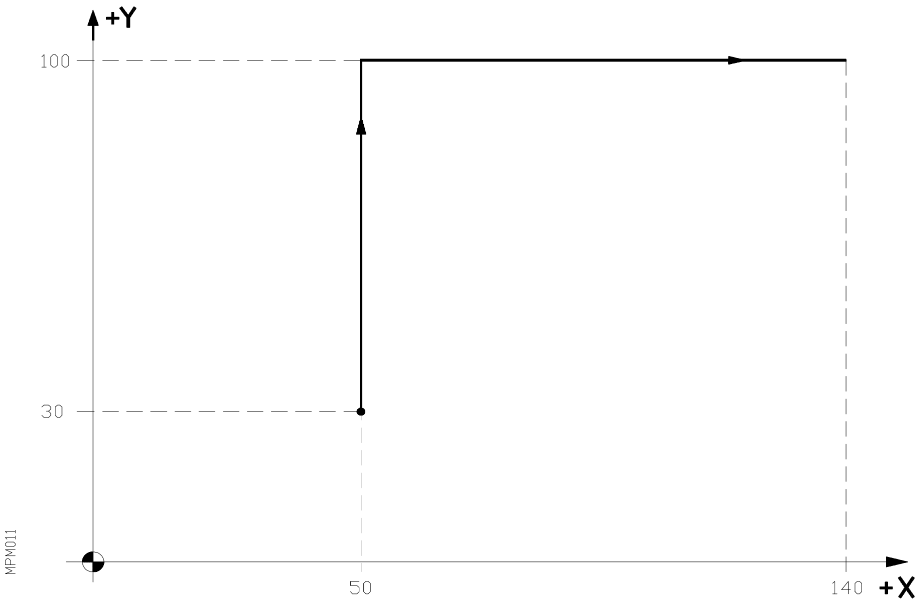
The faster the feedrate, the greater the difference between theoretical and actual profiles.

Function G05 is modal and incompatible with G07. Function G05 can be programmed as G5.

6.4.2. G07. Square corner

When operating on G07, the CNC does not execute the next block of the program until the exact position presently programmed has been reached.

Example:



N5 G91 G01 G07 Y70 F100
N10 X90

The theoretical and actual profiles coincide.
Function G07 is modal and incompatible with G05.
Function G07 can be programmed as G7.
When turned **ON** and after **M02**,**M30**, **EMERGENCY** or **RESET** if machine parameter, the CNC assumes function G07 or G05 depending on the value assigned to machine parameter P613(5), i.e.,

. With P613(5) = 0, it assumes G07
. With P613(5) = 1, it assumes G05

6.5. G08. ARC TANGENT TO PREVIOUS PATH

An arc tangent to the previous path can be programmed by means of G08. Center coordinates (I,J,K) are not required.

Cartesian coordinates (XY plane)

N4 G08 X+/-4.3 Y+/-4.3

N4 : Block number

G08 : Code defining circular interpolation tangent to previous path

X+/-4.3 : Coordinate values of the arc’s final point

Y+/-4.3 : Coordinate values of the arc’s final point

Polar coordinates:

N4 G08 R+/-4.3 A+/-4.3

N4 : Block number.

G08 : Code defining circular interpolation tangent to previous path.

R+/-4.3 : Radius (referred to polar origin) of the arc’s final point.

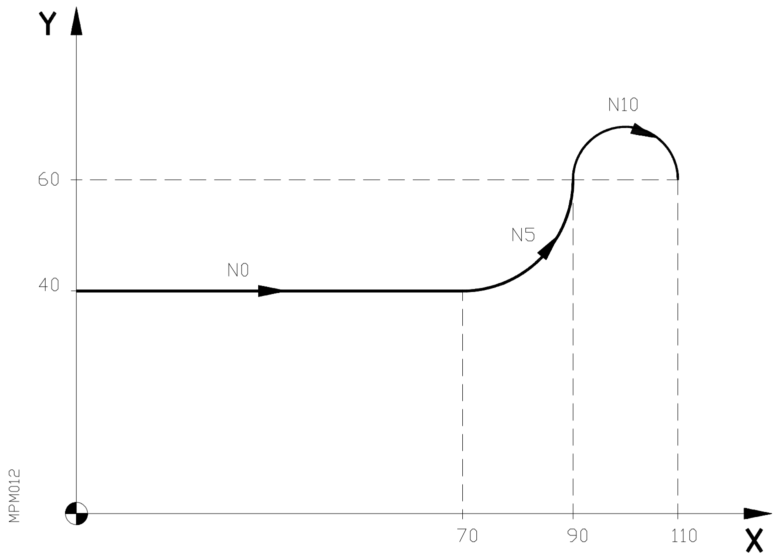
A+/-4.3 : Angle of the arc’s final point.

Example:

The starting point being X0 Y40 the programming of the following path is described:

- Straight line
- Arc tangent to the previous line
- Arc tangent to the previous arc

```
N0  G90  G01 X70   F100
N5  G08  X90  Y60
N10 G08  X110 Y60
```



An alternative way of programming (using I,J) would be the following:

```
N0  G90  G01  X70 F100
N5  G03  X90  Y60    I0  J20
N10 G02  X110 Y60   I10 J0
```

The function G08 is not modal. It replaces G00,G01,G02 or G03 only in the block in which it is written.

The previous path can be a straight line or an arc.

G08 replaces G02 and G03 only in the block in which it is written.

Atention:

A circle cannot be executed with G08 function, as there are infinite solutions. The CNC will display the error code 47.

6.6. G09. ARC PROGRAMMED BY THREE POINTS

Two points (the final plus one intermediate point) are sufficient to program an arc provided that the actual position is the starting point. In other words, an intermediate point is programmed instead of the center.

This feature can be useful when a part is programmed in **PLAY BACK** and after writing G09 in the block the machine can be manually shifted to the intermediate point of the arc and press **ENTER**. Then to the final point and press **ENTER**. In this way, the block will be stored in the memory.

Cartesian coordinates (XY plane)

N4 G09 X+/-4.3 Y+/-4.3 I+/-4.3 J+/-4.3

N4 : Block number.

G09 : Code identifying 3 point arc definition.

X+/-4.3 : X value of the arc's final point.

Y+/-4.3 : Y value of the arc's final point.

I+/-4.3 : X value of the intermediate point.

J+/-4.3 : Y value of the intermediate point.

Polar coordinates (XY plane)

N4 G09 R+/-4.3 A+/-4.3 I+/-4.3 J+/-4.3

N4 : Block number.

G09 : Code identifying 3 point arc definition.

R+/-4.3 : Radius (referred to polar origin) of the final point of the arc.

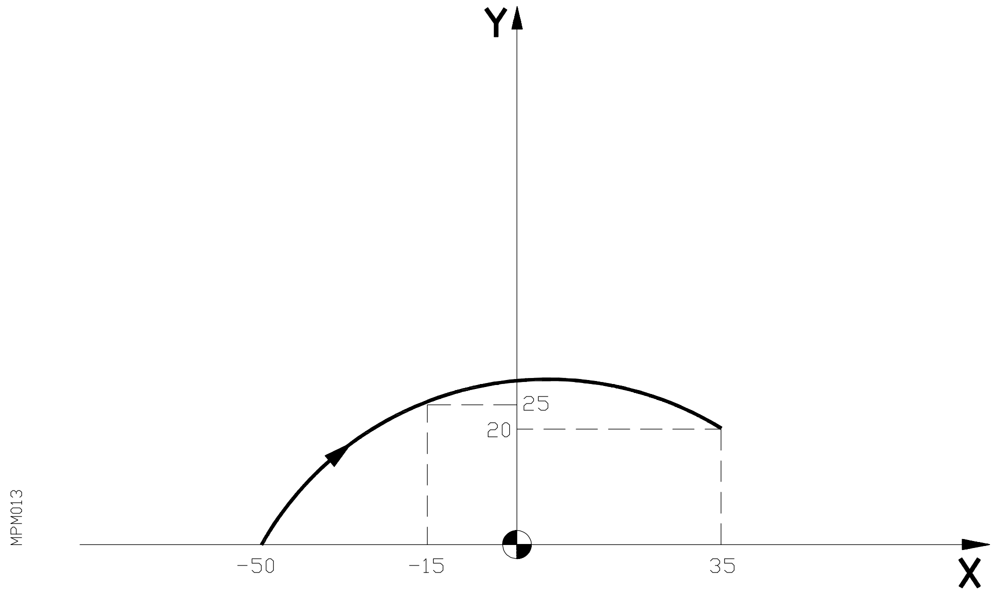
A+/-4.3 : Angle (referred to polar origin) of the final point of the arc.

I+/-4.3 : X value of the intermediate point.

J+/-4.3 : Y value of the intermediate point.

The intermediate point must always be programmed in cartesian coordinates.

Example:



N10 G09 X35 Y20 I-15 J25

G09 is not modal.

It is not necessary to program the direction of the arc (G02,G03) when G09 is programmed.

Function G09 replaces G02 and G03 only in the block in which it is written.

Atention:

A complete circle cannot be performed via G09 since three different points must be programmed (the starting and final points must be different). Otherwise error code **40** will be generated.

6.7. MIRROR IMAGE

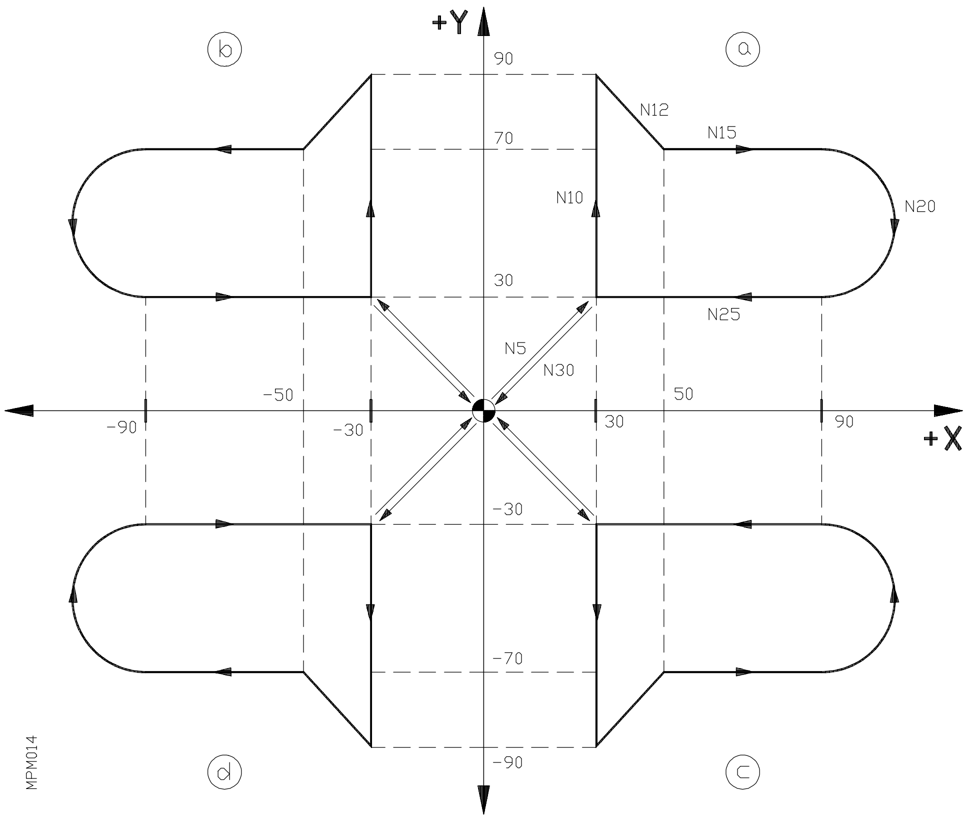
G10: Cancellation of mirror image
G11: Mirror image on the X axis
G12: Mirror image on the Y axis
G13: Mirror image on the Z axis

When the CNC operates on G11,G12,G13 it executes the movements programmed on X,Y,Z with the sign reversed.

Functions G11,G12,G13 are modal; i.e. once programmed they persist until G10 is programmed.

Functions G11,G12,G13 can all be programmed in the same block, since they are not incompatible.

Example:



a) N5 G91 G01 X30 Y30 F100
N10 Y60
N12 X20 Y-20
N15 X40
N20 G02 X 0 Y-40 I 0 J-20
N25 G01 X-60
N30 X-30 Y-30

b) N35 G11
N40 G25 N5.30

c) N45 G10 G12
N50 G25 N5,30

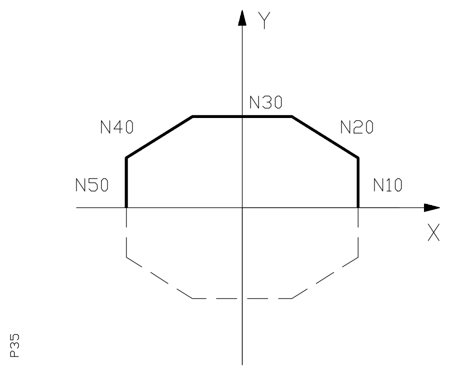
d) N55 G11 G12
N60 G25 N5.30
N65 M30

If mirror imaging is programmed while G73 (pattern rotation) is active the CNC will apply mirror image first and then the rotation.

In **4** (5) axis machines, mirror image cannot be applied to the **4th (5th) axis**.

The CNC assumes G10 on being turned ON, after executing **M02,M30** or after an **EMERGENCY** or **RESET**.

Case of continuous figures



N10 X— Y—
N20 ' '
N30 ' '
N40 ' '
N50 ' '
N60 G11 G12
N70 G25 N10.50
N80 M30

In continuous figures, the mirror image will only be used after having programmed half the part.

Afterwards, we will use G11 G12.

6.8. PLANE SELECTION

G17 : Selection of the XY plane
G18 : Selection of the XZ plane
G19 : Selection of the YZ plane

The main plane must be correctly selected in order to perform: Circular interpolation, controlled corner rounding, tangential approach, tangential exit, chamfering, canned cycles, pattern rotation, tool compensation.

The CNC applies radius compensation to the two axes of the plane selected and length compensation to the axis perpendicular to that plane.

As previously explained (G02/G03), in the case of four (five) axis machines the same codes (G17,G18,G19) are used for working with the fourth (fifth) axis.

If the W (V) axis is incompatible with the X axis.

G17 : Selection of the XY or WY or VY plane
G18 : Selection of the XZ or WZ or VZ plane

If they are incompatible with the Y axis.

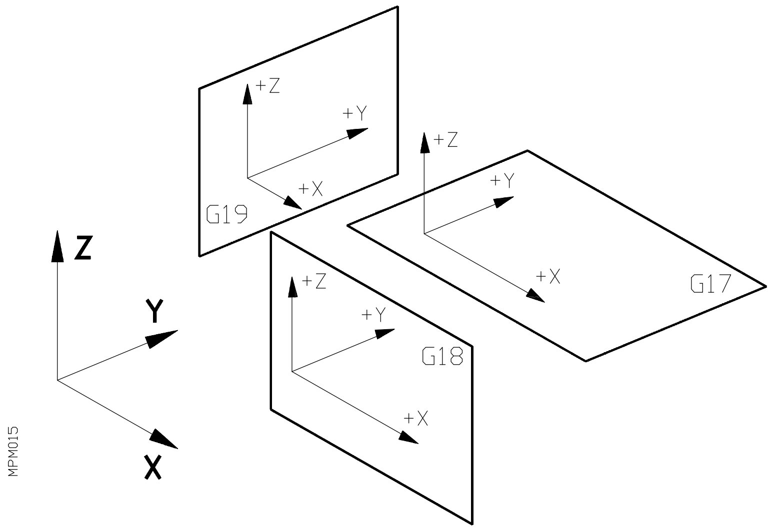
G17 : Selection of the XY or XW or VX plane
G19 : Selection of the YZ or WZ or VZ plane

If they are incompatible with the Z axis.

G18 : Selection of the XZ or XW or VX plane
G19 : Selection of the YZ or YW or VY plane

Functions G17,G18,G19 are modal and incompatible with one another.

The CNC assumes function G17 on being turned on, after executing **M02,M30** or after an **EMERGENCY** or **RESET**.



6.9. G25. UNCONDITIONAL JUMP/CALL

The function G25 can be used to jump to another block of the current program. In the same block in which the G25 function is programmed it is not possible to program more information. There are two possibilities:

Format a) N4 G25 N4

N4 - Block number
G25 - Code for unconditional jump
N4 - Number of the block the jump is aimed at

When the CNC reads this block, it jumps to the targeted block and the program continues.

Example:

```
N0   G00  X100
N5   Z50
N10  G25  N50
N15  X50
N20  Z70
N50  G01  X20
```

When the block 10 is reached, the CNC jumps to block **50** and then the program continues until it is finished.

b) N4 G25 N4.4.2

N4 → Block number
G25 → Code unconditional jump
N4.4.2 → Number of repetitions
→ Number of the last block to be executed
→ Number of the block to which the jump is targeted

When the CNC reads such a block, it jumps to the block identified between the **N** and the first decimal point. It then executes the section of the program between this block and the one identified between the two decimal points as many times as set by the last digit. This lastdigit can take a value within 0 and 99, unless it is programmed using a parameter in which case the limits are 0 and 255. If only N4.4 is written, the CNC will assume N4.4.1. When the execution of this section is finished the CNC goes to the block next to the one in which G25 N4.4.2 was programmed.

Example:

```
N0   G00  X10
N5   Z20
N10  G01  X50  M3
N15  G00  Z0
N20  X0
N25 G25  N0.20.8
N30  M30
```

When block **25** is reached, the CNC will jump to block **0** and will execute 8 times the section N0-N20. On completion of this, it will go to block **30**.

Functions **G26,G27,G28,G29** and **G30** will be described in the corresponding chapter of this manual **PARAMETRIC PROGRAMMING, OPERATIONS WITH PARAMETERS**.

6.10. G31-G32. STORAGE AND RETRIEVAL OF PART PROGRAM'S ZERO POINT

G31 : Store present program’s datum point
G32 : Retrieve datum point stored by G31

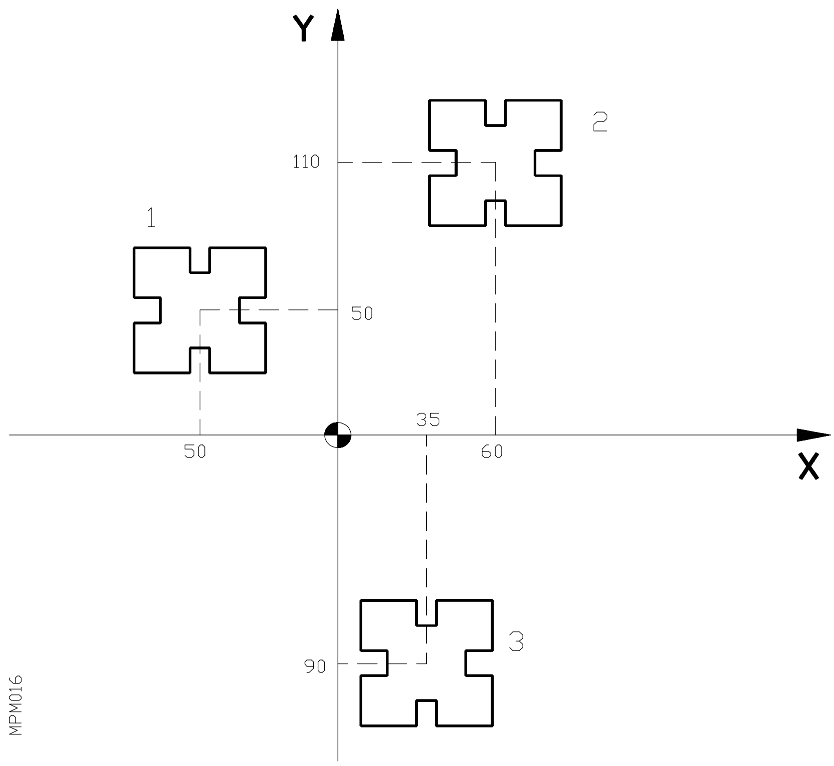
By means of the **G31** function, it is possible at any time to store the zero point which we are working with and recover it later by means of the **G32** function.

This feature is intended to simplify the operation with multi-zero part programs. A datum point can be stored any time with **G31**, change the zero point with **G92** or **G53-G59**, dimension the continuation of the program with respect to the new zero point and later retrieve the original zero point by **G32**.

No other function can be programmed in a block in which **G31** or **G32** is programmed. The format is:

N4 G31
N4 G32

N4 : Block number
G31 : Store current zero point
G32 : Recover the stored zero point with G31.



Example:
The tool's starting point is X0 Y0 Z5

```
N10  G00  G90  X-50  Y50  (Tool over the center of fig. 1)

N20  G20  N1.1            (Calling of subroutine number 1)
N30  X60  Y110            (Tool over the center of fig. 2)

N40  G20  N1.1
N50  X35  Y-90            (Tool over the center of fig. 3)

N60  G20  N1.1
N70  M30                  (End of program.)

N100 G22  N1              (Definition of subroutine number 1)
N110 G31                  (Store current datum point.)
N120 G92  X0 Y0           (Preselection of coordinate values)
N130 G1   Z-20  F350      (Shift the tool.)
N140 X—  Y—               (Programming of internal contour in Fig 1)

N ---
N ---
N ---
N ---
N ---
N200 G0   Z5              (Return the tool to starting point)

N210 G32                  (Retrieve datum point stored by G31)

N220 G24                  (End of subroutine.)
```

6.11. G33. THREADCUTTING

If the milling machine's spindle does have an encoder, threadcutting can be carried out with function **G33**.

G33 is modal, i.e. it remains active until cancelled by **G00**,**G01**,**G02**,**G03**,**M02**,**M03**, **EMERGENCY** or **RESET**.

Format:

N4 G33 Z+/-4.3 K3.4 (metric)
N4 G33 Z+/-3.4 K2.4 (inches)

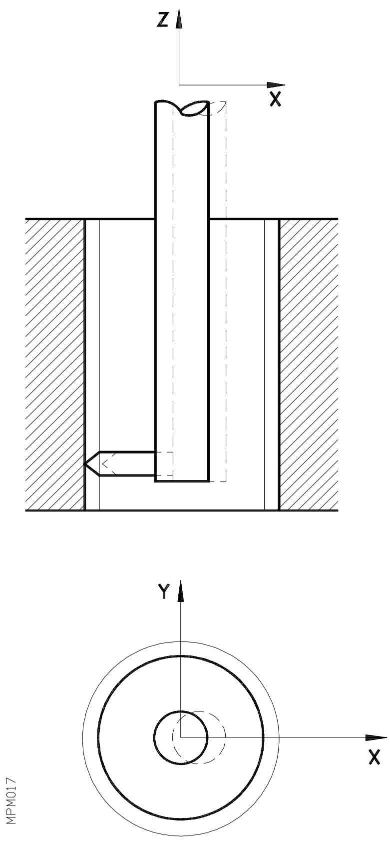
N4	- Block number
G33	- Threadcutting code
Z+/-4.3 (+/-3.4)	- Coordinate value of the final point of the thread. It will be absolute or incremental depending on G90 or G91.
K3.4 (2.4)	- Pitch Operating in G05 mode threads of different pitch can be cut without losing synchronism

While **G33** is active, The **FEEDRATE OVERRIDE** knob on the front panel has no effect and the feedrate is set for 100%. Also, the spindle speed cannot be altered from the front panel keys.

Example: Cut a thread using a boring tool placed 10 mm higher than the surface of the part. The surface is considered Z=0 and the thread is to be cut around the point X=0 Y=0.

This thread, 5 mm lead and 100 mm deep, must be cut in one pass.

```
N0  G90  G33  Z-100  K5
N5  M19
N10 G00  X3
N15 Z30
N20 X0 Z10 M03
```



Block N0

The tool will move up to **Z-100** cutting a thread of 5 mm pitch.

Block N5

When reading **M19** the CNC commands a very slow rotation of the spindle until it reaches the correct withdrawal position.

Block N10

The example has assumed that the tool is pointing in the **X** axis direction when stopped. (This position is determined when setting up the machine). The tool withdraws 3 mm in rapid to clear the return.

Block N15

The tool withdraws in rapid to **Z30** (30 mm above thepart's surface).

Block N20

The spindle is started again and goes in rapid to the starting point X0,Y0,Z10. From then on, the threadcutting can be repeated.

6.12. G36. AUTOMATIC RADIUS BLEND

This function, **G36**, rounds the corners with a programmed radius, without the need to calculate the coordinates of the center and the initial and final points of the arc.

G36 is not modal; i.e. it must be programmed every time a corner rounding is needed.

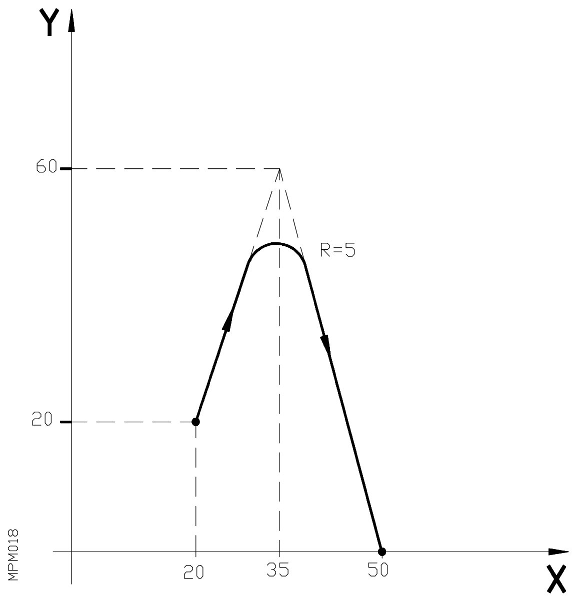
It must be programmed in the same block as the movement whose end must be rounded.

The rounding radius must be always positive (R 4.3 in mm) (R 3.4 in inch).

X .- X value of intersection point of the two G01 moves.
Y.- Y value of intersection point of the two G01 moves.

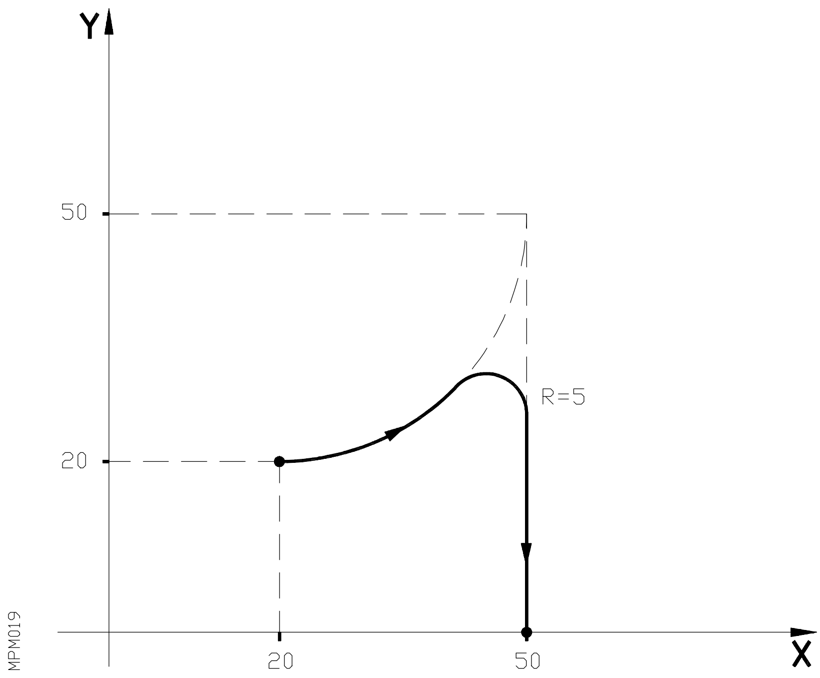
Examples:

1.



N50 G90 G01 G36 R5 X35 Y60 F100
N60 X50 Y0

2.

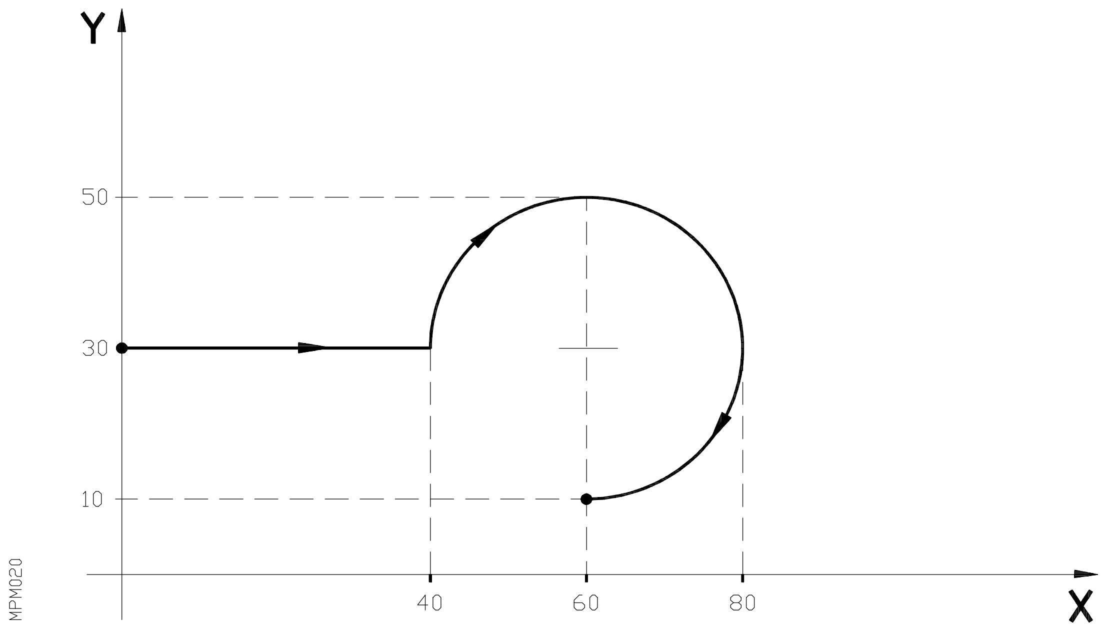


N50 G90 G03 G36 R5 X50 Y50 I0 J30 F100
N60 G01 X50 Y0

6.13. G37. TANGENTIAL APPROACH AT THE START OF MACHINING

The preparatory function **G37** can be used to link two paths tangentially without having to calculate the intersection points. Function **G37** is not modal, so it has to be programmed every time a machining operation with tangential entry is to be started.

Example:

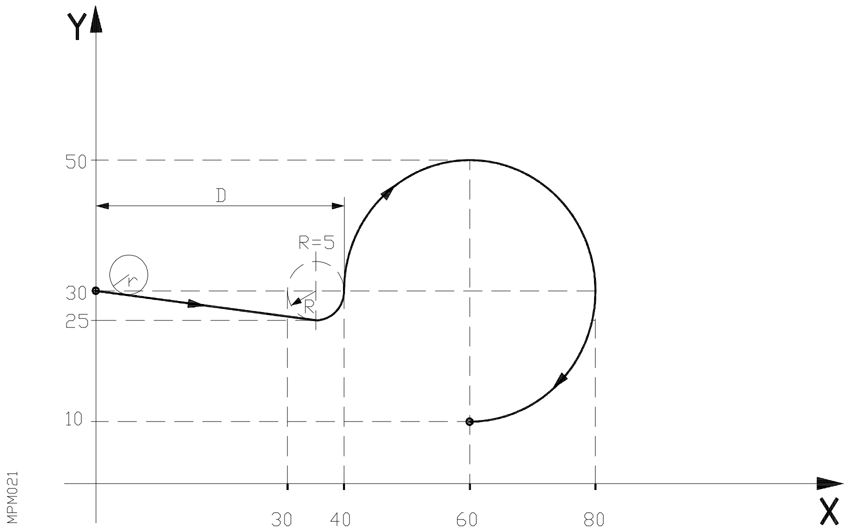


Let us suppose that the starting point is X0,Y30 and that an arc of circle is to be machined and the approach path is to be rectilinear. We shall program as follows:

```
N0  G90  G01  X40   F100
N5  G02  X60  Y10   I20  J0
```

In the same example, if we want the tool entry to the part being machined to be tangential to the path (See fig.), describing a radius of 5 mm., the following must be programmed:

```
N0  G90  G01  G37  R5  X40  F100
N5  G02  X60  Y10  I20  J0
```



As can be seen in the diagram, the CNC modifies the path of block **N0** so that the tool starts machining with a tangential entry to the part.

Function **G37** and the value of **R** have to be programmed in the block which incorporates the path that is to be modified.

The value of R has in all cases to be a continuation of **G37** and indicates the radius of the arc of circle which the CNC introduces in order to achieve a tangential entry to the part. This value of **R** must always be positive.

Function **G37** may only be programmed in a block which incorporates rectilinear movement (G00 or G01). If it is programmed in a block which incorporates circular movement (G02 or G03), the CNC will show a type **41** error.

* **G37** is programmed with the input radius.

Conditions to be borne in mind.

a)**D** ≥ 2 entry radius.

b)Radius r of the milling tool entry Radius **R**.

c)The entry section must be linear. It cannot be circular.

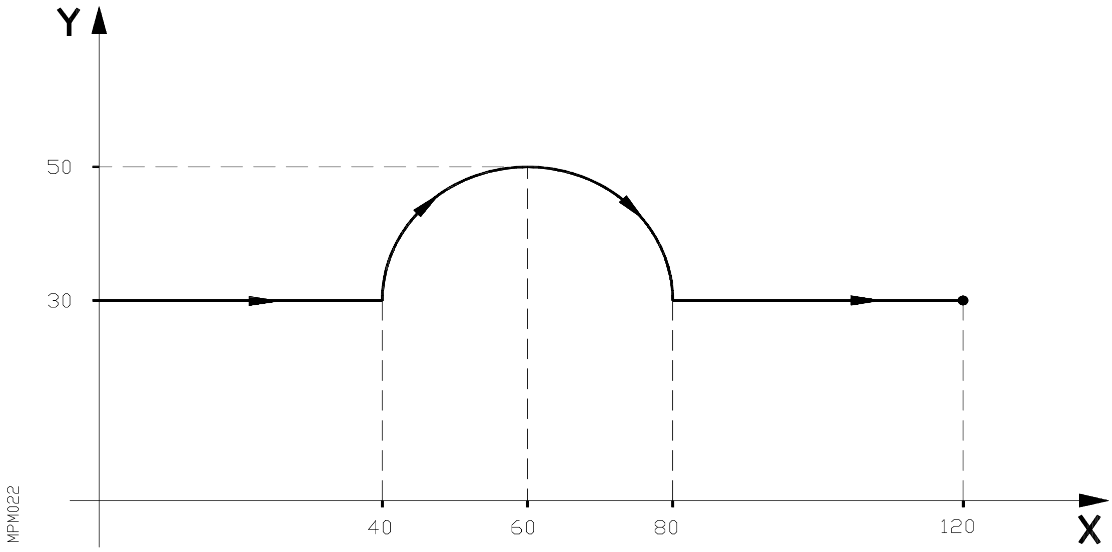
6.14. G38. TANGENTIAL EXIT ON COMPLETION OF MACHINING

Function **G38** enables a machining operation to be completed with a tangential exit of the tool without involving cumbersome calculations.

Function **G38** is not modal, so it has to be programmed every time a tangential tool exit is required.

The radius (R 4.3 in mm)(R 3.4 in inch) of the exit arc must be programmed to follow **G38**.

Example:



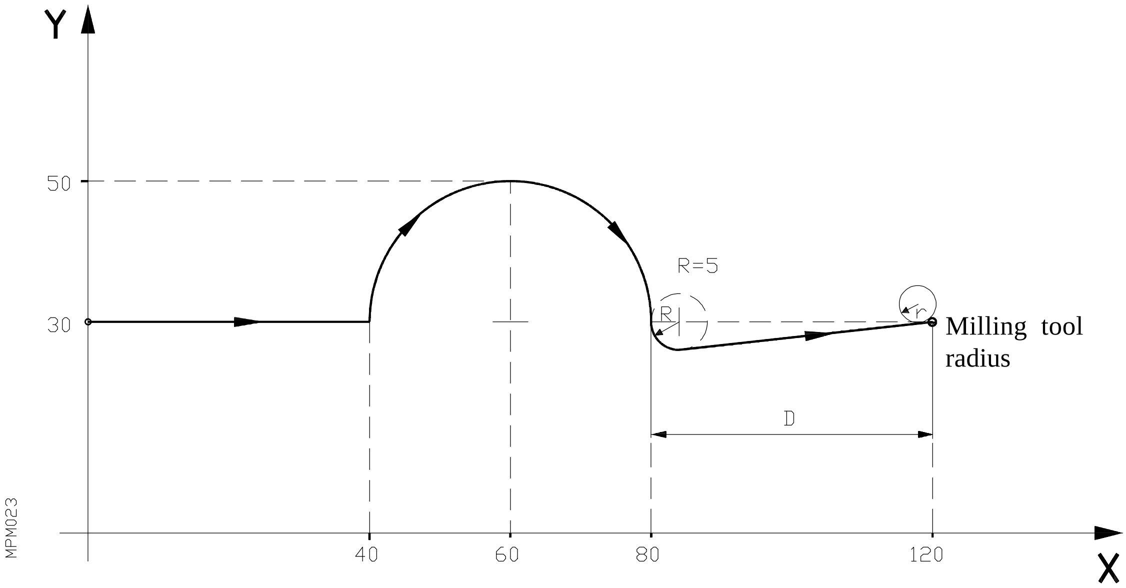
Let us suppose that the starting point is X0,Y30. The initial straight section is an approach movement involving no machining, the circular section performs a machining operation and the final straight involves no machining.

The program will be:

```
N0   G90   G01  X40 F100
N5   G02   X80  Y30 I20  J0
N10  G00  X120
```

If the tool exit on completion of machining is to be tangential, e.g. with an exit radius of **5** mm., the following must be programmed:

```
N0  G90  G01 X40 F100
N5  G90  G02  G38  R5  X80  Y30  I20  J0
N10 G00  X120
```



The movement programmed in the block following the one including **G38**, must necessarily be rectilinear (G00 or G01).

In the subsequent path is circular (G02 or G03), the CNC will show a type **42** error.

* Conditions for using **G38** are similar to **G37**.

6.15. G39. CHAMFERING

This function chamfers the corner between two straight lines without the need to calculate the coordinates of the two intersections.

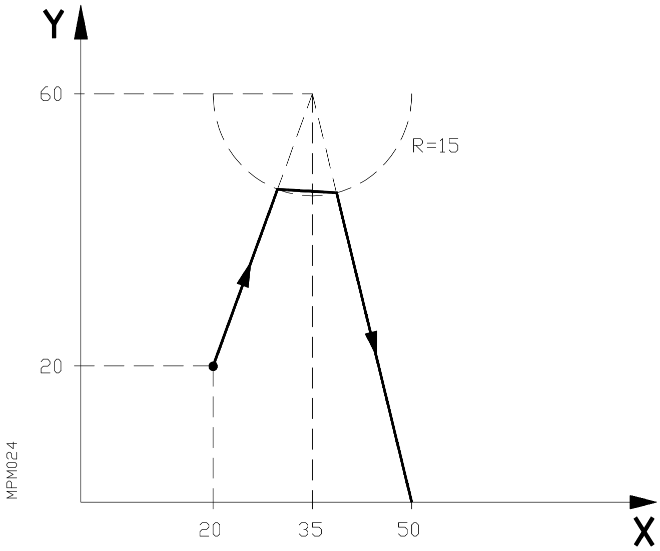
G39 is not modal, i.e. it must be programmed every time a chamfering is needed.

It must be programmed in the same block as the movement whose end must be chamfered.

Use the code R4.3 in mm (R3.4 in inch), always positive, to program the distance between the final point programmed and the point in which the chamfer is to start.

X .- X value of intersection point of the two G01 moves.
Y.- Y value of intersection point of the two G01 moves.

Example:



N0 G90 G01 G39 R15 X35 Y60 F100
N10 X50 Y0

6.16. TOOL RADIUS COMPENSATION

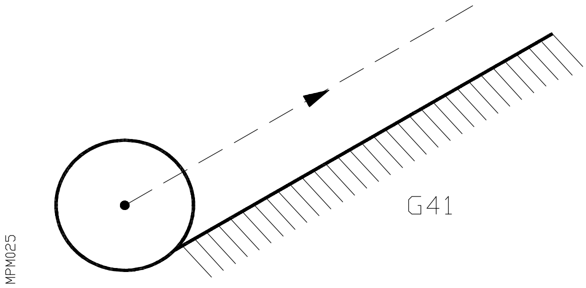
In normal milling work the path of the tool has to be calculated and defined taking its radius into account so as to obtain the required dimensions of the part produced.

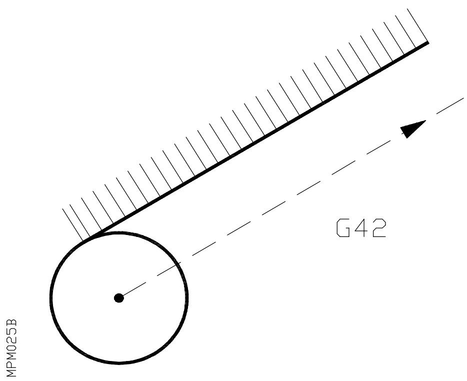
Tool radius compensation enables the contour of the part to be programmed directly without taking the dimensions of the tool into account.

The CNC automatically calculates the path to be followed by the tool, based on the contour of the part and the tool radius value stored in the tool table.

There are three preparatory functions for tool radius compensation:

G40 : Cancellation of tool radius compensation
G41 : Left hand tool radius compensation
G42 : Right hand tool radius compensation





G41. The tool is on the left of the part as seen from the direction of movement.

G42. The tool is on the right of the part as seen from the direction of movement.

The CNC has a table of up to 100 pairs of values for tool radius compensation. **R** identifies the tool radius and I the tool wear. The CNC will add (or subtract) the value of **I** to the value of **R**.

The maximum compensation values are:

R +/-1000 mm or +/-39.3699 inches
I +/-32.766 mm or +/-1.2900 inches

The compensation values must be stored in the tool table (operating mode 8) before starting the machining or else at the beginning of a part program by means of **G50**.

The values of **I**,**K** can also be checked and modified without stopping the cycle's execution (See Operation Manual).

Once the plane in which the compensation is to be applied has been determined by codes **G17**,**G18**,**G19** the compensation is made effective by means of **G41** or **G42** and acquires the table value selected by code Txx.xx (Txx.00-Txx.99).

Functions **G41** and **G42** are modal (persistent) and are cancelled by **G40**, **G74**, **G81**, **G82**, **G83**,**G84**,**G85**,**G86**,**G87**,**G88**,**G89**,**M02** and **M30** and by **EMERGENCY** or **RESET**.

6.16.1. Selection and initiation of tool radius compensation

Once **G17**,**G18** or **G19** has been used to select the plane in which tool radius compensation is to be applied, the code **G41** or **G42** must be used to initiate compensation.

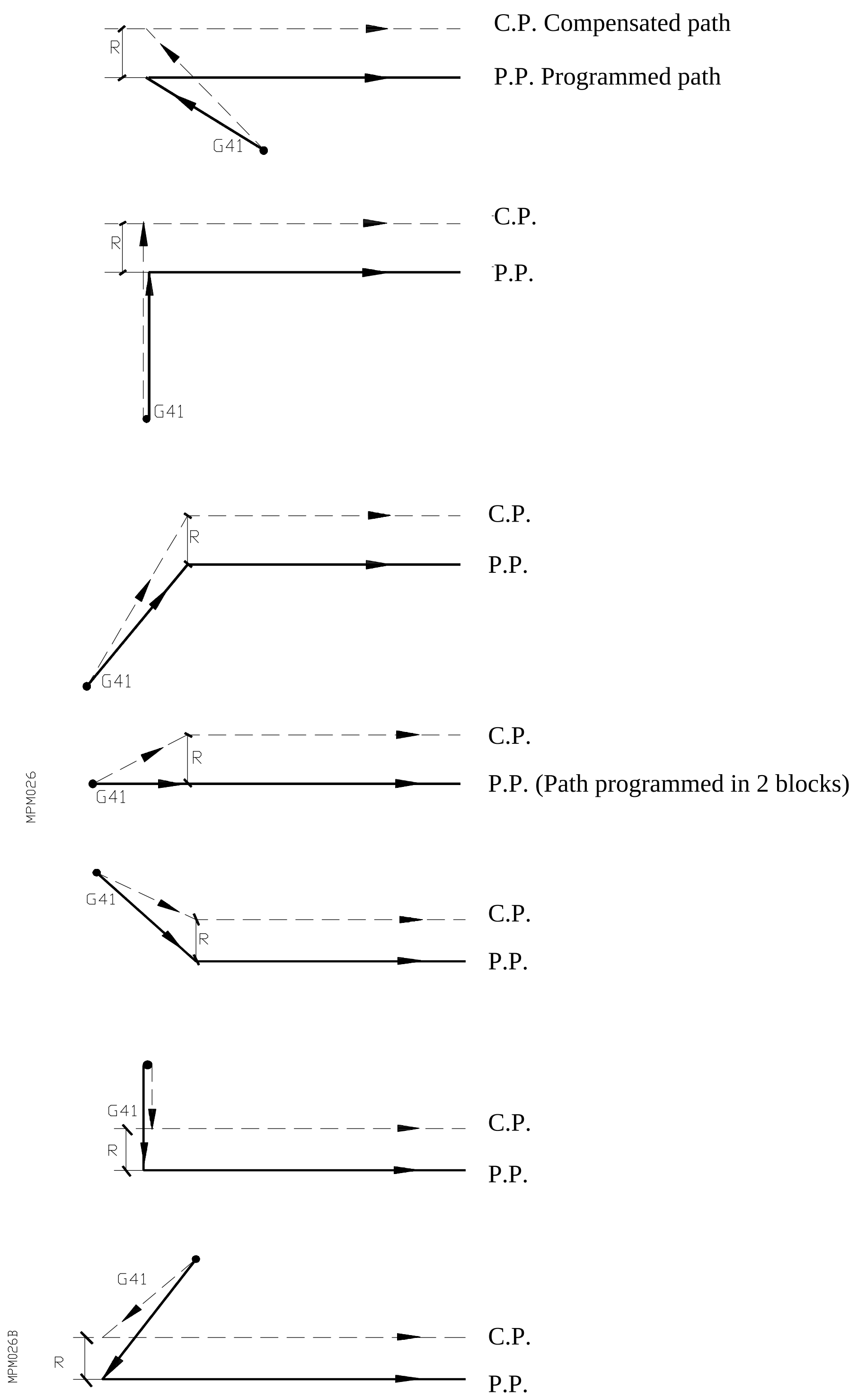
G41: The tool remains to the left of the part in the machining direction.

G42: The tool remains to the right of the part in the machining direction.

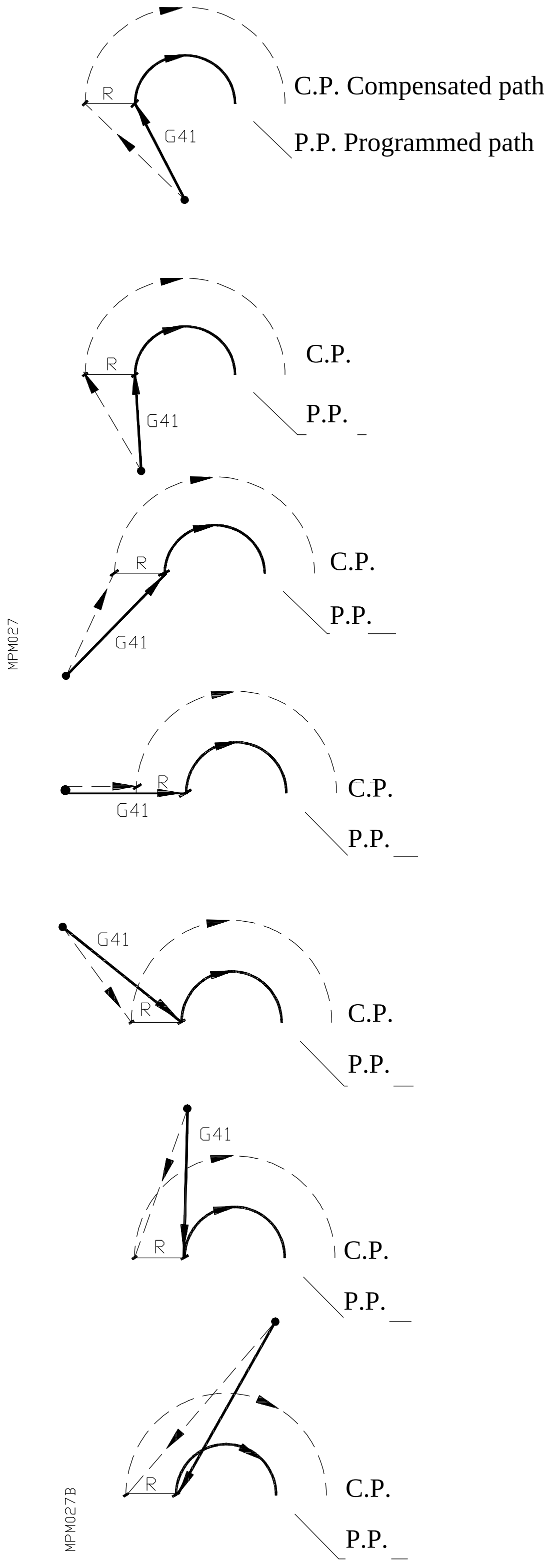
Either the block in which **G41**/**G42** is programmed or a previous block must include programming of function Txx.xx (Txx.00-Txx.99) to select from the tool table the correction value to be applied. If no tool is selected, the CNC assumes the value T00.00.

Tool radius compensation selection (G41/G42) can only be carried out when **G00** or G01 (rectilinear movements) is active. If the first call for compensation is made when **G02** or **G03** are active, the CNC will display error code **40**. The next pages illustrate various cases of initiation of tool radius compensation.

STRAIGHT-STRAIGHT PATH



STRAIGHT-CURVE PATH

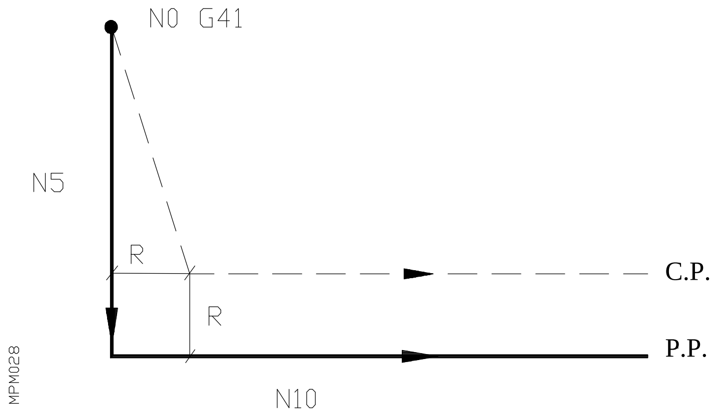


Special cases to be considered

a. If compensation is programmed in a block in which there is no movement, the initiation of the compensation differs from the case explained above (compare with diagram in section on Straight/straight path).

N0 G91 G41 G01 T00.00
N5 Y-100
N10 X+100

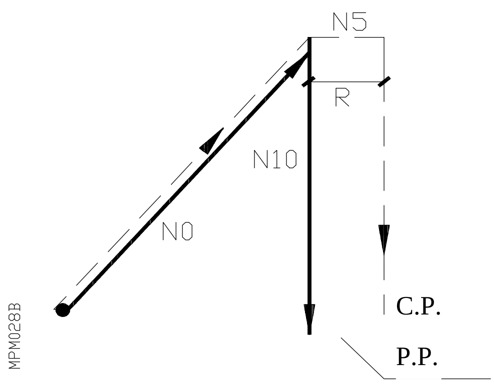
1)



b. If compensation is entered with zero movement programming:

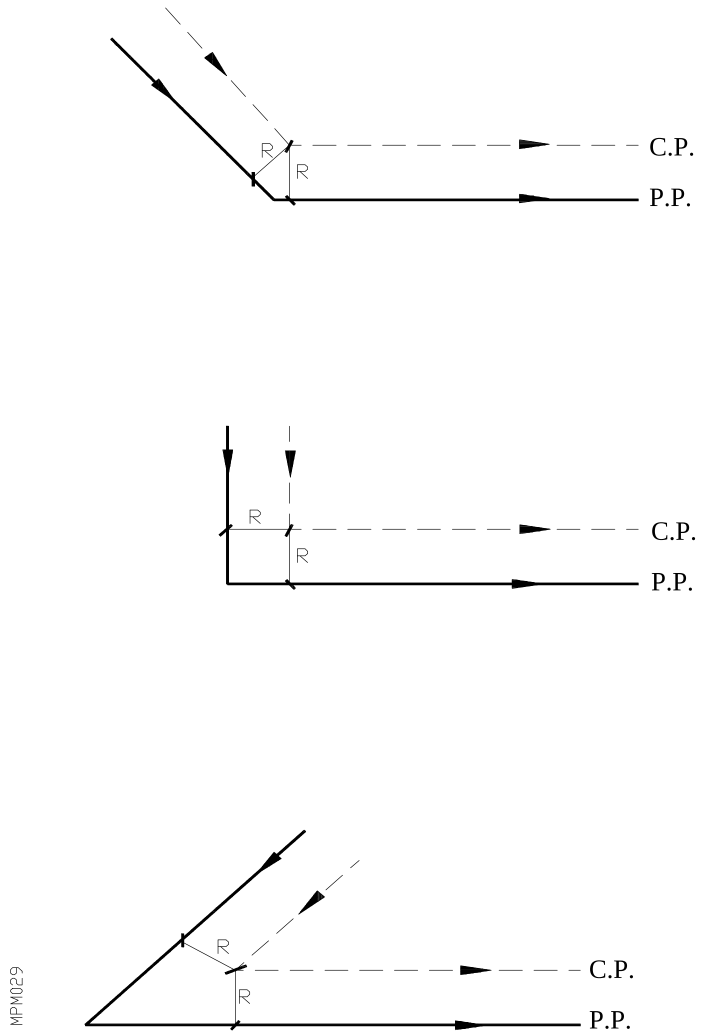
N0 G91 G01 X100 Y100
N5 G41 X0 T00.00
N10 Y-100

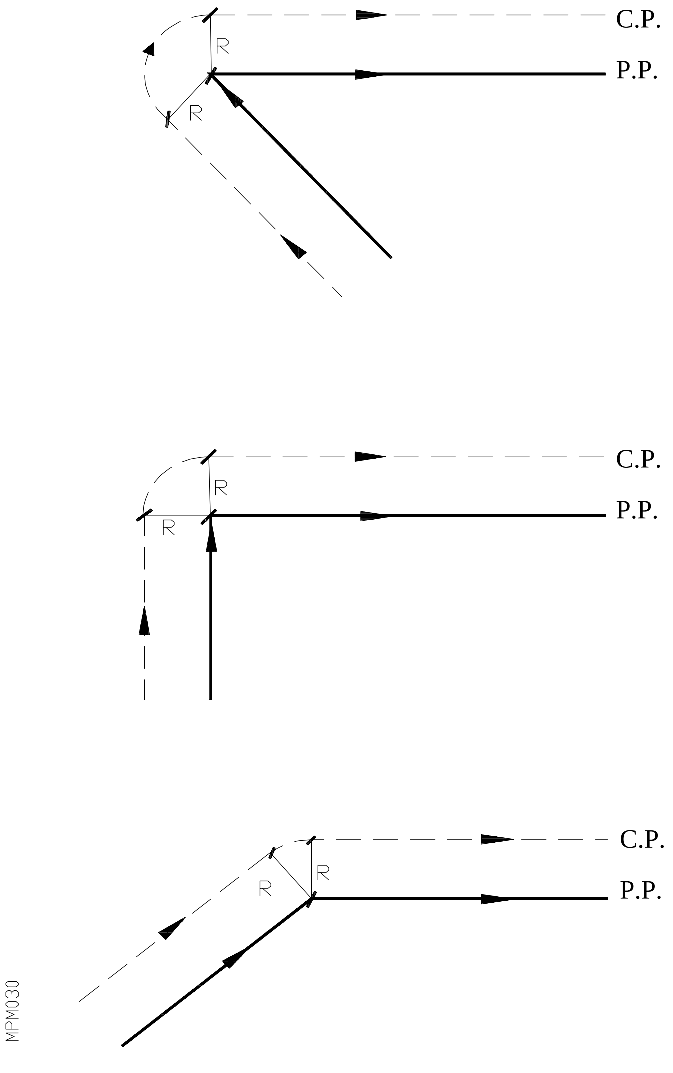
2)

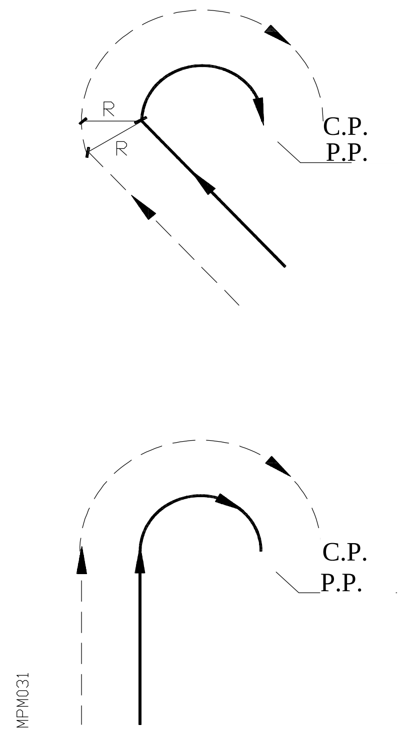


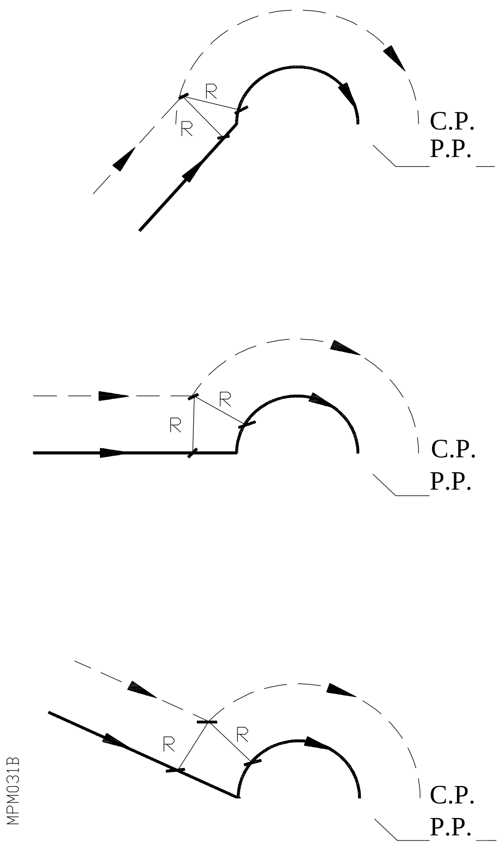
6.16.2. Operating with tool radius compensation

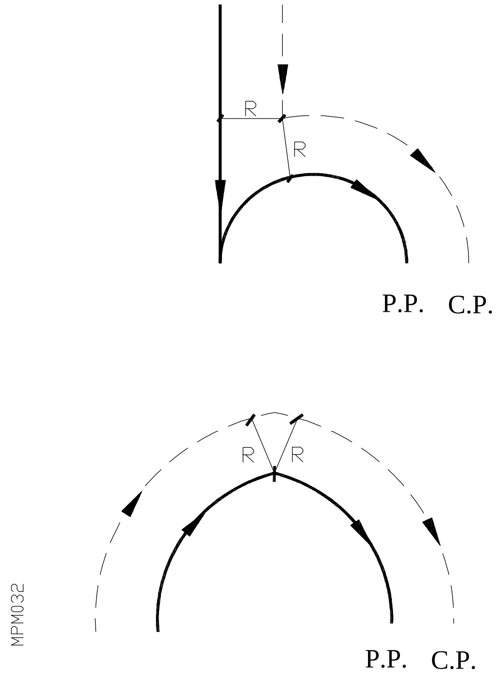
The graphs below illustrate the various paths followed by a tool controlled by a CNC programmed with radius compensation.

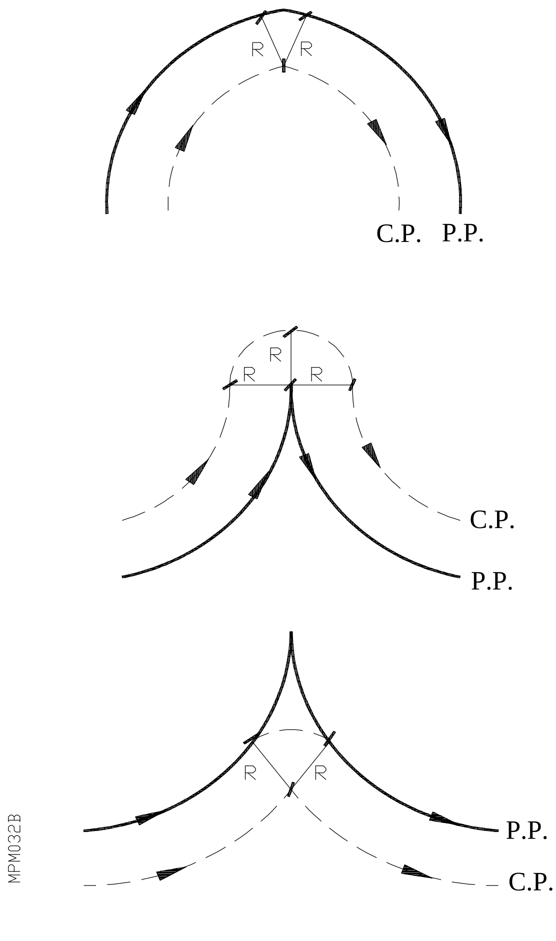



C.P.
P.P.
R
R
C.P.
P.P.
R
R
C.P.
P.P.
R
R
MPM030


R
R
C.P.
P.P.
C.P.
P.P.
MPM031


R
R
C.P.
P.P.
R
R
C.P.
P.P.
R
R
C.P.
P.P.
MPM031B


R
R
P.P.
C.P.
R
R
P.P.
C.P.
MPM032

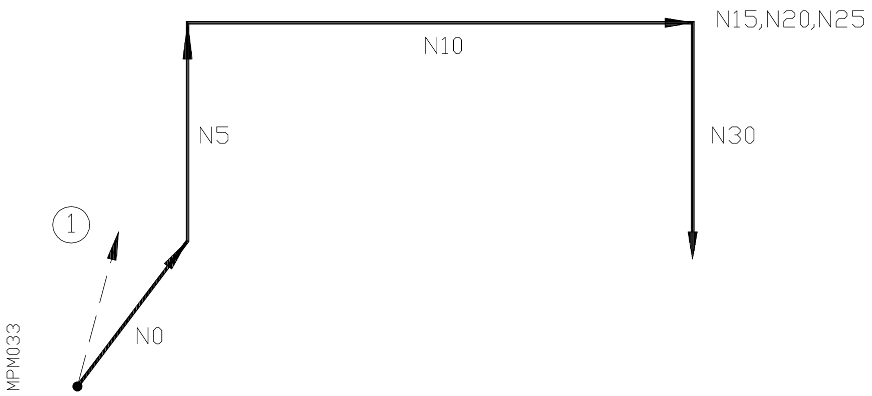

R
R
C.P.
P.P.
R
R
R
C.P.
P.P.
R
R
P.P.
C.P.
MPM032B

When the CNC operates with tool radius compensation, it reads four blocks ahead of the block being executed so that it can calculate in advance the path to be followed.

There are certain cases in which particular care has to be taken.

For instance:

Three or more blocks which do not include movement in the compensation plane, between blocks which do.



N0 G01 G91 G17 G41 X50 Y50 F100 T1.1
N5 Y100
N10 X200
N15 Z100
N20 M07
N25 Z200
N30 Y-100

Error **35** will be displayed at point (1). Only blocks containing G20, G21, G22, G23, G24, G25,G26,G27,G28 or G29 can be programmed and they will not originate error **35** as they will not have a block without movement.

6.16.3. Cancellation of tool radius compensation

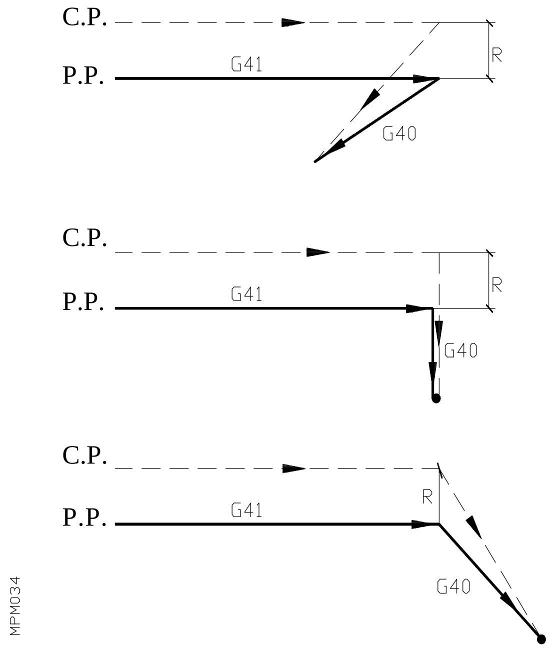
Tool radius compensation is cancelled by function **G40**.

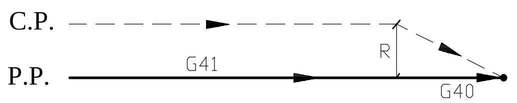
It should be borne in mind that tool radius compensation cancellation (G40) can only be carried out in a block in which a rectilinear movement is programmed (G00,G01).

If **G40** is programmed in a block containing **G02** or **G03**, the CNC will give alarm **40**.

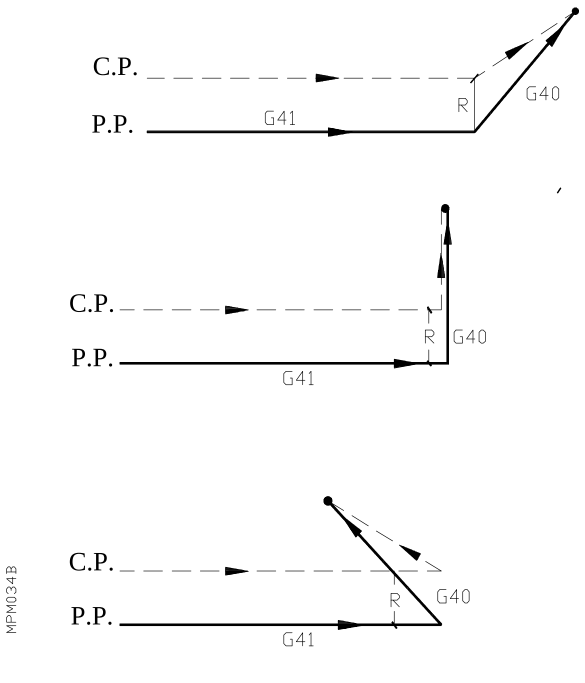
The following is a table of various cases of cancellation of compensation.

STRAIGHT PATH

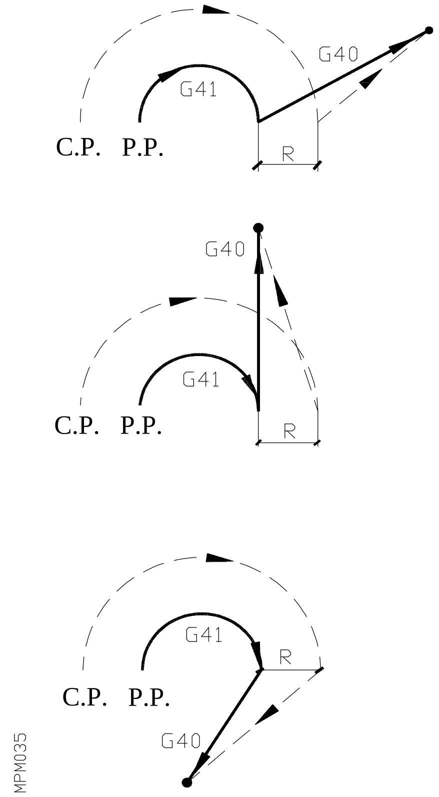


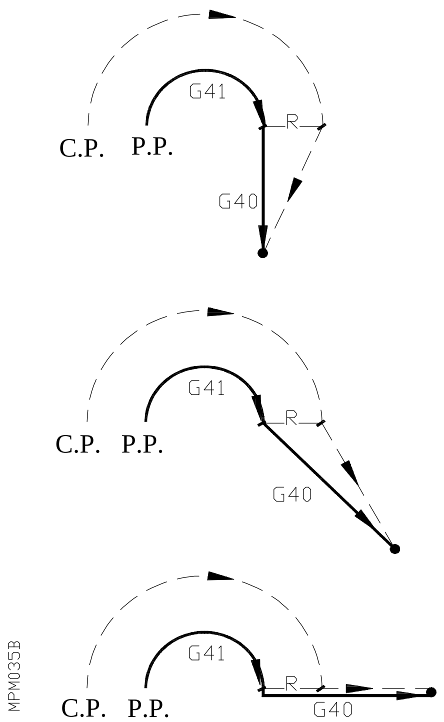


(Path programmed in 2 blocks)

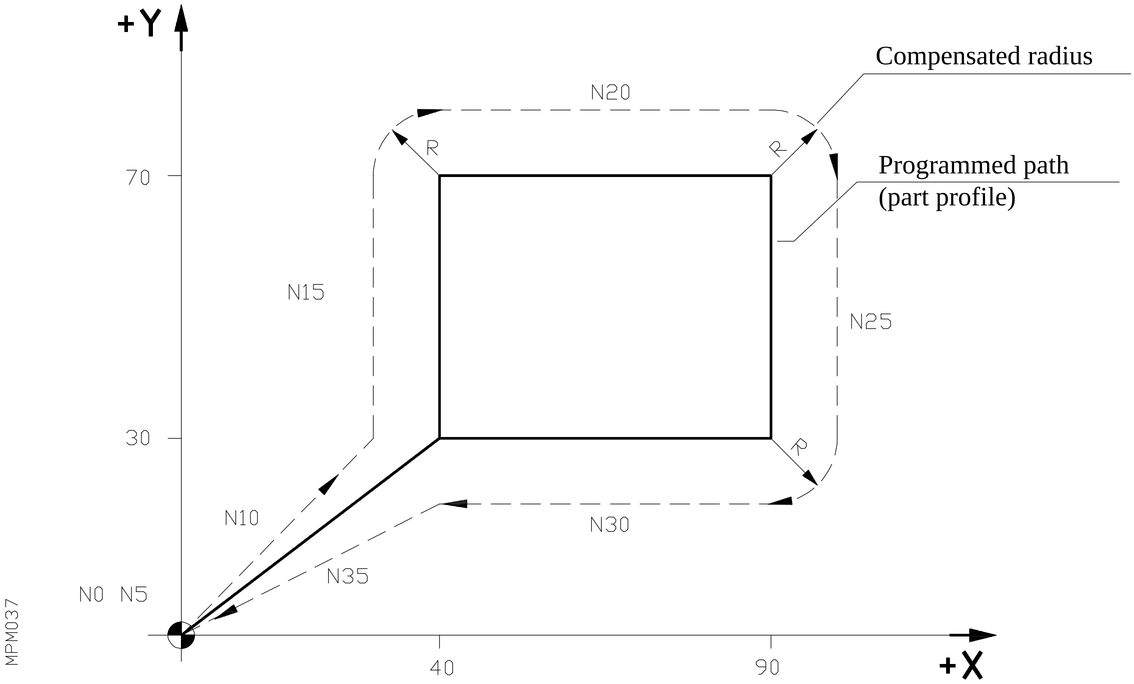


CURVE-STRAIGHT PATH





Example of machining with tool radius compensation

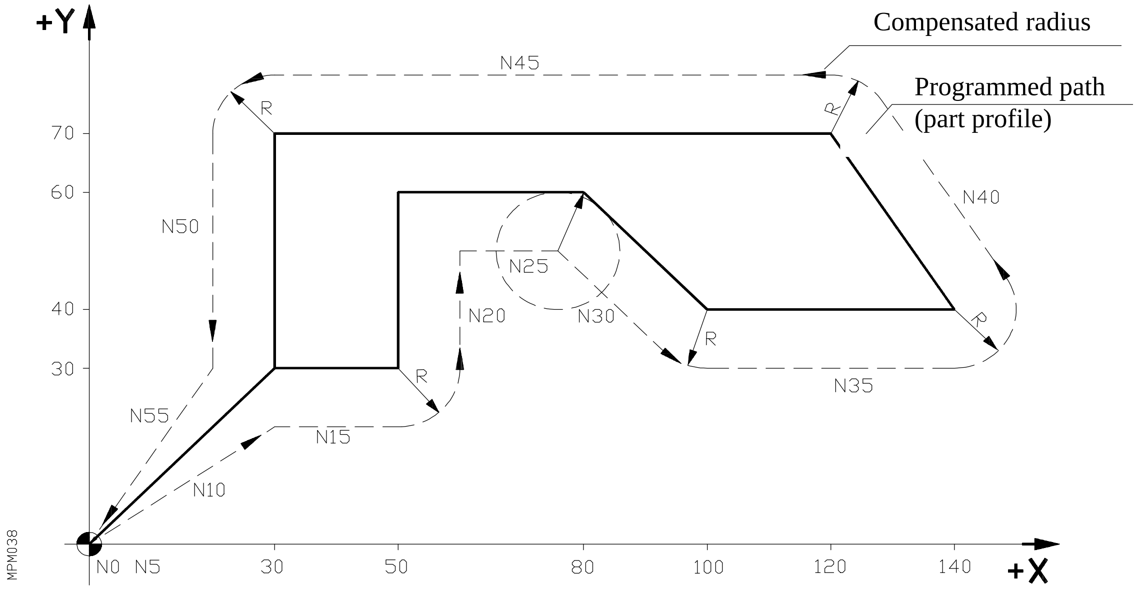


Tool radius : 10 mm.
Tool number : T1.1

It is assumed that there are no movements on the **Z** axis.

```
N0   G92  X0  Y0  Z0
N5   G90  G17 S100  T1.1  M03
N10  G41  G01  X40 Y30 F125
N15  Y70
N20  X90
N25  Y30
N30  X40
N35  G40  G00  X0  Y0  M30
```

Example of machining with tool radius compensation

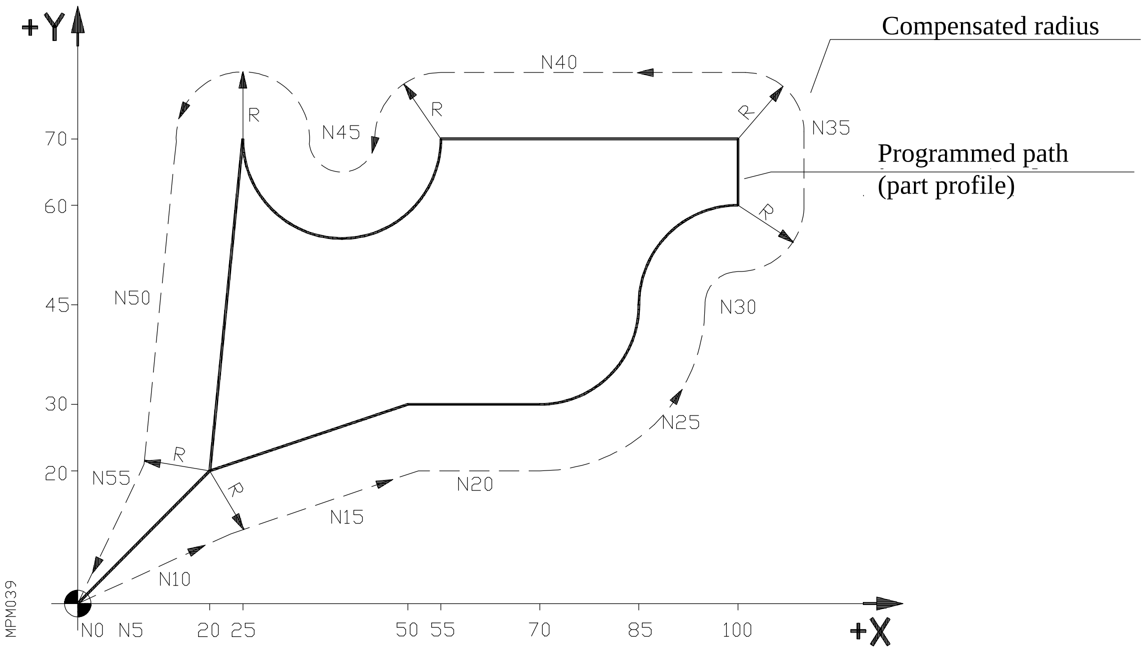


Tool radius : 10 mm.
Tool number : T1.1

It is assumed that there are no movements on the **Z** axis.

```
N0    G92   X0   Y0   Z0
N5    G90  G17  G01 F150  S100  T1.1  M03
N10  G42  X30  Y30
N15  X50
N20  Y60
N25  X80
N30  X100 Y40
N35  X140
N40  X120 Y70
N45  X30
N50  Y30
N55  G40 G00 X0 Y0  M30
```

Example of machining with tool radius compensation



Tool radius : 10 mm.
Tool number : T1.1

It is assumed that there are no movements on the **Z** axis.

```
N0    G92   X0     Y0   Z0
N5    G90   G01    G17 F150  S100 T1.1 M03
N10  G42   X20   Y20
N15  X50  Y30
N20  X70
N25  G03   X85   Y45 I0 J15
N30  G02   X100 Y60 I15  J0
N35  G01  Y70
N40  X55
N45  G02   X25 Y70 I-15 J0
N50  G01   X20 Y20
N55  G40  G00  X0    Y0 M05  M30
```

6.17. TOOL LENGTH COMPENSATION

This function makes it possible to compensate for possible differences in length between the tool programmed and the tool to be used.

As previously indicated in the section on tool radius compensation, the CNC has storage capacity for dimensions, radius and length of 100 tools (Txx.oo- Txx.99).

L identifies the tool length and **K** the tool wear. The CNC will add (or subtract) the value of **K** to the value of **L**.

The maximum length compensation values are:

L+/-1000 mm or +/-39.3699 inches K+/-32.766 mm (1.2900 inches)

The call codes for length compensation are:

G43 : Length compensation
G44 : Cancellation of length compensation

When **G43** is programmed, the CNC compensates the length according to the value selected from the tool table (Txx.00-Txx.99).

Length compensation is applied to the axis perpendicular to the principal plane.

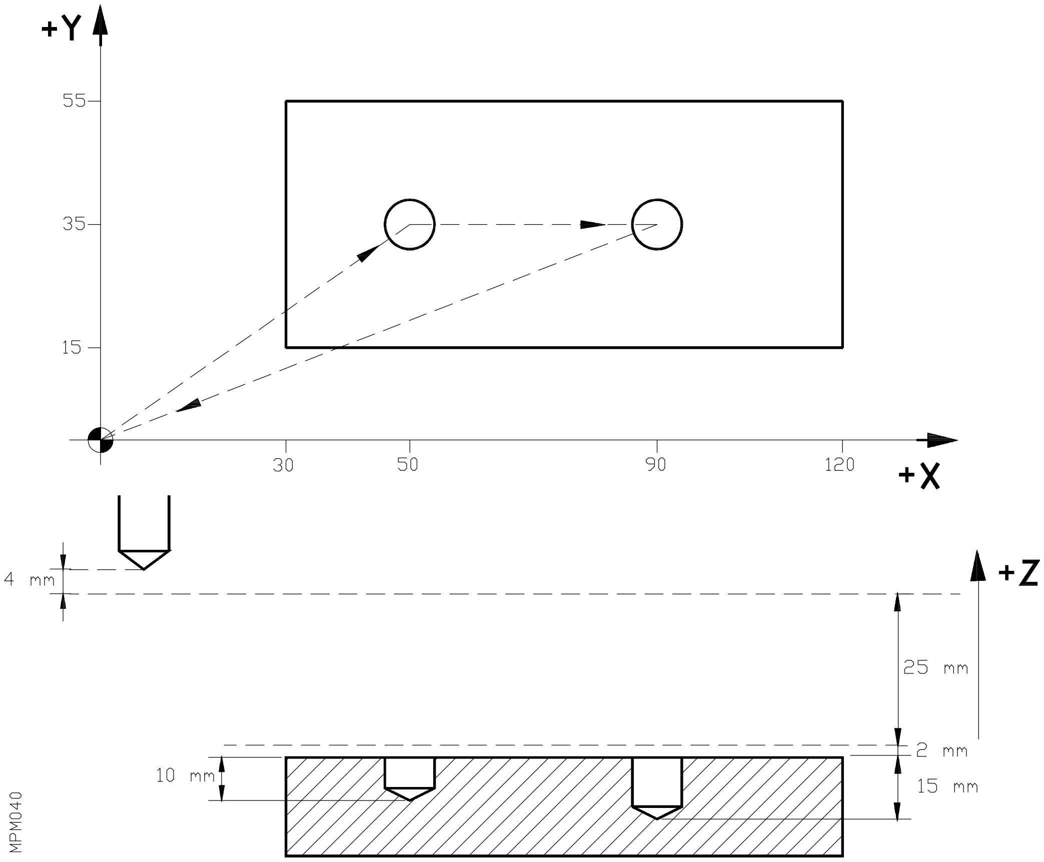
G17 : Length compensation on the Z axis
G18 : Length compensation on the Y axis
G19 : Length compensation on the X axis

The tool length compensation will be applied on the **4th axis (W)** or to the **5th axis (V)** when appropriate, i.e. when appropriate, i.e. when the axis linked to it is not on the main plane.

Function **G43** is modal (persistent) and is cancelled by **G44**, **G74**, **M02**, **M30**, **EMERGENCY** or **RESET**.

Length compensation can be used in conjunction with canned cycles, although in that case the precaution has to be taken of applying the compensation before the cycle starts.

Example of tool length compensation



It is supposed that the tool used is 4 mm shorter than the tool programmed.

The tool number is **T1.1** (the value recorded in the tool table is L-4).

```
N0    G92  X0   Y0   Z0
N5    G91  G00  G05  X50  Y35   S500  M03
N10  G43  Z-25  T1.1
N15  G01  G07  Z-12  F100
N20  G00  Z12
N25   X40
N30  G01 Z-17
N35  G00  G05  G44 Z42 M05
N40  G90  G07  X0  Y0
N45  M30
```

6.18. G47 - SINGLE BLOCK TREATMENT
G48 - CANCELLATION OF SINGLE BLOCK TREATMENT

As of the execution of function **G47**, the CNC executes all the blocks which come next as if it were a single block. This single block treatment is carried out until it is cancelled by means of the **G48** function. In this way, with the **G47** function active in the **SINGLE BLOCK** operation, these will be executed in continuous cycle until the **G48** function is executed, i.e., the execution will not stop when a block is finished but will continue by executing the following one.

In any operating mode, if execution is interrupted when the **G47** function is active, the CNC stops axis feed as well as the spindle. It will also stop axis feed when the **FEED HOLD** input is activated, as long as machineparameter P610(1)=1.

With the **G47** function active, the M.F.O. switch and the spindle speed variation keys will be disenabled, the program being executed at 100% of the programmed **F** and **S**.

The **G47** and **G48** functions are **MODAL**. When the CNC is switched on, after executing **MO2**, **M30**, **Reset** or **Emergency**, the CNC assumes the **G48** function.

6.19. G49. PROGRAMMABLE FEEDRATE OVERRIDE

With **G49** the programmed working feedrate **F** can be overridden.

The feedrate override Knob on the front panel will have no effect.

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

1/120 meaning the percentage value between 1% and 120% of the previously programmed **F** value.

Function **G49** is modal, so it will remain active until another value is programmed or is cancelled by programming: **G49 K0** or simply: **G49**.

G49 will also be cancelled when **M02**, **M30**, **RESET** or **EMERGENCY** are executed.

G49 K must be programmed alone in a block.

6.20. G50. Loading of the values in the tool offset table

The different tool values can be entered in the table by using **G50**. There are two possibilities:

a) Entering of all the values.

By means of the block N4 G50 T2 R+/-4.3 L+/-4.3 I+/-2.3 K+/-2.3 (mm) R+/-2.4 L+/-2.4 I+/-1.4 K+/-1.4 (inches). The values defined by R,L,I,K are loaded in the tool offset table direction identified by T2.

N4	- Block number
G50	- Tool offsets loading code
T2(T00-T99)	- Tool offset table direction
R+/-4.3 (R+/-2.4)	- Tool radius
I+/-2.3 (I+/-1.4)	- Tool wear offset (radius)
L+/-4.3 (L+/-2.4)	- Tool length
K+/-2.3 (K+/-1.4)	- Tool wear offset (length)

The values of **R**,**L**,**I**,**K** replace the values previously existing in the **T2** direction. If **R** and **L** are programmed and **I**, **K** are not, they are replaced in the table with the values of **R** and **L** by the new programmed values and the correction values **I**, **K** are zeroed.

b) Incremental modification of the I,K values.

By means of the block: N4 G50 T2 I+/-2.3 K+/-2.3 (mm) or N4 G50 T2 I+/-1.4 K+/- 1.4 (inches) the I,K values of the T2 address are modified.

N4	- Block number
T2(T01-T99	- Tool offset table address
I+/-2.3 (I+/-1.4)	- Value to be added to or subtracted from the I value previously recorded.
K+/-2.3 (K+/-1.4)	- Value to be added to or subtracted from the K value previously recorded.

In mode a) the tool offset table may be loaded without having to enter the values manually in operation mode **8**. Mode b) allows the compensation of tool wear. The radius compensation value will be **R+I**.

The length compensation value will be **L+K**.

No other information can be programmed in the block containing **G50**.

6.21. G52. COMMUNICATION WITH THE FAGOR LOCAL AREA NETWORK

The communication between the CNC and the rest of the **LAN NODES** is carried out thru registers in complement to two.

These registers may be double (D) or single (R). Next, the different command formats are described.

a) Transfer of a constant to a register of another LAN NODE.

G52 N2 R3 K5

or:

G52 N2 D3 H8

G52 : Communication with the LAN.
N2 : Address of the DESTINATION NODE (0/14).
R3 : Number of the single register (0/255).
D3 : Number of the double register (0/254).
K5 : Integer value in decimal (+/-32767).
H8 : Integer value in Hexadecimal (0/FFFFFFFF).

Atention:

To access a PLCI register, indicate the number of the node occupied by the CNC+PLCI

b) Transfer of a value of an ARITHMETIC PARAMETER of the CNC to a register of another LAN NODE.

G52 N2 R3 P3

or,

G52 N2 D3 P3

G52 : Communication with the LAN
N2 : Address of the DESTINATION node (0/14).
R3 : Number of the single register (0/255).
D3 : Number of the double register (0/254).

P3 : Number of the arithmetic parameter (0/254).

Atention:

To access a PLCI register, indicate the number of the node occupied by the CNC+PLCI

c) Loading the value of a register of another LAN NODE into an arithmetic parameter of the CNC.

G52 N2 P3 R3

or,

G52 N2 P3 D3

G52 : Communication with the LAN.
N2 : Address of the ORIGIN node (0/14).
P3 : Number of the arithmetic parameter (0/254).
R3 : Number of the single register (0/255).
D3 : Number of the double register (0/254).

Atention:

To access a PLCI register, indicate the number of the node occupied by the CNC+PLCI

d) Sending a text from the CNC to another LAN NODE.

G52 N2 = (TEXT)

G52 : Communication with the LAN
N2 : Address of the DESTINATION node (0/14).
( ) : Text delimiters.
Text : Text whose syntax is admitted by the DESTINATION node.

Example:

Let us suppose that the **NODE** 7 of the **LAN** is a **FAGOR CNC 82** connected as slave and its **X** and **Y** axes are to be positioned at the X100, Y50 point. The block to be executed by the CNC will be:

G52 N7 = (X100 Y50)

e) Process synchronization between LAN NODES.

G52 N2

This block will be completed when the LAN NODE N2 has ended the execution of the current operation. By using this type of blocks, the different operations of several LAN nodes can be synchronized.

Atention:

Any error at the FAGOR LAN occurring during the execution, the CNC will display error 111.

More information on the FAGOR LOCAL AREA NETWORK is found in the INSTALLATION AND START-UP MANUAL, chapter INCORPORATION OF THE 8025/30 CNC into the FAGOR LOCAL AREA NETWORK.

6.22. G53-G59 ZERO OFFSETS

7 different zero offsets can be selected by functions G53,G54,G55,G56,G57,G58 and G59. The values of these offsets are stored in the CNC memory after the tool dimensions table and are referred to the **machine reference zero**. The values can be entered in operation mode 8 via the keyboard or by program, using codes **G53- G59**.

To display the **G53-G59** table press **OP MODE**, then **8** and finally **G**.

Operation of **G53-G59**, these functions can be used in two different ways:

Format a) **To load the zero offset table .**

. Absolute loading of the values

Using a block like N4 G5? V+/-4.3 W+/-4.3) X+/-4.3 Y+/- 4.3 Z+/-4.3 (metric) or N4 G5? V+/-3.4 W+/-3.4) X+/-3.4 Y+/-3.4 Z+/-3.4 (inches) the values identified by (W),X,Y,Z are loaded in the table address defined by G5? (G53-G59).

N4	: Block number
G5?	: Offset code (G53,G54,G55,G56,G57,G58,G59).
V+/-4.3 V+/-3.4	: Zero offset value referred to the machine reference zero on the V axis.
W+/-4.3 W+/-3.4	: Zero offset value referred to the machine reference zero on the W axis.
X+/-4.3 X+/-3.4	: Zero offset value referred to the machine reference zero on the X axis.
Y+/-4.3 Y+/-3.4	: Zero offset value referred to the machine reference zero on the Y axis.
Z+/-4.3 Z+/-3.4	: Zero offset value referred to the machine reference zero on the Z axis.

. Incremental loading of the values

Block N4 G5? (H+/-4.3) L+/-4.3 H+/-4.3 I+/-4.3 J+/-4.3K+/-4.3 in mm or N4 G5? L+/ -3.4 H+/-3.4 I+/-3.4 J+/-3.4 K+/-3.4 in inches, increments by an amount H, I, J, K, the table values indicated by G5? (G53-G59).

N4 : Block number

G5? : Zero offset code (G53,G54,G55,G56,G57,G58,G59).

L+/-4.3 : Amount added or subtracted to the V
L+/-3.4 value previously stored in the table.

H+/-4.3 : Amount added or subtracted to the W
H+/-3.4 value previously stored in the table.

I+/-4.3 : Amount added or subtracted to the X
I+/-3.4 value previously stored in the table.

J+/-4.3 : Amount added or subtracted to the Y
J+/-3.4 value previously stored in the table.

K+/-4.3 : Amount added or subtracted to the Z
K+/-3.4 value previously stored in the table.

Format b) To apply a zero offset to the current program.

According to the value assigned to the machine parameter P619(7) there are two cases:

Case 1: P619(7) = 0

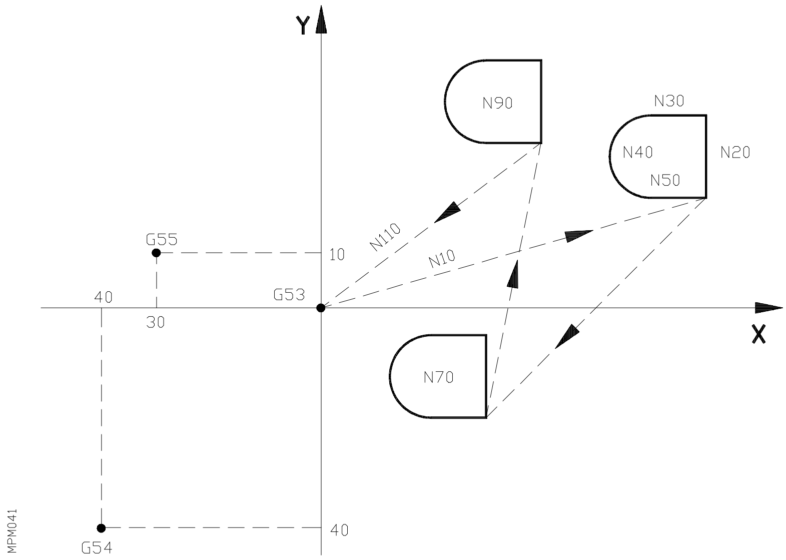
A block like N4 G5? is used to carry out a zero offset on the current program, according to the values stored in the G5? position of the zero offset table (G53-G59).

N4 : Block number
G5? (G53,G54,G55,G56,G57,G58,G59): Direction of the table in which the two zero setting values are stored.

Case 2: P619(7) = 1

When a function of the G54 .... G58 type is executed, the zero setting applied to each axis will be the value indicated in the table (G54 .... G58) plus the value indicated in position G59 of the table. G53 is not affected.

Example:



The values:

G53 X0 Y0
G54 X-40 Y-40
G55 X-30 Y10

are entered in the G53-G59 table. The starting point is X0 Y0

```
N10   G00  G90  X70  Y20
N20   G01  Y35  F200
N30   X60
N40   G03  X60  Y20  I0  J-7,5
N50   G01  X70  Y20
N60   G54
N70   G25  N10.50.1
N80   G55
N90   G25  N10.50.1
N100 G53
N110 X0 Y0
N120 M30
```

6.22.1. G59 as additive zero offset

If P619(7)=1 When any function of the G54... G59 type is executed, the zero offset applied to each axis will be the value indicated on the table (G54 ... G59) plus the value indicated in position G59 of the table. It does no affect G53.

If P619(7)=0 In this case, the zero offset which is applied to each axis will be the value indicated on the table.

6.23 FUNCTION "G64". MULTIPLE ARC PATTERN MACHINING CYCLE

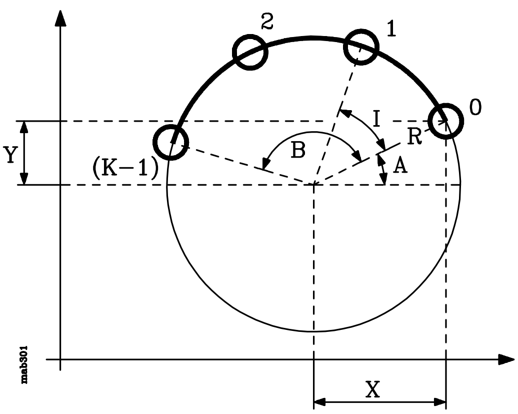
By means of this function, it is possible to perform circular movements.

This way, if a canned cycle is active when defining this function, the CNC will carry out the programmed movements and it will execute the canned cycle at each new position.

Therefore, it is possible to perform drilling, threading operations, etc. in an arc pattern.

The programming format for this cycle is as follows:

```
G64 | X  Y | B  I | C F Q U
    | R  A | B  K |
           | K  I |
```



The CNC takes as starting point, the one used to define the multiple machining cycle. The center of the arc may be defined in cartesian (XY) or polar coordinates (RA)

X Defines the distance from the starting point to the center along the abscissa axis.

Y 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 it in circular interpolations (G02, G03).

R Defines the distance from the starting point to the center.

A Defines the angle of the line joining the starting point and the center with respect to the X axis.

The machining positions are set by combining 2 of the parameters "B, I and K".

B Defines the total angular travel and it is given in degrees.

I Defines the angular step between machining operations.

K Defines the total number of machining operations along the arc including the cycle defining point.

It must be borne in mind that the cycle defining point has already been machined.

C Indicates the type of movement between the machining positions. If not programmed, a value of C=0 will be assumed.

C=0 Movement in rapid (G00).
C=1 Linear interpolation (G01).
C=2 Clockwise circular interpolation (G02).
C=3 Counter-clockwise circular interpolation (G03).

When setting C=0 or C=1, the sign of parameters "B, I and K" indicate the direction of the movement: "+" counter-clockwise, "-" clockwise.

When defining "**B I**" the moving direction is set by the sign of "**I**".
When defining "**B K**" the moving direction is set by the sign of "**B**".
When defining "**K I**" the moving direction is set by the sign of "**I**".

F Defines the moving feedrate from point to point. Obviously, it will only be relevant for "C" values other than "0". If not programmed, a value of F0 will be assumed which is set by machine parameters "P110" and "P210".

Q, U These parameters are optional and are used to indicate which points or between which points the machining operation is **not** to be performed.

Thus, when programming Q7, it indicates that point 7 is not to be machined and when programming Q10.013, it indicates that points 10 thru 13 are not be machined (that is, points 10, 11, 12 and 13).

When defining a group of points (Q10.013) it must be borne in mind that the last point has to be indicated with three digits since programming "Q10.13" will be interpreted as points 10 thru 130. (Q10.130).

The programming order for these parameters is Q U and the numbering order assigned to them must also be kept. That is, the numbering of the points assigned to Q must be smaller that the ones assigned to U.

Example:	Correct programming	Q12.015 U20.022
	Incorrect programming	Q20.022 U12.015

If these parameters are not programmed, the CNC interprets that the machining operations must take place on all points of the programmed path.

Basic operation:

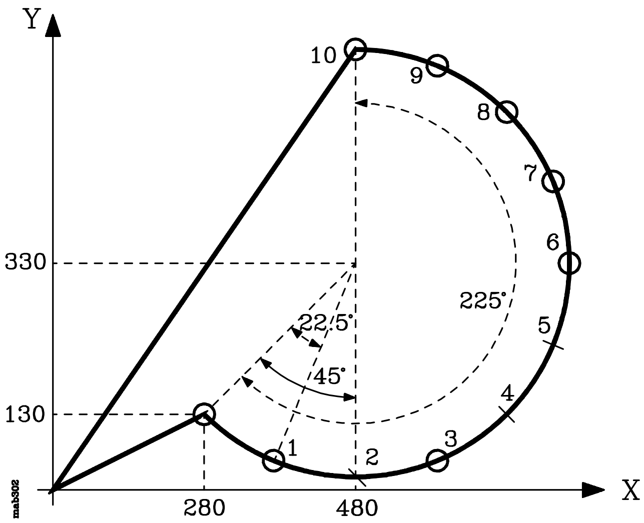
1.- This cycle calculates the next programmed point to be machined.

2.- Movement to that point at the feedrate programmed by "C" (G00, G01, G02 or G03).

3.- Once at this new point, it will execute the selected canned cycle.

4.- The CNC will repeat steps 1, 2 and 3 until the end of the programmed path. Once the multiple machining cycle is completed, the tool will be positioned at the last machined point of the programmed path.

Programming example assuming point X0 Y0 Z0 as the starting point:




```
G81 G98 G01 G91 X280 Y130 Z-8 I-22 F100      ; Positioning and canned cycle definition
G64 X200 Y200 B225 I22.5 C3 F200 Q2 U4.005   ; Multiple machining cycle definition
G80                                          ; Canned cycle cancellation
G90 X0 Y0                                    ; Positioning
M30                                          ; End of program
```

It is also possible to write the multiple machining cycle defining block as follows

G64 **R282.843 A45** B225 I45 C3 F200 Q2 U4.005
G64 X200 Y200 **B225 K11** C3 F200 Q2 U4.005
G64 X200 Y200 **K22.5 I11** C3 F200 Q2 U4.005

6.24. G65. INDEPENDENT AXIS EXECUTION

With function G65 it is possible to move one axis independently while other axes are being interpolated.

In the following program:

```
N0   G65 W100 F1
N10  G01 X10 Y10 Z5 F1000
N20  G01 X20
```

When executing block "N0", the W axis starts moving at a feedrate of F1. Then, block "N10" starts executing the XYZ interpolation at F1000 while the W axis keeps moving at F1.

If **"P621(4)=0"**, the CNC executes block "N20" once "N10" is completed regardless of whether "N0" is completed or not (W axis has reached position or not).

If **"P621(4)=1"**, the CNC waits until blocks "N0" and "N10" are completed (all axes have reached position) before executing block "N20".

6.25. G70/G71. UNITS OF MEASUREMENT

G70 : Programming in inches
G71 : Programming in millimeters

Depending on whether **G70** or **G71** is programmed, the CNC takes the subsequent coordinates as being in inches or millimeters respectively.

Functions **G70/G71** are modal and incompatible with one another.

The CNC assumes the units set by parameter **P13** when being turned on, after **M02,M30**, **EMERGENCY** or **RESET**.

6.26. G72. SCALING FACTOR

G72 allows the machining of parts of similar shape but different size using the same program.

G72 must be programmed alone in a block. There are two different methods of programming **G72**.

6.26.1. Method a). Scaling factor to affect all axes

Format :

N4 G72 K2.4

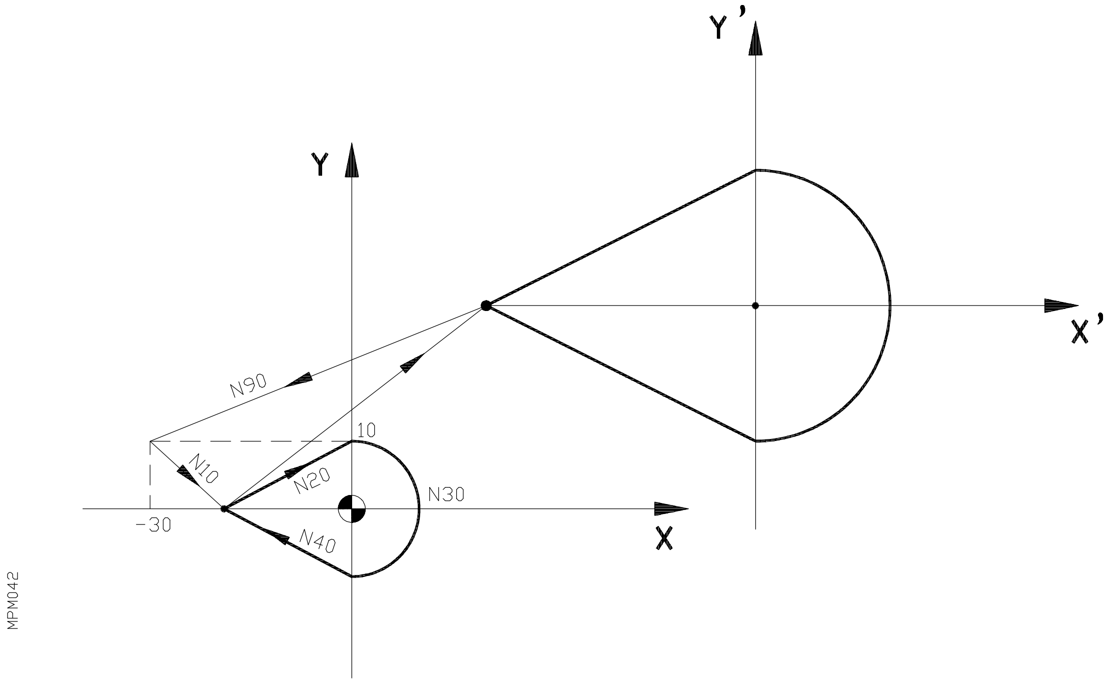
N4 - Block number
G72 - Scaling code
K2.4 - Value of the scaling factor

Min. value K0.0001 (x 0.0001)
Max. value K100 (x 100)

Tool radius and length compensation are compatible with this scaling mode.

All coordinate values programmed after **G72** will be multiplied by **K** until the scaling is cancelled by K=1, or after **M02,M30**, **EMERGENCY** or **RESET**.

Example:



Starting point is X-30 Y10

N10 G00 G90 X-19 Y0
N20 G01 X0 Y10 F150
N30 G02 X0 Y-10 I0 J-10
N40 G01 -19 Y0
N45 G31.. (Store datum point)
N50 G92 X-79 Y-30 .... (Change datum point)
N60 G72 K2 .... (Apply Scaling factor 2)
N70 G25 N10.40.1
N80 G72 K1.. (Cancel scaling factor)
N85 G32 .... (Retrieve original datum point)
N90 G0 X-30 Y10 .... (Return to initial point)
N100 M30 .... (End of program)

6.26.2. Method b). Scaling factor affecting one axis only

Format:

N4 G72 V,W,X,Y,Z 2.4

N4	: Block number
G72	: Function which defines the scale factor
V,W,X,Y,Z	: Axis to which the scale factor is applied.
2.4	: Scaling factor value

Min. value 0.0001 Max. value 15.9999

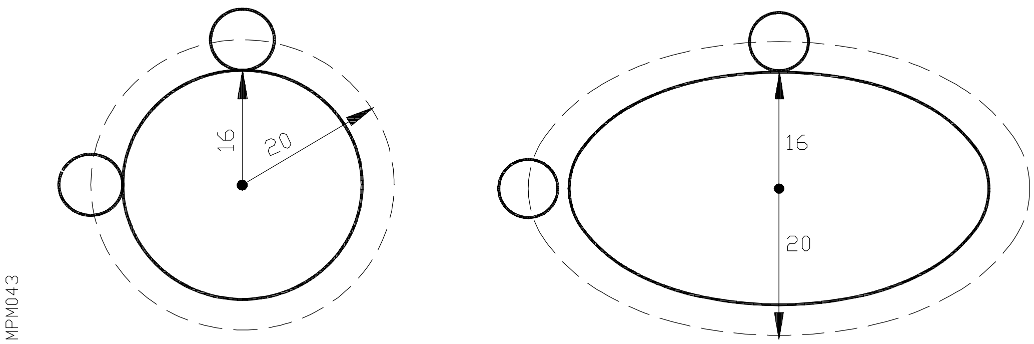
In this case the axis to which the scale factor is applied must be at the zero point (value 0) when the factor is applied or cancelled.

The coordinate value of the axis affected must be zero when **G7**2 is applied. When the scaling factor affects only one axis, the coordinate values of the datum point cannot be altered by functions such as **G3**2, **G9**2 or **G5**3 thru **G5**9.

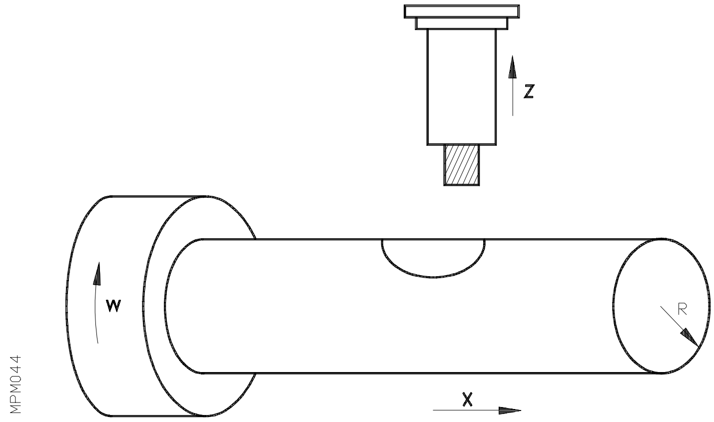
The scaling is cancelled by factor 1, after **M0**2**,M3**0, **EMERGENC**Y or **RESE**T. If an scaling factor is entered, any preceding factor is automatically cancelled, regardless of the axis affected.

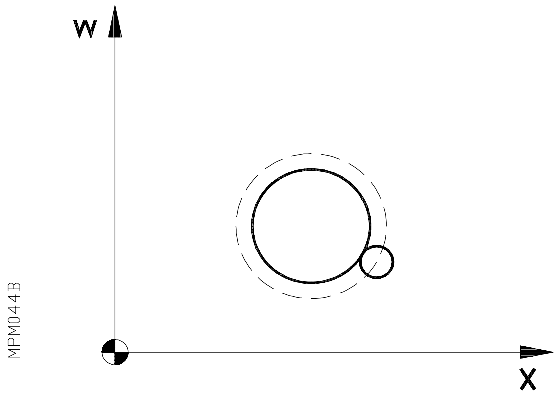
Tool length compensation is compatible with this scaling mode.

Tool radius compensation can only be used when the scaling factor is applied to a rotary axis. If the axis is linear the radius compensation will be affected by the scaling factor applied to the axis.



Machining on a cylindrical surface. If a scaling factor equal to $\frac{360}{2\pi R}$ R being the cylinder's radius) is applied to a rotary axis, it can be handled as a linear axis. Thus any path can be programmed on the cylinder's surface, with tool radius compensation.





If, within the same program, both scaling methods are used, the CNC will apply to the axis affected by method b) a factor equal to the multiplication of both values.

When checking a program in **DRY RUN** execution modes 0**,1** o**r 4** the coordinates values and the graphic displayed are not affected by any method b) scaling factor i.e. when the scaling factor is applied to one axis only the values and graphics displayed will represent the programmed values.

6.27. G73. PATTERN ROTATION

This feature allows the rotation of the coordinate axes around the part program's datum point on the main plane.

Format:

N4 G73 A+/-3.3

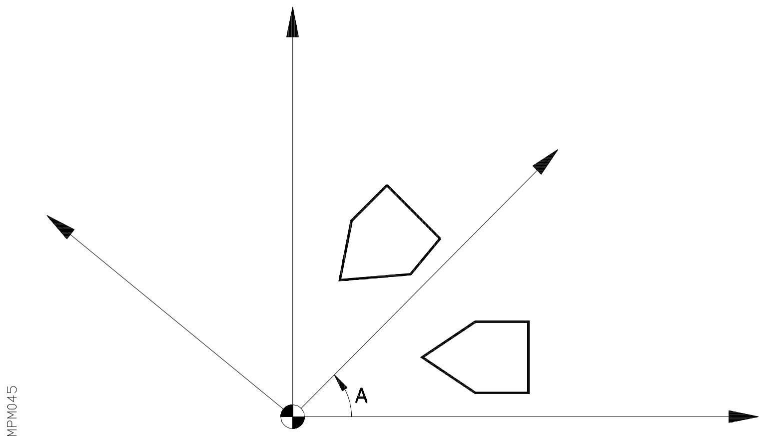
N4 - Block number
G73 - Pattern rotation code
A+/-3.3 - Rotation angle

Min. value - 0.000 degrees
Max. value - 360.000 degrees

G73 is incremental, i.e., if more than one **G7**3 is programmed. Their respective A values will be added together.

G73 must be programmed alone in a block.

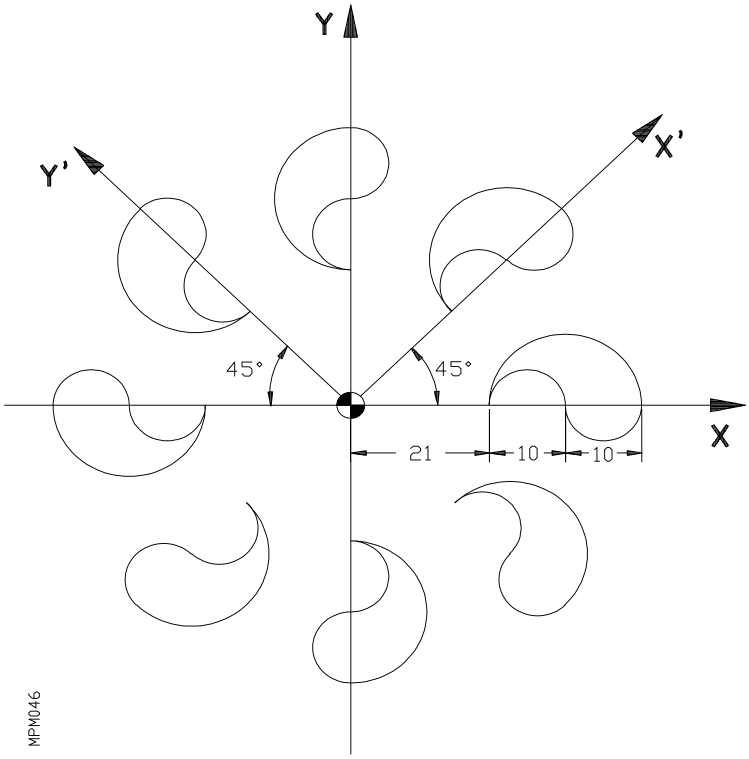
Pattern rotation is cancelled with: **G1**7**, G1**8**, G1**9**, G73** (without A value). An **M0**2**, M3**0**, EMERGENC**Y or **RESE**T.



When the part is programmed in absolute (G90) and cartesian coordinates all the points have to be identified by both coordinate values on the main plane, even when this requires repeating some values, and moreover a point cannot be defined by one angle plus a cartesian coordinate.

Example:

Starting point is X0 Y0 and the path of the tool is programmed in the **X**Y plane without taking its dimensions into consideration.



N10 G01 X21 Y0 F300
N20 G02 A0 I5 J0
N30 G03 A0 I5 J0
N40 A180 I-10 J0
N50 G73 A45
N60 G25 N10.50.7
N70 M30

In four-axis machines the rotation can also be applied to a plane including th**e fourth axis (W**) if it is linear and active when **G7**3 is programmed.

If the axis linked to th**e fourth (W**) is programmed afterwards the rotation function will be cancelled.

This same treatment is given in 5 axis machines when th**e 5th axis** V is one of the axes which make up the main plane.

6.28. G74. MACHINE REFERENCE SEARCH

When **G74** is programmed in a block, the CNC moves the axes to the **machine-reference** point.

a) REFERENCE SEARCH FOR ALL AXES

* If machine parameter P725 = 0 and **G74** is programmed alone in the block. The CNC moves first the axis which is perpendicular to the plane programmed.

  This is Z if operating on **G17**, Y on **G18** **,X** on **G19**. The movements of the remaining axes then follow.

  In **four-axis** machines, if the active axis when programming **G74**, is the one associated to the **fourth (W)**, the axes will be moved as described, (W) being the last. However, if the active axis is the **fourth (W)** when **G74** is programmed, it will replace its associated axis in the prescribed order and the associated axis will be the last on moving.

  In **5** axis machines the movement of the **5th axis** V, when **G74** only **G74** is programmed, this is always done after the movement of the **4th axis** W.

* If P725 has a value between 1 and **99** and **G74** is programmed alone in the block, the CNC will automatically execute the subroutine whose number is in P725.

  If P725 = 74, subroutine **74** will be executed. For example:

  N0 G22 N74
  N10 G74 X Y W V (this might be the order required by the user)
  N20 G74

b) REFERENCE SEARCH FOR ONE AXIS OR MORE THAN ONE IN A SPECIFIC ORDER

If machine-reference search is required in an order other than the above, **G74** is programmed, followed by the axes in the required order.

No other function can be programmed in a block in which **G74** is programmed.

In both cases, a) and b), when the axes reach the **machine-reference**, the distances between this point and the last part's datum point programmed are displayed.

6.29. PROBES

6.29.1. Definition

Probes are basically simple switches provided with a high level of sensitivity.

When the probe touches a surface, a signal is sent to the CNC of the machine, and the position of the axes are automatically recorded. In the case of machine tool applications, this same signal acts on the control of the machine until an adequate, precise and rapid positioning of the tool or part is obtained.

Probes do not measure, they simply send positioning signals to be treated in the CNC of the machine and according to specific tolerances.

6.29.2. Characteristics

Probes are of modular construction for better adaptation to the needs of the user. This system consists of a stylus, probe, transmission system and interface. The stylus is the part which enters into contact with the surface.

They are provided with a system to absorb impact with the surface.

The tip of the probe includes the stylus.They are of solid and compact construction in order to protect the stylus. Different extension modules can be fitted in order to obtain the right geometrical configuration for each application.

Probes have three different systems:

- **Cabling**
- **Inductive**
- **Optical**

Cabling: The signal is transmitted through the cable. Its most important disadvantage is its rigidness in moment. It is used in lathes and machining centers for final adjustments of tools where the probe has a fixed measuring position and the tools are brought close to the probes. It is also used in digitizing systems.

Inductive: It allows greater ease of movement. The signal is transmitted without physical contact, by means of two opposing plates, across the work area.

Optical: Communication is made by means of infrared rays. One of its advantages is the freedom to position the signal receiver outside the work area. Its applications are the same as those of the inductive probe.

6.29.3. Most common applications

There are different applications, as shown below:

Fine adjustment of the tool: These check the cutting point of each tool and compensate, if necessary, the distance to the work place or stop production should a tool break.

Fine adjustment of the part: by means of canned probecycles which will be seen below.

Digitizing system. For copying parts by means of the collection of information point by point. The probe is given the job of sending positional data by means of a series of predetermined movements along the surface of the part.

In the case of the **FAGOR 8025/30 MS CNC** the system generates CNC programs automatically enabling the machining of complex parts with a great deal of reliability.

It is recommendable to use an **interface** which is an electronic link between the probe and the control of the machine.

This controls the status of the probe continuously, provides energy to the induction modules and transmits a signal to the control of the machine when the probe has tripped.

6.29.4. G75. PROBING

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

Format:

N4 G75 (V+/-4.3) (W+/-4.3) X+/-4.3 Y+/-4.3 Z+/-4.3

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

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

If the axes arrive in position before the probe signal is received, the CNC will act as follows:

> If machine parameter "P621(6)=0", the CNC interrupts the program execution and it issues error 65. But, if "P621(6)=1", the CNC will consider the block completed and it will go on to execute the next block.

After executing this block, the values of the different axes can be allocated to the desired parameters. The combination of this feature with mathematical operations with parameters permits creating special subroutines to measure parts or tools.

G75 implies **G01** i.e. the CNC assumes function **G01** and **G40** after a **G75** block.

<u>Atention:</u>

This CNC allows manually measuring the length of the tools with a probe. See the OPERATING MANUAL.

6.29.5. G75 N2. Probing canned cycles

The **MS** model CNC offers various probing canned cycles to accomplish the following:

. Measure the tool dimensions. Position the tool at a specific point on the part before machining it.

. Measure the part after it has been machined

The programming format is as follows:

G75 N P0 = K.. P1 = K..**

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

The CNC's probing canned cycles are:

N00 : Tool length calibration.
N01 : Probe calibration.
N02 : Surface measuring.
N03 : Surface measuring with tool offset.
N04 : Outside edge measuring.
N05 : Inside edge measuring.
N06 : Angle measuring.
N07 : Edge and angle measuring.
N08 : Hole centering.
NO9: Boss centering.
N10 : Hole measuring.
N11 : Pocket measuring.

After the figures defined in the cycle (**N****), the calling parameters (**P?=K?**) necessary for each cycle must be programmed. The calling parameters used in the cycles are the following?

P0 : Theoretical **X** value.

P1 : Theoretical **Y** value.

P2 : Theoretical **Z** value.

P3 : Safety distance.

P4 : Probing feedrate.

P5 : Tolerance.

P6 : Table number of the tool to be calibrated.

P7 : Axis being probed.

P7=0 X Axis,
P7=1 Y Axis,
P7=2 Z Axis.

P8 : Hole's or pocket's theoretical diameter.

P9 : Initial probing feedrate for cycles: N01, N08, N09, N10, N11.

P10: Withdrawal distance after cycles: N01, N08, N09, N10, N11.

P11: Tool calibration on:

. the axis P11 = 0
. one end P11 = 1

GENERAL CONSIDERATIONS

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

. Parameters P4 and P9 relevant to the probing feedrate must be programmed in mm/min. or 0.1 inch/min.

. Parameter P3 must be greater than zero.

. Parameter P5 must be equal or greater than zero.

. Parameter P7 can only be 0, 1 or 2. Parameter P11 can only be 0 or 1.

Error 3 will be issued if one of the last four conditions are not met.

BASIC OPERATION

Once the probe is positioned near the surface to be probed, the movements of the axes during a probing cycle are:

<u>Approach</u>

It is executed in rapid mode **G00** from the starting point of the cycle to a safety distance P3 away from the theoretical value.

<u>Probing</u>

It is executed at a feedrate determined by **P4** until the CNC receives the probe signal.

If before moving a maximum distance, which depends on the type of cycle selected, the CNC has not received the probe signal, the CNC will act as follows:

> If machine parameter "P621(6)=0", the CNC interrupts the program execution and it issues error 65. But, if "P621(6)=1", the CNC will consider the block completed and it will go on to execute the next block.
>
> In order to simplify the explanation of the canned cycles, machine parameter "P621(6) is assumed to be set to "0".

While probing, the **Feedrate override** knob will have no effect on the feedrate which will be fixed at 100%

<u>Withdrawal</u>

Once the probing corresponding to selected cycle is finished, the axes will withdraw, in rapid move **G00**, back to the starting point.

Depending on the selected cycle, the CNC will update, if necessary, the tool table; by the same token, the values of the arithmetic parameters will have a specific meaning which will be described in the sections for each cycle.

The exit conditions of all probing cycles are: G00, G07,G40,G90 and G94 .

The type of probe used in this cycles may be either one located in a fixed position on the machine (used to calibrate the tools) or one placed in the tool magazine (used to measure parts).

The latter probe will act as if it were a tool and must be calibrated prior to the execution of the cycle and the values entered in the appropriate tool table position.

The different values of the probe will be entered in the tool table in the following way:

. **The radius R** of the probe's ball will be entered manually in operating mode **8**.

. Next, the tool calibration cycle (N00) will be performed, after which the CNC will load the **L** value of the probe and will set the **K** value to zero.

. Then, the probe calibration cycle (N01) will performed. The CNC will load automatically the probe offset values (I,K) which will be the possible errors due to an improper setting of the probe in the tool holder.

While executing a probing canned cycle, if the CNC receives the probe signal without the probing movement itself being executed, it will issue an error **65** stopping all axes(collision).

When the probe uses an infrared system to send the signal, it is necessary to indicate, with machine- parameter, which **M** function must the CNC send to activate the probe.

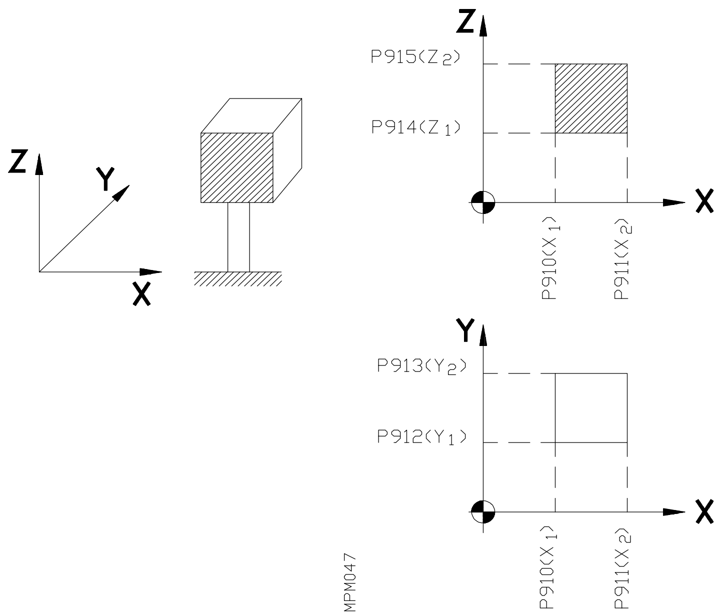
This **M** function will be activated by the CNC at the beginning of the probing cycle and must be cancelled by programming another **M** function.

N00. Tool length calibration cycle

This cycle will be used to measure the tool's length on the axis perpendicular to the main working plane. To do this, a probe must be placed in a fixed position on the machine and with its sides parallel to the axes.

The CNC must know this position on each axis and with respect to the machine-reference-zero. These values must be entered in the following parameters: P910 Minimum (X1) value according to X axis.

P911 Maximum (X2) value according to X axis.
P912 Minimum (Y1) value according to Y axis.
P913 Maximum (Y2) value according to Y axis.
P914 Minimum (Z1) value according to Z axis.
P915 Maximum (Z2) value according to Z axis.



The tool's approximate **L** value must be previously entered in the tool table. Once the tool has been selected, it can be calibrated by executing this cycle.

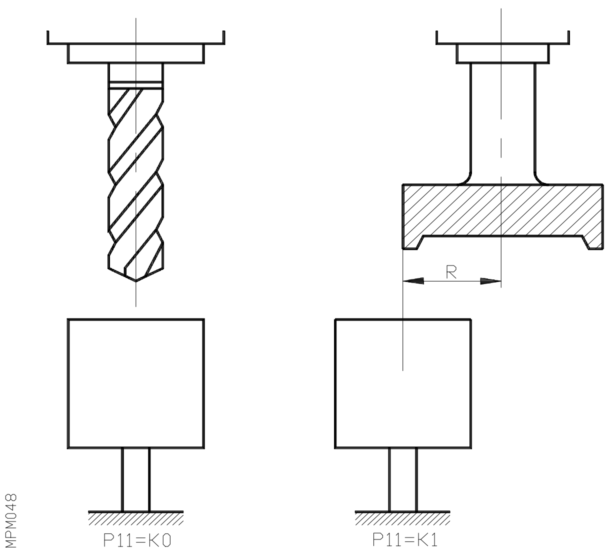
Cycle programming format: **G75 N00 P3=K— P4=K— P11=K—**

G75 N00 = Tool calibration cycle code.
P3 = Safety distance.
P4 = Probing feedrate.
P11 = Tool calibration on :

. the axis P11=0
. one end P11=1

This cycle will probe the tool over the probe, being the probing axis the one perpendicular to the main working plane. That is, the **Z** axis with G17, the Y axis with G18 and the **X** axis with **G19**.

Depending on the value of **P11**, the probing will be done with the tool's center (P11 =K0) or with one end (P11=K1).

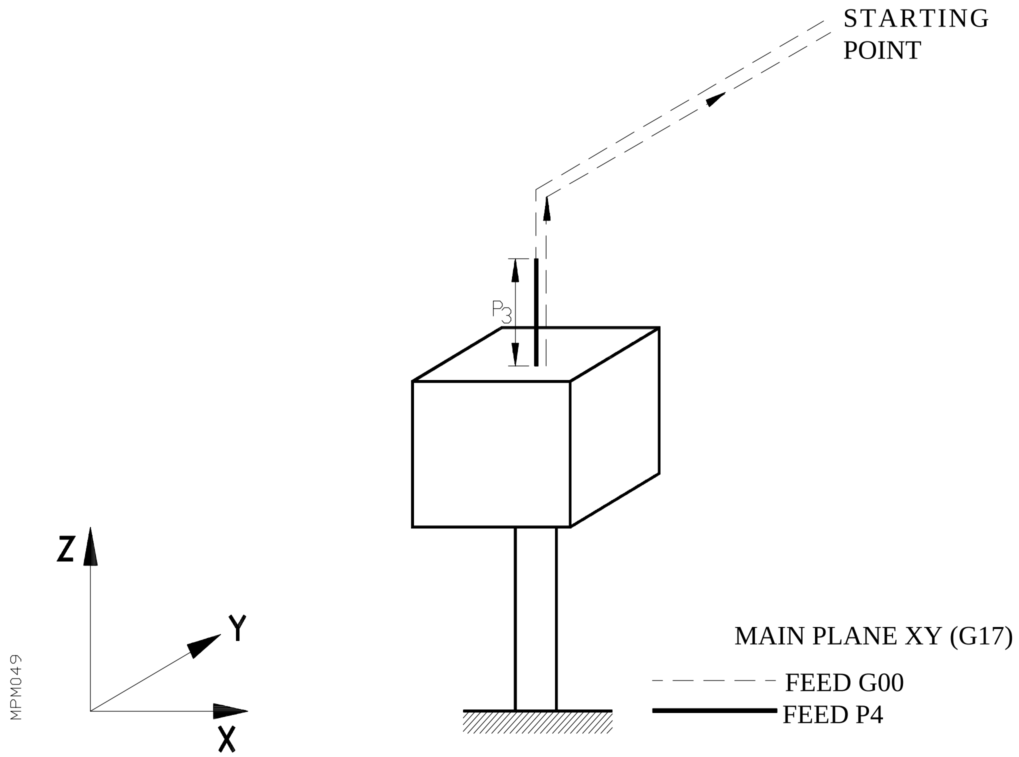


The tool will be positioned next to the probe's surface by first moving the axis of the main plane at rapid move **G00** and then the axis perpendicular to this plane up to a distance **P3** from the probe's surface also at rapid move **G00**.

Next, the probing cycle will be performed at a feedrate determined by **P4** to a maximum distance of **2P3**.

If after reaching a distance of **2P3** the CNC has not received the probe's signal, error 65 will be displayed.

Once the probe's signal is received, the CNC will stop the movement, load the real measurement value and return to the starting point of the cycle; as shown by the diagram below



The measured tool length value is automatically loaded by the CNC in the pertinent tool table position as L value, setting the **K** value to zero. This cycle does not modify the **R** and **I** values which must be entered manually either in the operating mode **8** or by programming function **G50**.

At the end of this cycle, the parameter table will show the following values:

P93 = Real length minus the tool length **L**, that was in the tool table prior the execution of this cycle, on the **X** axis (YZ working plane).

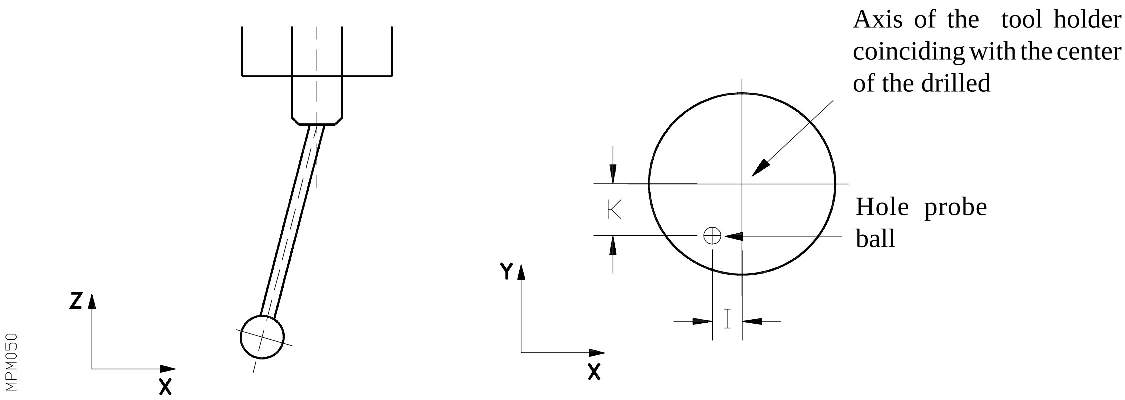
P94 = Real length minus the tool length **L**, that was in the tool table prior the execution of this cycle, on the **Y** axis (XZ working plane).

P95 = Real length minus the tool length **L**, that was in the tool table prior the execution of this cycle, on the **Z** axis (XY working plane).

N01. Probe calibration cycle

This cycle is used to determine the offset values of the probe. This values will be introduced in the relevant tool-offset number of the table in the **I** and **K** positions.

The offset values will be the error, in the axis of the main plane, between the center line of the tool holder and the center of the probe's ball. To execute this cycle, a hole must be previously drilled and its inside dimensions taken.



The programming format is as follows:

G75 N01 P0=K—P1=K—P2=K—P3=K—P4=K—P8=K—P9=K—P10=K- -

G75 N01 = Probe calibration cycle code.
P0 = Real **X** value of the drilled hole's center.
P1 = Real **Y** value of the drilled hole's center.
P2 = Real **Z** value of the drilled hole's center.
P3 = Safety distance.
P4 = Probing feedrate.
P8 = Drilled hole's diameter.
P9 = Initial probing feedrate.
P10 = Withdrawal distance after initial probing.

This cycle starts by positioning the probe at the center of the hole (XP0, YP1, ZP2), it then executes four probings movements (2 per axis) inside the hole.

At the end of the cycle, the probe returns to the starting point and the **I** and **K** values of the tool table are updated.

The probing movements of this cycle are similar to those explained for the hole centering cycle (N08) in a later section.

Once executed calibrating cycles **N00** and **N01**, the probe's values (except its radius) will be entered in the relevant tool table. These values are:

R = Probe's ball's radius (to be entered manually in operating mode **8** or by programming function **G50**.

L = Probe's length.

I = Main plane's offset value on the abscissa (X axis in the XY plane).

K = Main plane's offset value on the ordinate (Y axis in the XY plane).

This probe, mounted in the tool holder, will be used to execute the rest of the cycles.

N02. Surface measuring cycle

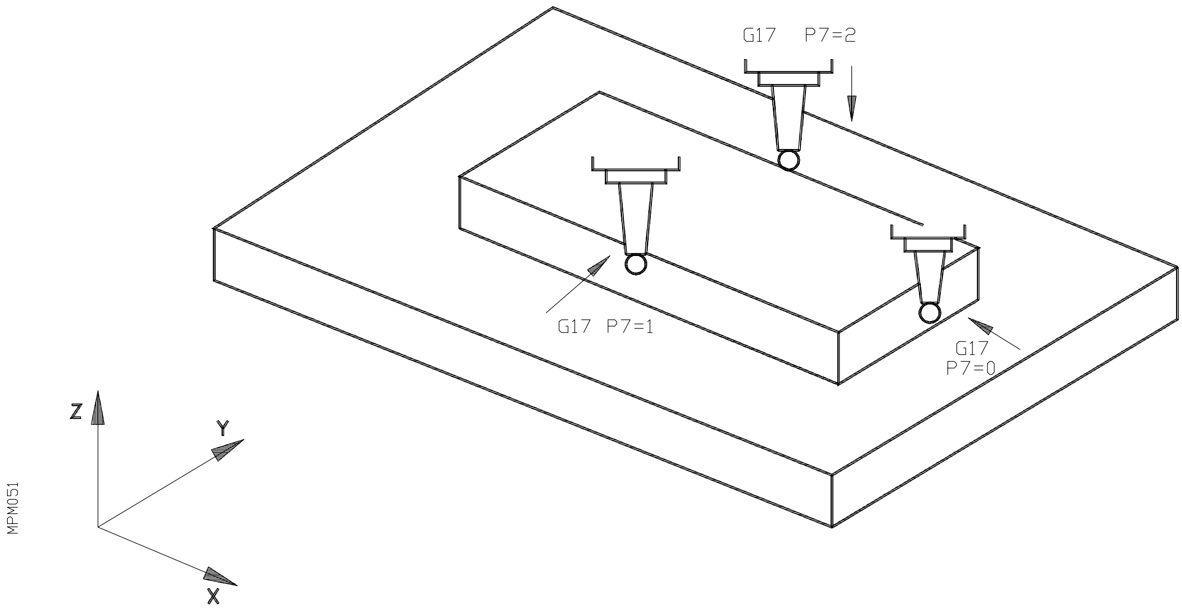
Programming format:

G75 N02 P0=K— P1=K— P2=K— P3=K— P4=K— P7=K—

G75 N02 = Surface measuring cycle code.
P0 = Theoretical **X** value of the point to be measured.
P1 = Theoretical **Y** value of the point to be measured.
P2 = Theoretical **Z** value of the point to be measured.
P3 = Safety distance.
P4 = Probing feedrate.
P7 = Axis being probed.

P7 = 0 X Axis
P7 = 1 Y Axis
P7 = 2 Z Axis

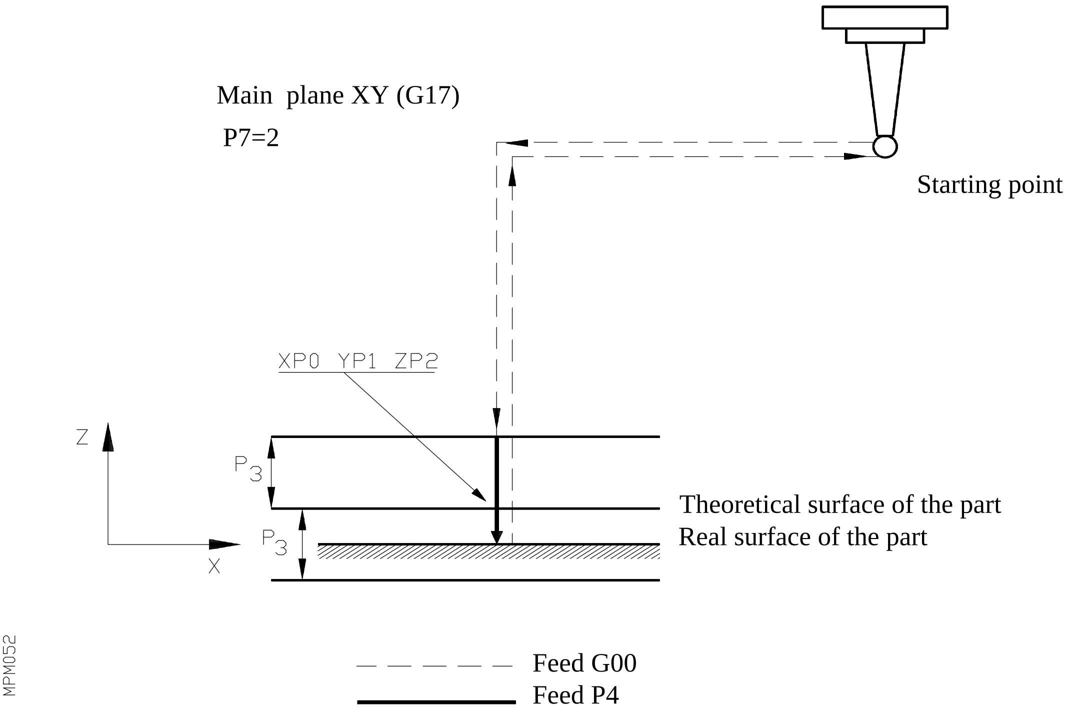
The probing movement will be performed only on the axis selected with **P7**.



The probe will be positioned near the point to be measured at a distance **P3**; the probing movement will be performed at a feedrate established by **P4** for a maximum distance of **2P3**.

If the CNC does not receive the probe's signal before reaching **2P3**, error 65 will be displayed.

Once the probing is done, the CNC will stop the movement, load the real values measured and will return to the cycle's starting point as shown in the diagram below.



Once the cycle is ended, the parameter table will show the following values:

P90 = X measured value
P91 = Y measured value
P92 = Z measured value
P93 = Real measured value minus theoretical value on X axis (P90-P0)
P94 = Real measured value minus theoretical value on Y axis (P91-P1)
P95 = Real measured value minus theoretical value on Z axis (P92-P2)

Parameters P93, P94 and P95 will indicate the offset value to be added to the part's datum so the part's theoretical values will be the same as the real ones. To do so, a function of the following type may be used:

G53/G59 I P93 J P94 K P95

N03. Surface measuring cycle with tool correction

Programming format:

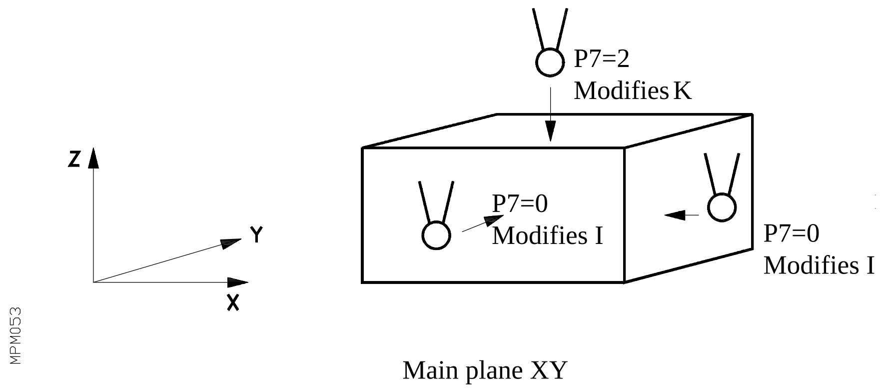
G75 N03 P0=K— P1=K— P2=K— P3=K— P4=K— P5=K— P6=K- - P7=K—

G75 N03 = Surface measuring cycle code with tool offset.
P0 = Theoretical X value of the point to be measured.
P1 = Theoretical Y value of the point to be measured.
P2 = Theoretical Z value of the point to be measured.
P3 = Safety distance.
P4 = Probing feedrate.
P5 = Tolerance.
P6 = Tool offset number.
P7 = Axis being probed.

P7 = 0 X Axis
P7 = 1 Y Axis
P7 = 2 Z Axis

With this cycle, besides executing everything described for the surface measuring cycle (N02), the CNC willmodify the tool table values according to the tool offset number indicated by **P6**. This modification will be carried out if the measuring error is not within the tolerance indicated by **P5**.

The CNC will modify in the tool table the **I** (radius) and the **K** (length) values according to the working plane and the axis selected by **P7**.



Main plane XY

N04. Outside edge measuring cycle

Programming format:

G75 N04 P0=K— P1=K— P2=K— P3=K— P4=K—

NG75 N04 = Outside edge measuring cycle code.
P0 = Theoretical X value of the point to be measured.
P1 = Theoretical Y value of the point to be measured.
P2 = Theoretical Z value of the point to be measured.
P3 = Safety distance.
P4 = Probing feedrate.

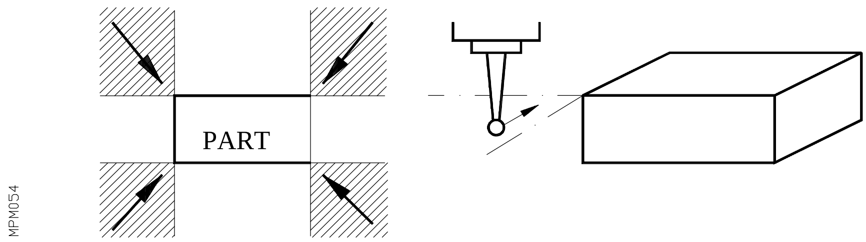
In this cycle two probings will be performed. The first one on the abscissa of the main plane, that is:

. On the X axis for the XY plane (G17)
. On the X axis for the XZ plane (G18)
. On the Y axis for the YZ plane (G19)

The second probing will be performed on the ordinate ofthe main plane, that is:

. On the Y axis for the XY plane (G17)
. On the Z axis for the XZ plane (G18)
. On the Z axis for the YZ plane (G19)

This cycle's starting point must be located in a specific area depending on the edge that is to be measured. The diagram below shows the striped areas where the probe must be located when the cycle is being called for to measure the pertinent edge.



The probe's movements will be the following: Let us suppose that the main plain is XY and the edge to be measured is the lower lefthand edge of the part (see fig.).

1. The probe will be positioned in rapid at a distance P3 of the first side to be measured.

2. The axis perpendicular to the main plane, in this case **Z**, will move in rapid to a coordinate value defined by P2.

3. The first probing will be performed by moving the **X** axis a maximum distance of 2P3 at a feedrate defined by P4 until the probe's signal is received. If, after reaching the maximum distance 2P3, the CNC has not received the probe's signal, error 65 will be displayed.

4. Once the first probing is done, the measured value will be loaded and then the **X** axis will return in rapid.

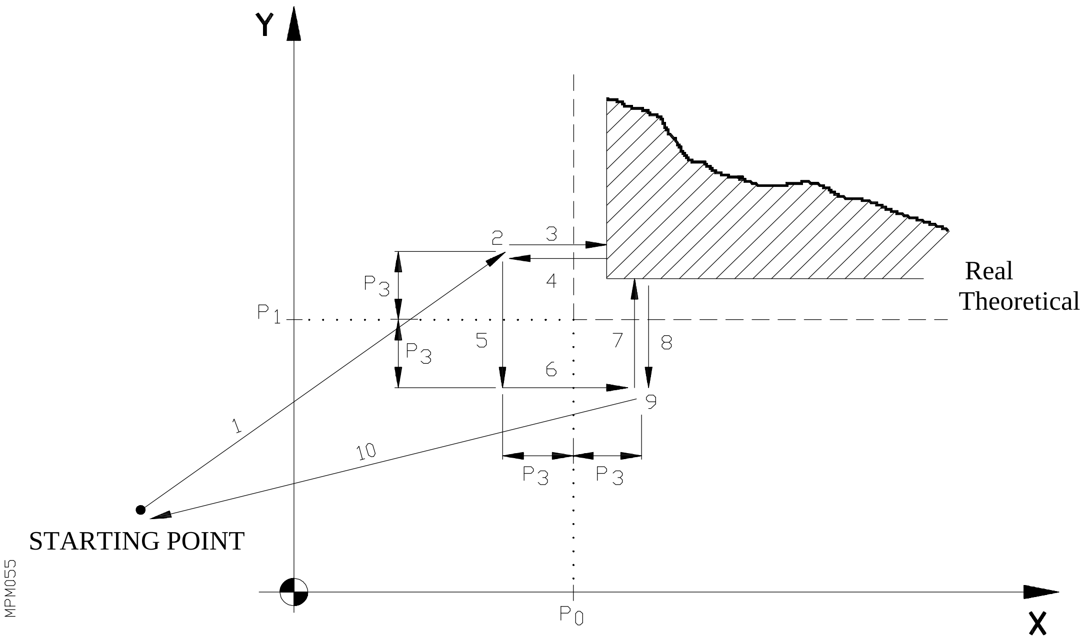
5 and 6. Next the probe will be positioned in rapid at a distance P3 of the other side to be measured as shown by the diagram.

7. The second probing will be performed by moving the **Y** axis a maximum distance of 2P3 at a feedrate defined by P4 until the probe's signal is received. If, after moving the maximum distance, the CNC has not received the probe's signal, error 65 will be displayed.

8. Once the second probing is done, the measured value will be loaded and then the Y axis will return in rapid.

9. The **Z** axis will move in rapid up to the **Z** value of the cycle's starting point.

10. **X** and **Y** axes will return in rapid to the cycle's starting point.



Once the cycle is finished, the parameter table will show:

P90 = Measured X value
P91 = Measured Y value
P92 = Measured Z value
P93 = Real value minus theoretical value on the X axis (P90-P0)
P94 = Real value minus theoretical value on the Y axis (P91-P1)
P95 = Real value minus theoretical value on the Z axis (P92-P2)

Parameters P93, P94 and P95 will indicate the offset value to be added to the part's datum point so the theoretical values will be the same as the part's realvalues. To do so, a function of the following type may be used:

G53/59 I P93 J P94 K P95

N05. Inside edge measuring cycle

Programming format:

G75 N05 P0=K— P1=K— P2=K— P3=K— P4=K—

G75 N05 = Inside edge measuring cycle code.
P0 = Theoretical X value of the point to be measured.
P1 = Theoretical Y value of the point to be measured.
P2 = Theoretical Z value of the point to be measured.
P3 = Safety distance.
P4 = Probing feedrate.

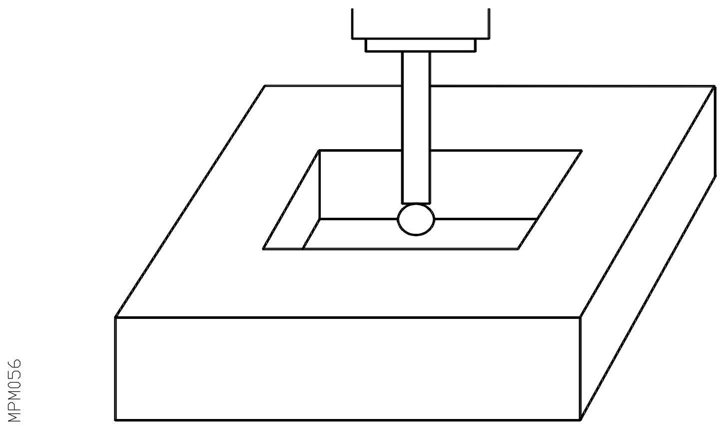
In this cycle two probings will be performed. The first one on the abscissa of the main plane, that is:

. On the **X** axis for the XY plane (G17)
. On the **X** axis for the XZ plane (G18)
. On the **Y** axis for the YZ plane (G19)

The second probing will be performed on the ordinate of the main plane, that is:

. On the **Y** axis for the XY plane (G17)
. On the **Z** axis for the XZ plane (G18)
. On the **Z** axis for the YZ plane (G19)

The probe must be located inside the pocket before calling for the cycle.



The probe's movements will be the following:

Let us suppose that the main plain is XY and the edge to be measured is the upper righthand edge of the part (see fig.).

1. The probe will be positioned in rapid at a distance P3 of the first side to be measured.

2. The axis perpendicular to the main plane, in this case **Z**, will move in rapid to a coordinate value defined by P2.

3. The first probing will be performed by moving the **X** axis a maximum distance of 2P3 at a feedrate defined by P4 until the probe's signal is received. If, afterreaching the maximum distance 2P3, the CNC has not received the probe's signal, error **65** will be displayed.

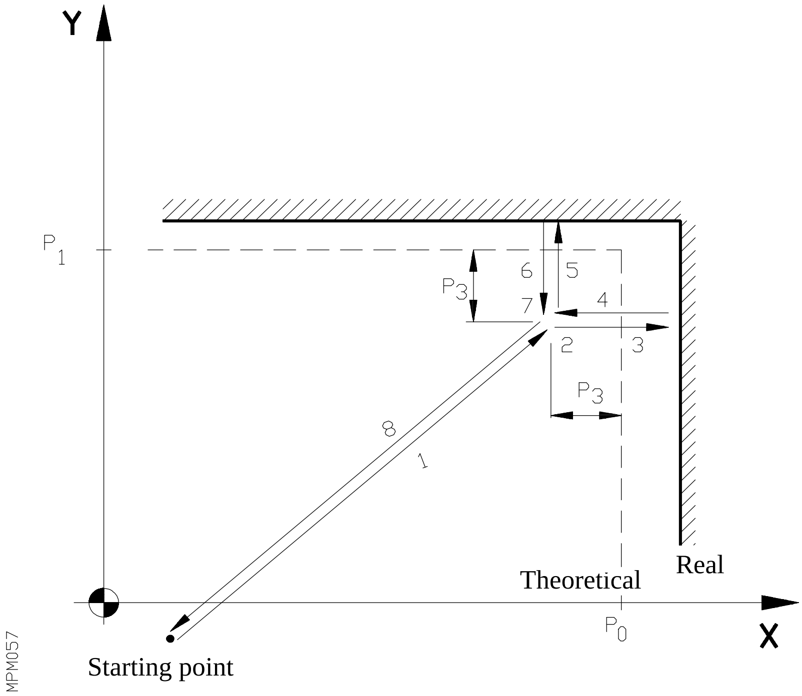
4. Once the first probing is done, the measured value will be loaded and then the **X** axis will return in rapid. Next, the probe will be positioned in rapid at a distance P3 of the other side to be measured as shown by the diagram.

5. The second probing will be performed by moving the **Y** axis a maximum distance of 2P3 at a feedrate defined by P4 until the probe's signal is received. If, after moving the maximum distance, the CNC has not received the probe's signal, error **65** will be displayed.

6. Once the second probing is done, the measured value will be loaded and then the **Y** axis will return in rapid.

7. The **Z** axis will move in rapid up to the **Z** value of the cycle's starting point.

8. **X** and **Y** axes will return in rapid to the cycle's starting point.



Once the cycle is finished, the parameter table will show:

P90 = Measured X value
P91 = Measured Y value
P92 = Measured Z value
P93 = Real value minus theoretical value on the X axis (P90-P0)
P94 = Real value minus theoretical value on the Y axis (P91-P0)
P95 = Real value minus theoretical value on the Z axis (P92-P0)

Parameters P93, P94 and P95 will indicate the offset value to be added to the part's datum point so the theoretical values will be the same as the part's real values. To do so, a function of the following type may be used:

G53/59 I P93 J P94 K P95

N06. Angle measuring cycle

Programming format:

G75 N06 P0=K-P1=K-P2=K-P3=K-P4=K-

G75 N06 = Angle measuring cycle code.
P0 = Theoretical X value of the point to be measured.
P1 = Theoretical Y value of the point to be measured.
P2 = Theoretical Z value of the point to be measured.
P3 = Safety distance.
P4 = Probing feedrate.

In this cycle two probings will be performed on the ordinate of the main plane, that is:

. On the **Y** axis for the XY plane (G17)
. On the **Z** axis for the XZ plane (G18)
. On the **Z** axis for the YZ plane (G19)

The probing movements are:

Let us suppose that the main plane is XY and we want to measure the inclination angle of the part with respect to the axes of the machine (see fig.).

1. The probe will be positioned in rapid at a distance 2P3 of the side to be measured.

2. The axis perpendicular to the main plane, in this case the **Z** axis, will move in rapid to a position indicated by P2.

3. The first probing will be performed moving the **Y** axis (ordinate of the XY plane) a distance 3P3 at a feedrate defined by P4 until the probe's signal is received. If after reaching a distance of 3P3, the CNC has not received this signal, error **65** will be displayed.

4. Once the first probing is finished, the measured value will be loaded and the **Y** axis will return in rapid.

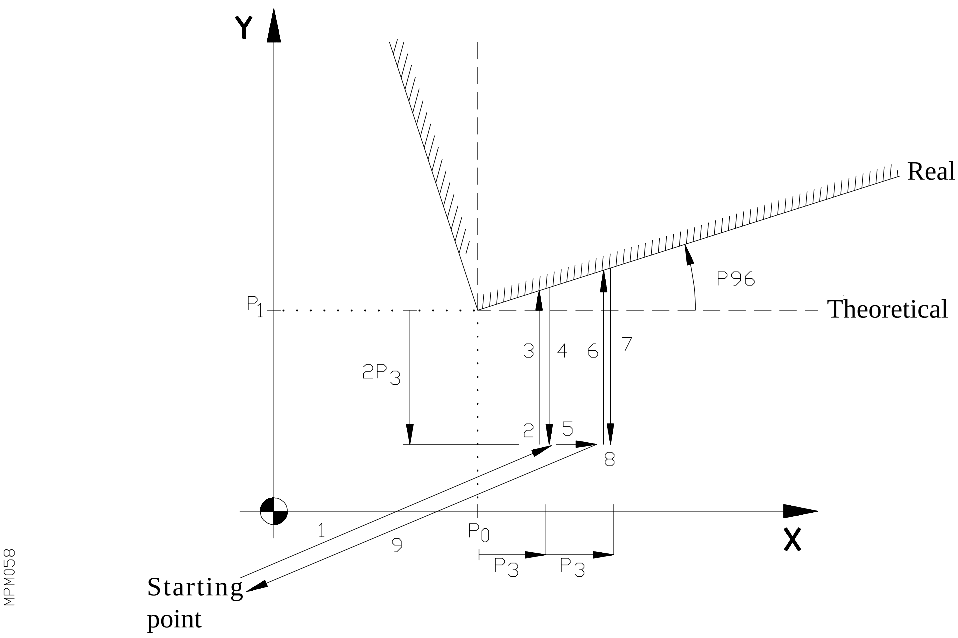
5. The **X** axis will move an incremental distance P3 in rapid.

6. The second probing will be performed at a feedrate defined by P4 and to a maximum distance of 4P3.

7. The **Y** axis will return in rapid.

8. The **Z** axis will return in rapid to the **Z** value of the starting point.

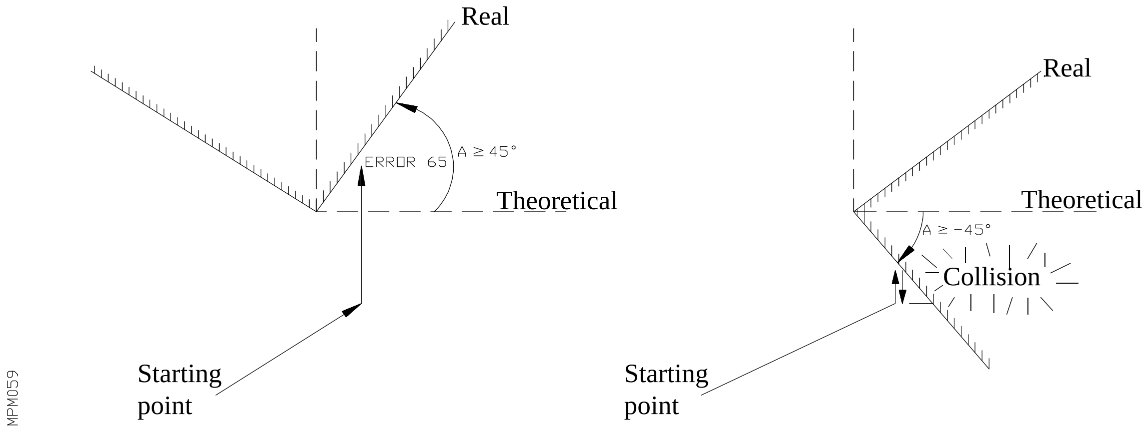
9. The **X** and **Y** axis will return in rapid to the starting point.



With this probing cycle, the maximum angle to be measured must be within +/-45 degrees.

If the angle is +45 degrees or larger, error 65 will be displayed during the first probing movement.

If the angle is -45 degrees or larger, the probe will collide with the part when executing a rapid move (G00), in that case, the CNC will stop the movement and display error 65.



When the cycle is finished, the CNC will have the value of the angle in the parameter **P96**

If the measured point of the part is the part's datum point, by means of the function to rotate the coordinate system:

G73 A P96

the axes of the machine will be the same as the sides of the part; thus, to execute the program, there will be no need to take into account the angle of the part.

N07. Outside edge and angle measuring cycle

Programming format:

G75 N07 P0=K— P1=K— P2=K— P3=K— P4=K—

G75 N07 = Edge and angle measuring cycle code.
P0 = Theoretical X value of the point to be measured.
P1 = Theoretical Y value of the point to be measured.
P2 = Theoretical Z value of the point to be measured.
P3 = Safety distance.
P4 = Probing feedrate.

In this cycle three probings will be performed. The first one on the abscissa of the main plane, that is:

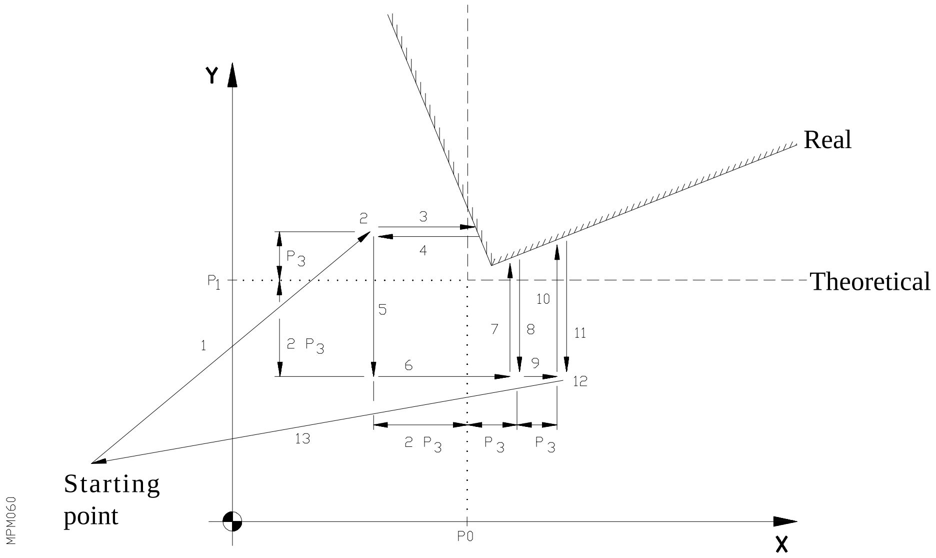
. On the **X** axis for the XY plane (G17)
. On the **X** axis for the XZ plane (G18)
. On the **Y** axis for the YZ plane (G19)

The other two probings will be performed on the ordinate of the main plane, that is:

. On the **Y** axis for the XY plane (G17)
. On the **Z** axis for the XZ plane (G18)
. On the **Z** axis for the YZ plane (G19)

It must be borne in mind that the probe's starting pointmust be in specific area as explained before for the outside-edge-measuring cycle.

The maximum angle must be within +/-45 degrees for the same reason as the angle-measuring cycle.



The probe's movements will be the following:

Let us suppose that the main plain is **XY** and the outside edge to be measured is the lower lefthand edge of the part and the inclination angle of the part with respect to the axes of the machine (see fig.).

1. The probe will be positioned in rapid at a distance 2P3 of the first side to be measured.

2. The axis perpendicular to the main plane, in this case **Z**, will move in rapid to a position defined by P2.

3. The first probing will be performed by moving the **X** axis a maximum distance of 3P3 at a feedrate defined by P4 until the probe's signal is received.

4. Once the first probing is done, the measured value will be loaded and then the X axis will return in rapid.

5 and 6. Next, the probe will be positioned in rapid at a distance 2P3 of the other side to be measured, as shown by the diagram.

7. The second probing will be performed by moving the **Y** axis a maximum distance of 3P3 at a feedrate defined by P4 until the probe's signal is received.

8. Once the second probing is done, the measured valuewill be loaded and then the **Y** axis will return in rapid.

9. The **X** axis will move an incremental distance P3 in rapid.

10. The third probing will be performed at a feedrate defined by P4 and to a maximum distance of 4P3.

11. The **Y** axis will return in rapid.

12. The **Z** axis will return in rapid to the **Z** value of the starting point.

13. The **X** and **Y** axis will return in rapid to the starting point.

Atention:

In any of the rapid probing movements (3), (7), (10), if after reaching the maximum distance (3P3), (3P3), (4P3) the CNC has not received the probe's signal, error **65** will be displayed.

When the cycle is over, the CNC's parameter will show:

P90 = Real X value of the edge.
P91 = Real Y value of the edge.
P92 = Real Z value of the edge.
P93 = Real X value minus theoretical X value of the edge (P90-P0).
P94 = Real Y value minus theoretical Y value of the edge (P91-P1).
P95 = Real Z value minus theoretical Z value of the edge (P92-P2).
P96 = Angle.

Parameters P93, P94 and P95 indicate the offset amount to be added to the part's datum point so the theoretical values of the part are the same as the real values. To do so, the following function may be used:

G53/59 I P93 J P94 K P95

But, if it is desired to make the part's datum point coincide with the probed point, the datum point may be moved by the following function:

G53/59 I P90 J P91 K P92

and also, by means of the function to **ROTATE** the coordinate system:

G73 A P96

The axes of the machine may be made to coincide with the sides of the part (as long as the measured edge coincides with the part's datum point) so the program can be executed without taking into account the angle.

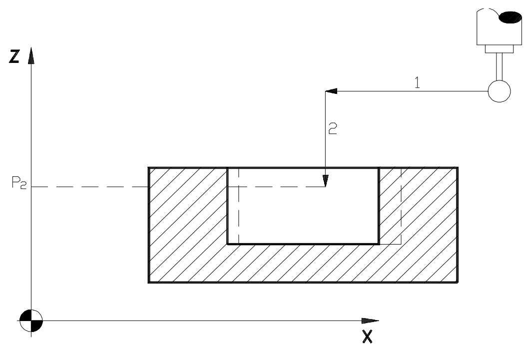
N08. Hole centering cycle

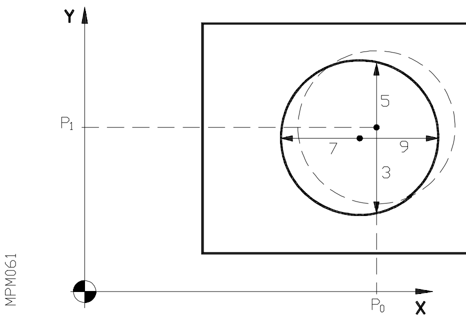
Programming format:

G75 N08 P0=K— P1=K— P2=K— P3=K— P4=K— P8=K— P9=K- - P10=K—

G75 N08 = Hole centering cycle code.
P0 = Theoretical X value of the hole's center.
P1 = Theoretical Y value of the hole's center.
P2 = Theoretical Z value of the hole's center.
P3 = Safety distance.
P4 = Probing feedrate.
P8 = Theoretical hole's diameter.
P9 = Initial probing feedrate.
P10 = Withdrawal distance after initial probing.

In this cycle, four probing movements will be performed on the sides of the hole; the first two on the ordinate of the main plane (Y axis of the XY plane) and the other two on the abscissa of such plane (X axis on the XY plane). Once the probing movements are completed, the CNC will finish the cycle by positioning the probe at the real center of the hole calculated by the CNC.





Next, the cycle movements are described in better detail.

Let us suppose that the main plane is **XY**. See fig.

The probe will be positioned at the theoretical center of the hole (XP0, YP1, ZP2). Next, the main plane axes will be probed (movement 1) and then, the axis perpendicular to the main plane (movement 2).

Both movements will be executed in rapid mode G00.

3. First probing movement, **Y** axis.

   This movement is divided into:

   . Movement at a feedrate determined by P9 until the probe’s signal is received.

   . Probe’s withdrawal in G00 to a distance determined by P10.

   . Movement at a feedrate determined by P4 until the probe’s signal is received again.

4. The **Y** axis returns to the theoretical value Y=P1 in rapid.
5. Second probing movement, **Y** axis (similar to point 3).
6. The **Y** axis returns to the real center calculated on this axis.
7. Third probing movement, **X** axis (similar to point 3).
8. The **X** axis returns to the theoretical value X=P0 in rapid.
9. Forth probing, **X** axis (similar to the point 3).
10. The **X** axis returns to the real center calculated on that axis, thus positioning the probe in the real center of the hole and ending this cycle.

Atention:

If the real diameter of the hole is larger than P8+P3, the CNC will display error 65 when executing a probing movement.

Once the hole centering cycle is ended, the parameter table will show:

P90 = Real X value of the center of the hole.
P91 = Real Y value of the center of the hole.
P92 = Real Z value of the center of the hole.
P93 = Real value minus theoretical value on the X axis (P90-P0).
P94 = Real value minus theoretical value on the Y axis (P91-P1).
P95 = Real value minus theoretical value on the Z axis (P92-P2).
P96 = Real diameter value of the hole.
P97 = Real value minus theoretical value of the diameter of the hole (P96-P8).

Parameters P93, P94, P95 indicate the offset value to be added to the part's datum point so the theoretical values will be the same as the real ones. To do this, the following function type may be used:

G53/G59 I P93 J P94 K P95

N09. Boss centering cycle

Programming format:

G75 N09 P0=K— P1=K— P2=K— P3=K— P4=K— P8=K— P9=K- - P10=K—

G75 N09 = Boss centering cycle.
P0 = Theoretical X value of the center of the boss.
P1 = Theoretical Y value of the center of the boss.
P2 = Theoretical Z value of the center of the boss.
P3 = Safety distance.
P4 = Probing feedrate.
P8 = Theoretical diameter of boss.
P9 = Initial probing feedrate.
P10 = Withdrawal distance after initial probing.

In this cycle, four probing movements will be performed on the sides of the boss; the first two on the ordinate of the main plane (Y axis of the XY plane) and the other two on the abscissa of such plane (X axis on the XY plane).

The diagram illustrates the movements of the axes during this cycle. Movements 5, 10, 15 and 20 are the probing movements and each one of them is divided into the following:

. Probing at a feedrate determined by P9 until the signal from the probe is received.

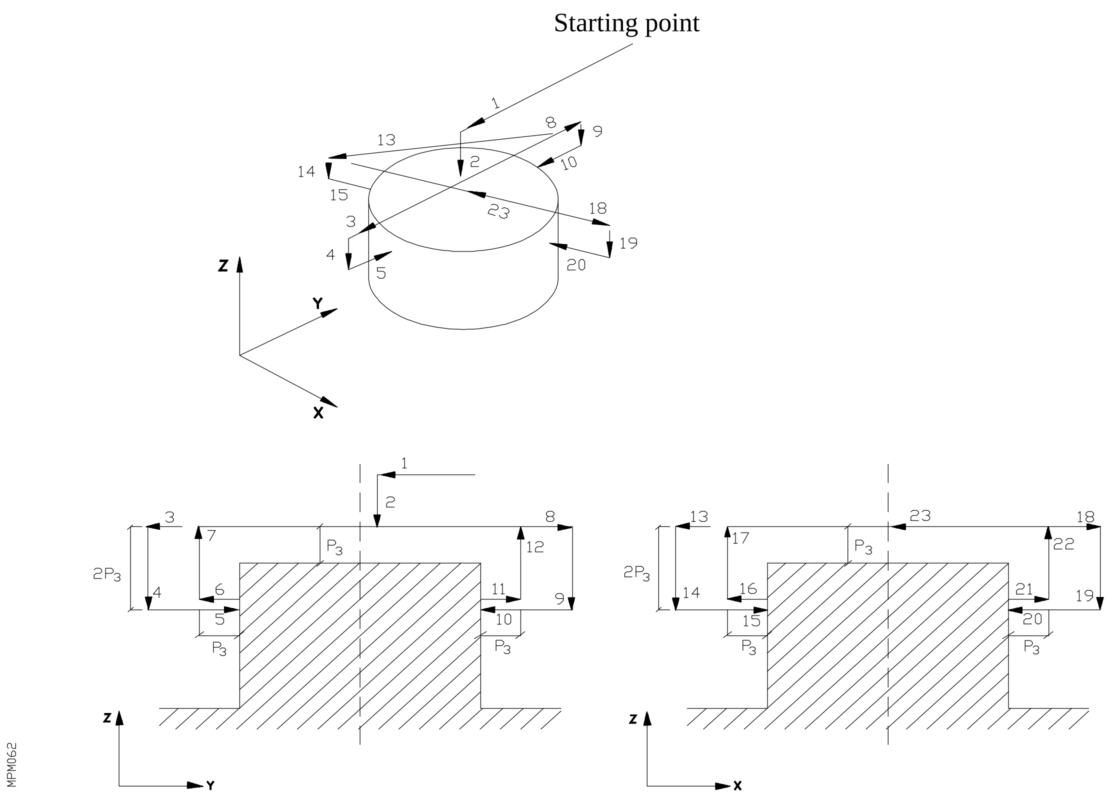
. Withdrawal of the probe in rapid to a distance determined by P10.

. Probing at a feedrate determined by P4 until the signal from the probe is received.

The rest of the movements will be performed at **G00** (rapid mode). The cycle will end with the main plane axes positioned at the real center of the boss and the axis perpendicular to the main plane at a distance P3 from this center.

Atention:

If the real diameter of the boss is larger than **P8+P3**, the CNC will display error **65** when executing a probing movement.



Once the boss centering cycle is ended, the parameter table will show:

P90 = Real X value of the center of the boss.
P91 = Real Y value of the center of the boss.
P92 = Real Z value of the center of the boss.
P93 = Real value minus theoretical value on the X axis (P90-P0).
P94 = Real value minus theoretical value on the Y axis (P91-P1).
P95 = Real value minus theoretical value on the Z axis (P92-P2).
P96 = Real diameter value of the boss.
P97 = Real value minus theoretical value of the diameter of the boss (P96-P8)

Parameters P93,P94,P95 indicate the offset value to be added to the part’s datum point so the theoretical values will be the same as the real ones. To do this, the following function type may be used:

G53/G59 I P93 J P94 K P95

N10. Hole measuring cycle

Programming format:

G75 N09 P0=K— P1=K— P2=K— P3=K— P4=K— P8=K— P9=K- - P10=K—

G75 N10 = Hole measuring cycle code
P0 = Theoretical X value of the center of hole.
P1 = Theoretical Y value of the center of hole.
P2 = Theoretical Z value of the center of hole.
P3 = Safety distance.
P4 = Probing feedrate.
P8 = Theoretical diameter of hole.
P9 = Initial probing feedrate.
P10 = Withdrawal distance after initial probing.

This cycle is identical to the hole-centering cycle **N08** described before except that at the end of the cycle, the probe returns to the cycle's starting point. To do this, first the axis perpendicular to the main plane moves and then the two axes of the main plain move. Both movements are performed at **G00** rapid mode.

N11. Boss measuring cycle

Programming format:

G75 N09 P0=K— P1=K— P2=K— P3=K— P4=K— P8=K— P9=K- - P10=K—

G75 N09 = Boss measuring cycle code.
P0 = Theoretical X value of the center of the boss.
P1 = Theoretical Y value of the center of the boss.
P2 = Theoretical Z value of the center of the boss.
P3 = Safety distance.
P4 = Probing feedrate.
P8 = Theoretical diameter of the boss
P9 = Initial probing feedrate.
P10 = Withdrawal distance after initial probing.

This cycle is identical to the boss-centering cycle **N09** described before except that at the end of the cycle, the probe returns to the cycle's starting point. To do this, first the axis perpendicular to the main plane moves and then the two axes of the main plain move. Both movements are performed at **G00** rapid mode.

6.30. DIGITIZING WITH THE FAGOR 8025/8030 MS CNC

6.30.1. Digitizing

Digitizing consists of memorizing the coordinates from a guided sweep of the probe on the model. This is done at the speed allowed by the probe. The data which is obtained is used later during the milling stage. This method has the following advantages:

* Machining can be done at the maximum speed allowed by the machine tool.
* There are no vibrations during the copying stage, and for this reason reproduction is much more exact and the need for manual finishing is avoided in the majority of cases.
* Digitized information can be used to machine as many times as may be necessary, without any need for copying the original model again.
* Probing speed can be adjusted between 0 and 1000 mm/min. The best results are obtained with speeds of from 200 and 500 mm/min. Probing feed rate can be adjusted between 0 and 1500 mm/min.

The digitizing stage consumes about a quarter of the total processing time. The time during which the machine tool is being used should not be thought of as being unproductive, as in the long run, less time is consumed than in direct copying. Furthermore, no manual operations are required so this can be done at night or during a weekend.

If it is wished to get maximum performance from machine tools, a measuring machine can be used exclusively for digitizing models. The programs generated will feed the different milling machines used solely for machining work. The measuring machine can also be used for dimensional control of parts from machining operations.

6.30.2. CHARACTERISTICS OF DIGITIZING WITH THE FAGOR 8025/30 MS CNC

Any digital probe can be used with the 8025/30 CNC.

During the digitizing phase, a simple program moves the probe on the pattern. The exploration can have the form of a **rectangular grid**, **concentric circumferences**, **spiral**, **diametric**, etc., so that it adapts as well as possible to the geometry of the model to be reproduced. It is also possible to define various areas and use a different exploration method in each of these.

One very important difference of the FAGOR digitizing method with respect to other systems which also use digital probes is that this one moves practically on the surface of the pattern.

. ADVANTAGES OF THE FAGOR METHOD

Less time is needed for the digitizing stage.

It can be used in large machines, even though the axis which moves the probe is very heavy, as it is not submitted to continuous rocking movements which could damage its mechanism.

After the data has been collected a program is generated which can be stored in the memory of the MS 8025/30 CNC or in the disc of a computer, by using the FAGORDNC communication system. This second option is the one used normally, as the programs which are generated by digitizing are usually of a large size than the memory capacity of the control (32 Kb).

If the pattern has any type of symmetry, only one part can be digitized and then, by applying mirror images 9G11, G12, G13), transfers (G92, G53 .... G59) and axis turns (G73), the complete pattern can be reproduced. This allows a reduction both in the digitizing time and the length of the program.

Reproduction can be obtained with smoothed paths if, instead of going from one point to another in a straight line (G1), G8 functions are used (Tangent circumference to the previous path) and G9 (circumference defined by three points).

The application of scale factors (G72) allows a complete family of parts to be made from a single pattern.

All these functions, the coordinates of the points, as well as machining conditions (feed rate, tool to be used, spindle revolutions, etc.), can be entered automatically during the digitizing stage by means of the G76 function, for which reason it is not necessary to edit the program which is generated afterwards.

Should it be necessary to make modifications the control reserves 100 blocks before the first block (N100) generated by the digitizing process.

The program can occupy up to several Mb of memory. During the machining stage, it is necessary to transmit it as an infinite program using FAGORDNC. DNC software guarantees safe transmission of data by means of an RS 232C lines. For this reason, it has a communications protocol which automatically retransmits the data should there be an error in transmission or reception.

Finally, it is also possible to send the program generated from digitizing to a CAD/CAM system capable of reconstructing the geometry of the pattern. Once there, the original design could be modified and the process is completed by machining the definitive design.

6.30.3. Preparation of a digitizing operation and later execution at the machine.

. CONCEPTION OF THE SYSTEM. THE PROBE.

The probe can be fastened to the toolholder of the milling machine or machining center, as if it were a tool, converting the machine tool into an automatic digitizing system.
The tip (interchangeable) of the needle of the probe isprovided with a ball which is threaded to the probe and follows the surface of the pattern during digitizing. Each probe involves a family of tips with different ball radii for multiple applications.

The **diameter of the ball** of the needle or tip should be the same as the tool used in subsequent machining.

The corrections of radii for other tools are also possible but another treatment of the digitized program is required (G41, G42, G43).

The different probe needles have variable weights. In fact, in the **probe system, needles must have a maximum weight of 200 gm** approximately to avoid possible errors of interpretation of contact.

. CALIBRATING THE PROBE

For this, we use the N01 cycle with which we determine the offset values for the probe, which will be entered by the CNC in the corresponding corrector, which we have chosen previously. (T00 by default.) The offset values are the error which may exist in the axes of the main plane between the axis of the toolholder and the center of the measurement probe ball.

In order to execute this cycle it is necessary to machine a hole beforehand, inside which we will carry out the probings.

Once the hole has been made, the diameter and X,Y,Z coordinates of which we know (this is due to that fact that we have chosen the place previously and moved to it with the CNC **jog controls**) we change the tool for the probe and move in Z until we are inside the hole.

Next, we execute the N01 probe calibration cycle. Previously the programming format is completed and the tool corrector is chosen where we want offset I,K to appear. T00 corrector is taken by default. All these operations can be done in **TEACH-IN**.

On exiting from the cycle the control automatically updates the I,K offset of the table and the probe goes back to the starting point. Next we complete the rest of the information on the table:

R : Radius of the ball
L : Length of the probe (depends on the zero part).
If zero part is on the surface of part, L will be zero also.

This type of probe placed on the toolholder of the spindle will be used to carry out the remaining probingcycles. If we change the probe for another, we must repeat the entire process. Once the probe has been calibrated we can proceed with the probing of the surface chosen.

. DIGITIZING OF THE PATTERN

Digitizing consists of the reading of points on a surface with a measurement probe.

Points are read with the combination of the preparatory functions of the CNC:

- Function **G75** allows the reading and acceptance of the points by the CNC.
- The **G76** function allows these to be stored in the CNC itself, if the contents are less than 32 Kb, or in a computer.

The program obtained in this way allows the reproduction of the points and the generation of the surface which has been digitized previously in two ways:

- From the CNC itself, if the contents are less than 32 Kb.
- Of from a computer by means of the FAGORDNC application using the option: EXECUTION OF THE INFINITE PROGRAM.

1-Sampling program

This is a CNC program which guides the probe along the surface to be digitized in a succession of points which is as extensive and dense as the computer systems available permit.

The probe travels over the surface of the model at defined intervals of space, defined in the sample program. The coordinates of these points will be read and the different blocks of the machining program will be generated.

By observing the model to be digitized and depending on its geometry we can choose different types of sampling:

-Rectangular probing according to the X axis.
-Rectangular probing according to the Y axis.
-Circular probing.
-Diametric probing.
-Profile monitoring probing.
-Combinations of these.
-Etc.

Later, examples of these sampling programs will be seen.

2 - Considerations on the sampling program.

The execution of the sampling program implies the following steps:

a) The probe will go to a specific point above the surface of the pattern.

b) Next, with the aid of function G75 the reading of the different coordinates (W), (V), X, Y, Z.

After G75 the probe will lower as far as the programmed coordinate until it receives the external signal of the probe. Once this has been received, the block will be considered complete, the real position of the point of contact of the probe being accepted as the theoretical position.

If the axes arrive at a programmed position before receiving the probe signal, the CNC will indicate error 65.

c) With the aid of a block which contains the G76 function a block can be generated which will be sent automatically, either to the CNC memory or to a computer via DNC.

The information after G76 can be:

- Coordinates of the (W), (V), X, Y, Z axes.
- G,F,S,T functions.

This entire process will be repeated for one of the points until the chosen sampling program is complete.

3 - Final considerations.

Digitizing is always carried out within a defined volume. The planes which delimit this volume are parallel to the machine axes. Thanks to the appropriate distribution of the planes parts of the contour can be digitized.

It is possible to divide the surface of the pattern into several parts and **define a different sampling network for each area**, all this by means of the combination of different sampling sweeps which Fagor offers as an example.

The sequence of points must have a logical form for later machining, where the tool, with the same shape as the probe ball, will travel over the line of points stored in the program. If it is necessary to machine in several runs the program must be executed several times by applying successive **origin displacements or changes in the tool length compensation.**

In a previous block, **the control automatically reserves 100 blocks in which preparatory functions can be defined which affect all the program: rounded edges, scale factor, axis turn, etc**.

Thanks to different processes within the digitizing program, we can optimize the probing of the pattern. For example, geometrical aid functions can also be entered in the generation block G76 with which it is possible to round off the machining profile calculated point by point.

One of the multiple applications of the G76 function is the creation of a program known as the mathematical function. The path followed is calculated by means of a parametric program and executing it in **DRY RUN**. These programs have a special sense when the mathematical function is very complex and the control cannot process all the calculation in real time simultaneously with machining. The path breaks down previously into successive points, with the possibility of rounding off, for example, being stored as a new program.

. FAGORDNC FOR DIGITIZING

Once executed, FAGORDNC selects the **DIGITIZING** option. Once this has been done, the computer waits to receive data from the CNC. Now we execute the probing program which has been chosen previously for the pattern. **When the CNC stops digitizing the whole surface of the pattern, the computer will indicate the PROGRAM RECEIVED message**.

The programs stored in the computer can be modified with any text editor which generates ASCII characters, as if they were texts. In this way we can modify the depth of the run, work rate, etc., or program machining conditions in the first 100 blocks reserved for this.

In order to execute the program store in the computer and after executing this, the FAGORDNC communicationsprogram, we will choose the **INFINITE PROGRAM EXECUTION** option. The computer will ask for the program number, and afterwards, the number of times that it will repeat the program, and finally, we will choose between executing the program in **AUTOMATIC, DRY RUN, "G" FUNCTIONS, THEORETICAL PATH**. After this sequence of keystrokes, the computer starts sending the program generated to the numerical control, following the path of the previously digitized surface. Once the program has been completely executed, the computer will show the **PROGRAM EXECUTED** message.

It is very important to be familiar with the OPERATING SYSTEM of the computer to carry out all these processes. On occasions, it is of invaluable help.

. PARAMETERS INVOLVED WITH DIGITIZING.

P612 bit 7 indicates the type of impulse (+ or -).
P720 if G75 sends M.

The 9-pin **A6** connector is used for receiving the signals from a measurement probe. (Specifications in the Installation and Start-up Manual).

6.30.4. G76. Automatic block generation

This function (G76) is used to generate blocks that are automatically loaded into the CNC or to a computer (via DNC).

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

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

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

The programming format is:

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

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

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

a) (V+/-4.3)(W+/-4.3) X+/- 4.3 Z+/-4.3

   Loads the axes with the indicated values.

b) (V)(W) X Y Z

   Loads the axes with the theoretical values that they show at this time.

c) (VP2)(WP2) XP2 ZP2

   Loads the axes with the values of the parameter at this time.

In the same way, if FP2 or SP2 are programmed after G76, the CNC will load the F or S in the new program with the values of the parameter in that moment.

Example: Let us suppose that the **X** coordinate of the point where the machine finds itself is 78.35. If we run the following program:

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

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

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

It is necessary to program all five digits of the program number in blocks of type **G76 P5** or **G76 N5.**

The CNC must be in **DNC ON** (operating mode 7) in order to load the new program into a computer (see DNC manual).

If the number of the program to be generated exists already in memory (e.g. **P12345**) it must be in the last position of the program map; but if **G76 P12345** is executed, the old program is erased and the new one can be generated.

When the program number exists in memory but is not the last one in the memory map, the CNC will issue error 56.

Atention:

When a program is edited it goes to the last position in the memory map and when it is executed it goes to the first position.

When a program is being generated, another program cannot be generated until the generation of the previous one is cancelled by means of **M2**, **M30**, **RESET** or **EMERGENCY**.

Some of the applications of the G76 function are, for example, the creation of a program after the calculation of a path by means of a parametric program, or the **DIGITIZING** of a model with a measuring probe (G75) generating a point-to-point program as large as desired.

Example G76: DIGITIZING ALONG THE X AXIS

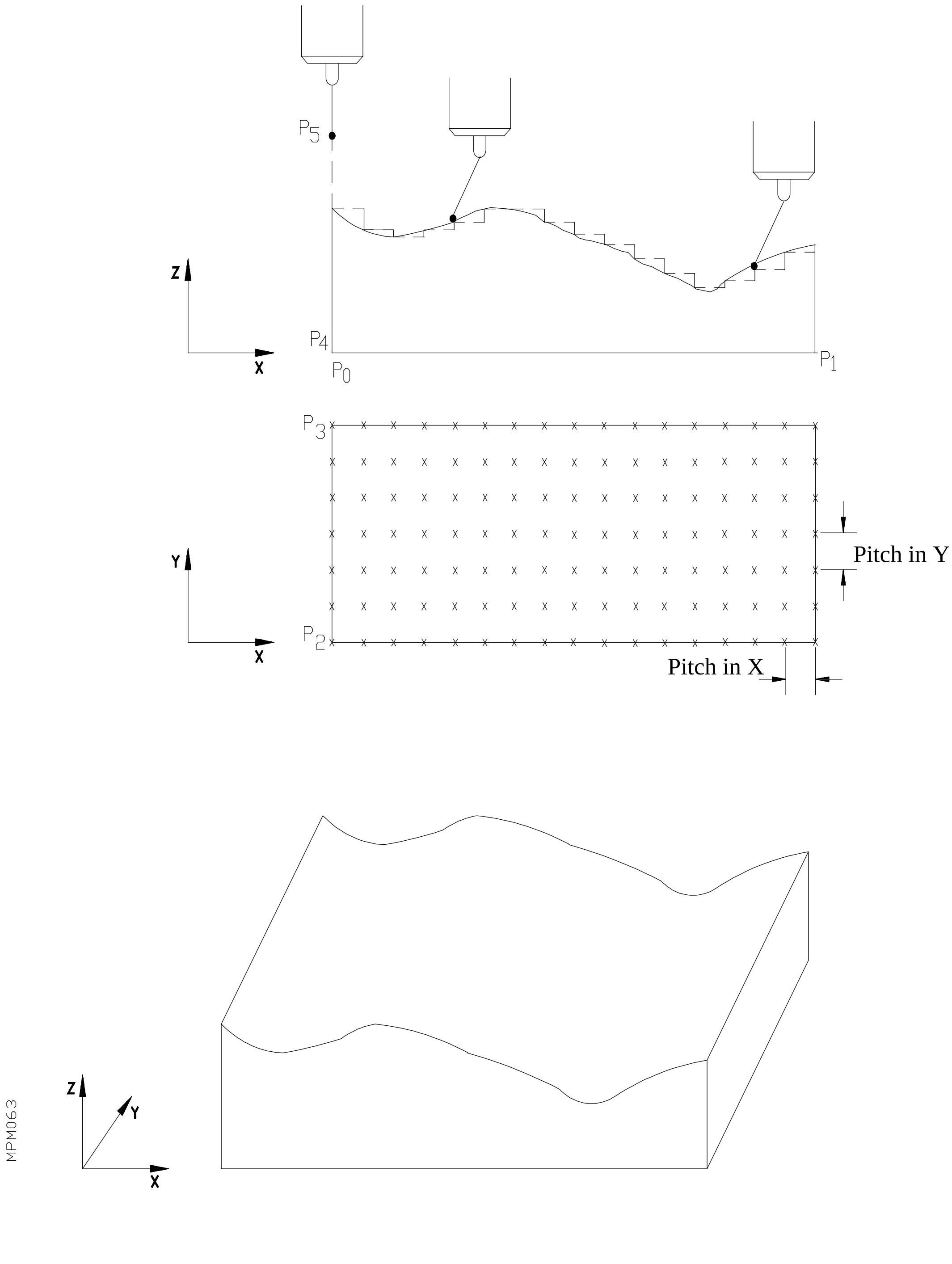
Creation of a program by copying the points of a part with a measuring probe (G75).

Calling parameters:

P0 = Minimum X value to sweep.
P1 = Maximum X value to sweep.
P2 = Minimum Y value to sweep.
P3 = Maximum Y value to sweep.
P4 = Minimum Z value to sweep.
P5 = Maximum Z value to sweep.
P6 = Maximum step value on X.
P7 = Maximum step value on Y.
P8 = Regular movement feedrate.
P9 = Probing movement feedrate.

Parameters used for calculations P10= Z axis increment for G75.

P11= Number of steps on X.
P12= Number of steps on Y.
P13= Starting point's X value.
P14= Starting point's Y value.
P15= Starting point's Z value.
P16= Step counter for X axis.
P17= Indicates which values must be loaded (0=XZ, 1=YZ)
P18= Current X value.
P19= Current Y value.
P99= Increase of Z for successive runs


P5
Z
X
P4
P0
P1
P3
Y
X
Pitch in Y
P2
Pitch in X
MPM063
Z
Y
X

Example in inches:

```
% 00075

N10   (digitizing along the X axis)
N20   G76 N12345 (Program to be loaded into computer)
N40   G76 F200 (Feedrate)
N50   P0=K0.5 (minimum X)
N60   P1=K11.5 (maximum X)
N70   P2=K0.3 (minimum Y)
N80   P3=K2.7 (maximum Y)
N90   P4=K0 (minimum Z)
N100  P5=K2.25 (maximum Z)
N110  P6=K0.05 (maximum step in X)
N120  P7=K0.05 (maximum step in Y)
N130  P8=K100 (regular movement feedrate)
N140  P9=K200 (probing feedrate)
N145  P99=K-0.0394 (Z successive runs)
N150  P10=P1F2P0 P11=P10F4P6 P12=F12P11 P11=F11P12
N160  G26 N170
N170  P11=P12F1K1 P6=P10F4P11
N180  P10=P3F2P2 P12=P10F4P7 P13=F12P12 P12=F11P13
N190  G26 N200
N200  P12=P13F1K1 P7=P10F4P12
N210  P10=P4F2P5 P10=P10F2K1
N220  P13=X P14=Y P15=Z P17=K0 P18=P0 P19=P2
N230  G7 G0 G90 XP0 YP2
N240  G76 G0 G90 XY
N250  ZP5
N260  G76 Z
N263  G76 G91 G Z-P99
N265  G76G92 ZP5
N270  G76 G1 G5
N280  G1 G91 G75 ZP10 FP9 (digitizing)
N290  G0 Z0.0394
N300  P16=K0
N310  G1 G91 G75 ZP10 FP9
N320  P17=F11K1
N330  G27 N380
N340  G76 YZ
N350  P17=K0
N360  G25 N390
N370  G76 XZ
N380  P16=P16F1K1 P18=P18F1P6 P11=F11P16
N390  G28 N430
N400  G90 XP18 FP8
N410  G25 N320
```

```
N420  P17=K1 P6=F16P6 P18=P18F1P6 P19=P19F1P7
N430  G90 YP19 FP8
N440  G25 N310.430.1
N450  P12=P12F2K1
N460  G27 N440
N470  G0 G90 ZP15
N480  G76 G0Z
N490  XP13 YP14
N500  G76 XY M30
N510  M30
```

After the execution of this program, the CNC will have generated and loaded into the computer the following P12345 program: N100 G1 F500

```
N101  G0 G90 X— Y—
N102  Z—
N103  G1 G5
N—    Y— Z—
N—    Y— Z—  Etc.
```

The sequence of the points must be logical so the tool can follow them while machining the same way as the probe did.

In the described example a grid sweeping pattern has been followed on the main plane **XY** with the probe touching along the **Z** axis.

If this sweeping pattern is not suitable for the model to be copied, other patterns can be used like concentric circles, etc. on any plane **XY**, **XZ**, **YZ** and even with the auxiliary axis **V**, **W**.

It is also possible to divide the surface in separate areas and define a different sweeping pattern for each area.

If the machining must be done in various passes, the program will have to be executed applying successive zero-offsets or changes in tool length compensation.

All preparatory functions (square corner, scaling factor) that will affect the whole program can be defined in a previous block.

The CNC reserves automatically 100 blocks. Geometrical functions can also be included in a G76 type block:

. G08 Arc tangent to the previous path.
. G09 Arc defined by three points.

With these functions it is possible to smoothen the point-to-point machining profile.

6.30.5. OTHER DIGITIZING EXAMPLES

1. Example G76: DIGITIZING ALONG THE Y AXIS

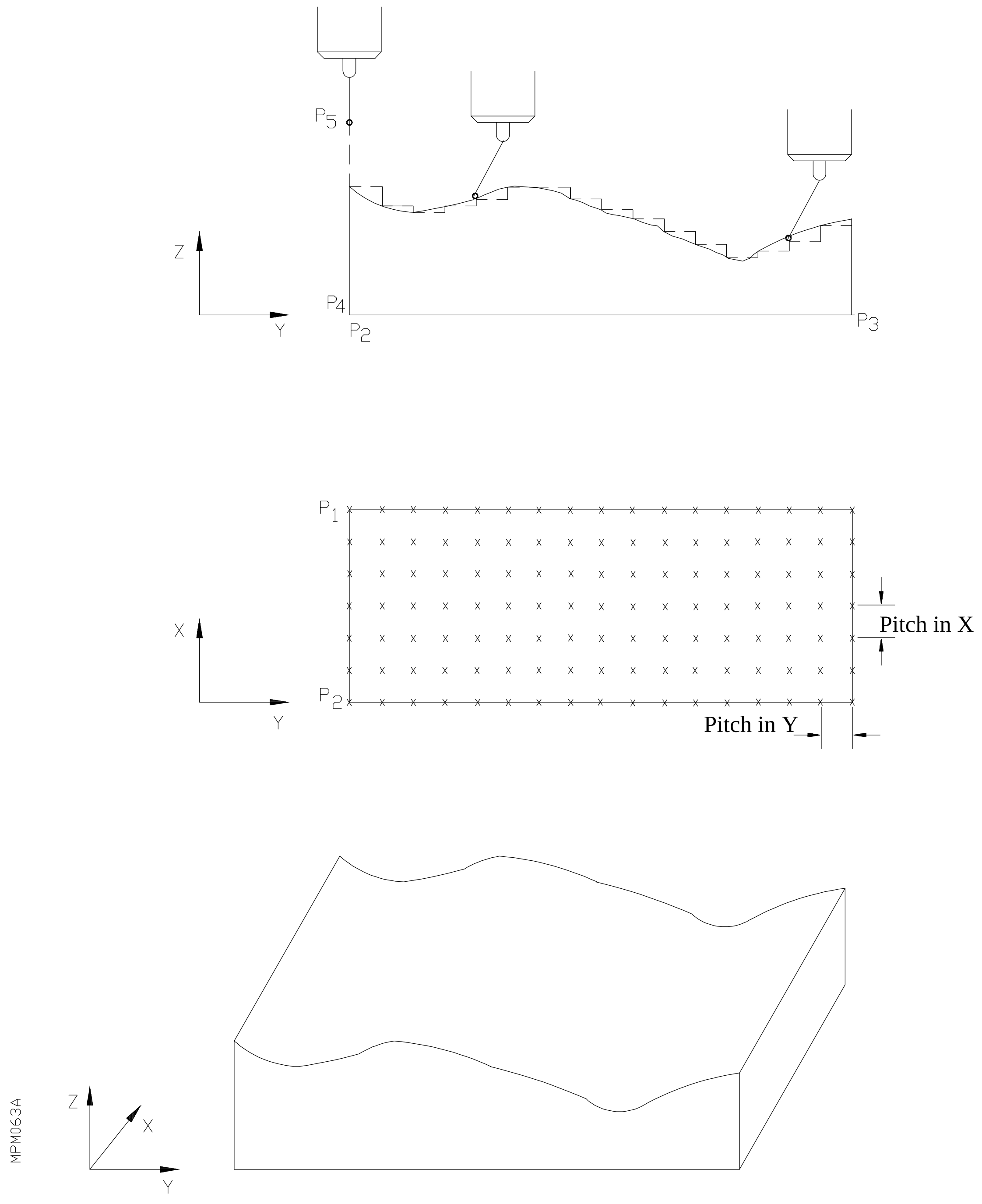
Creation of a program by copying the points of a part with a measuring probe (G75).

Calling parameters:

P0 = Minimum X value to sweep.
P1 = Maximum X value to sweep.
P2 = Minimum Y value to sweep.
P3 = Maximum Y value to sweep.
P4 = Minimum Z value to sweep.
P5 = Maximum Z value to sweep.
P6 = Maximum step value on X.
P7 = Maximum step value on Y.
P8 = Regular movement feedrate.
P9 = Probing movement feedrate.

Parameters used for calculations

P10= Z axis increment for G75.
P11= Number of steps on X.
P12= Number of steps on Y.
P13= Starting point's X value.
P14= Starting point's Y value.
P15= Starting point's Z value.
P16= Step counter for X axis.
P17= Indicates which values must be loaded (0=XZ, 1=YZ)
P18= Current X value.
P19= Current Y value.
P99= Increase of Z for successive runs


P5
Z
Y
P4
P2
P3
P1
X
Pitch in X
P2
Y
Pitch in Y
Z
X
Y
MPM063A

Example in inches:

%00076

```
N5     (digitizing along the Y axis)
N10    (P=Memory N = Computer)
N20    G76 N54321 (Program to be loaded into computer)
N30    (Machining conditions)
N40    G76 F200
N50    P0=K0.3 (minimum Y)
N60    P1=K2.7 (maximum Y)
N70    P2=K0.5 (minimum X)
N80    P3=K11.5 (maximum X)
N90    P4=K0    (minimum Z)
N100   P5=K2.25 (maximum Z)
N110   P6=K0.05 (maximum step in Y)
N120   P7=K0.05 (maximum step in X)
N130   P8=K500 (regular movement feedrate)
N140   P9=K200 (probing feedrate)
N145   P99=K-0.0394 (Z successive runs)
N150   P10=P1F2P0 P11=P10F4P6 P12=F12P11 P11=F11P12
N160   G26 N170
N170   P11=P12F1K1 P6=P10F4P11
N180   P10=P3F2P2 P12=P10F4P7 P13=F12P12 P12=F11P13
N190   G26 N200
N200   P12=P13F1K1 P7=P10F4P12
N210   P10=P4F2P5 P10=P10F2K2
N220   P13=X P14=Y P15=Z P17=K0 P18=P0 P19=P2
N230   G7 G0 G90 XP0 YP2
N240   G76 G0 G90 XY
N250   ZP5
N260   G76 Z
N263   G76 G91 G Z-P99
N265   G76G92 ZP5
N270   G76 G1 G5
N280   G1 G91 G75 ZP10 FP97 (digitizing)
N290   G0 Z0.0394
N300   P16=K0
N310   G1 G91 G75 ZP10 FP9
N320   P17=F11K1
```

```
N330  G27 N380
N340  G76 XZ
N350  P17=K0
N360  G25 N390
N370  G76 YZ
N380  P16=P16F1K1 P18=P18F1P6 P11=F11P16
N390  G28 N430
N400  G90 XP18 FP8
N410  G25 N320
N420  P17=K1 P6=F16P6 P18=P18F1P6 P19=P19F1P7
N430  G90 YP19 FP8
N440  G25 N310.430.1
N450  P12=P12F2K1
N460  G27 N440
N470  G0 G90 ZP15
N480  G76 G0Z
N490  XP13 YP14
N500  G76 XY M30
N510  M30
```

2. Example G76: CIRCULAR DIGITIZING

Creation of a program by copying the points of a part with a measuring probe (G75).

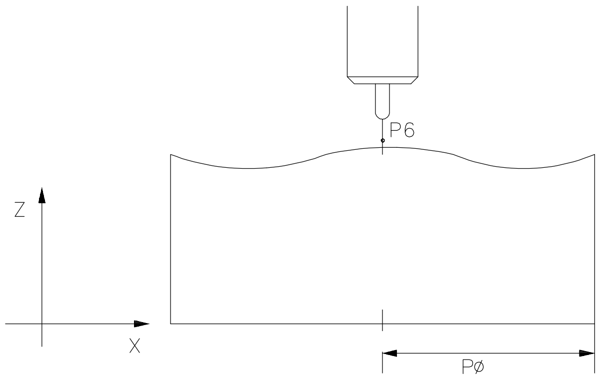
Calling parameters:

P0 = Radius value.
P1 = Pi π value.
P2 = Increment value of the radius to sweep.
P4 = Increment value of the arc to sweep.
P6 = Descent value in Z.
P8 = Regular movement feedrate.
P9 = Probing movement feedrate.

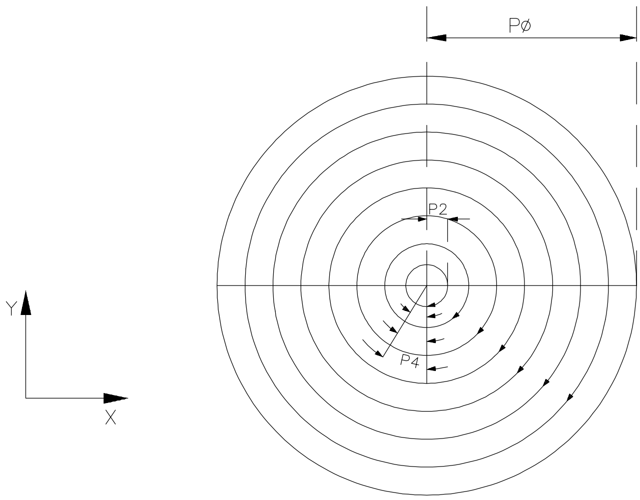
Parameters used for calculations

P13= Whole part of the angle.
P22= Accumulated radius value.
P31= Angle to rotate.

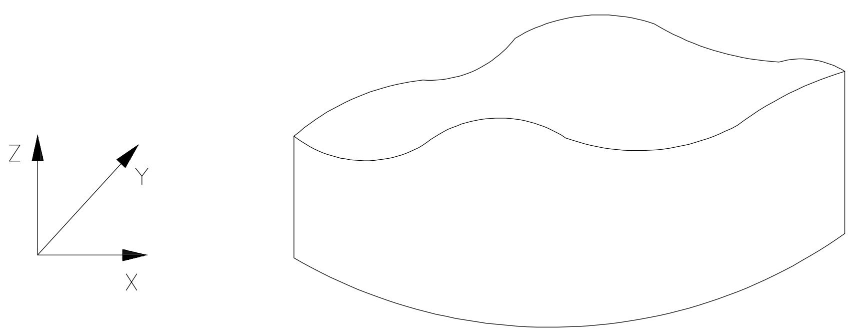
MP01



MP02



MP03



```
%00053

N1    (circular digitizing)
N5    (P=Memory N = Computer)
N6    G76 N90000 (Program to be stored in the computer)

N7     G92XYZ
N8     G76 XYZ
N9     G76 G91 Z-0.1 (Successive runs)
N10    G76 G92 Z0
N11    G76 G90
N12    P13=K0P31=K0 P22 = K0.4 (Radius)
N13    P1= K3.14159 (¶)
N14    P2=K0.05(Radius increment)
N15    P4=K0.05 (Arc increment)
N16    P22=P2 (Accumulated radius value)
N17    P6=K2.25 (Z axis descent)
N18    P8=K200 (Displacement feed)
N19    P9=K40 (Probing feed)
N20    G20 N1
N21    G90 G1 RP22
N30    G21 N1
N40    G20 N1
N50    G1 G5 G91 AP31 FP8
N55    G76 XY
N60    G28 N40
N70    P22=F11P0
N80    G28 N20
N82    G90 G Z
N84    G76 Z
N86    G90 G X Y
N87    G76 X Y
N88    G76 M30
N90    M30
N95    (Subroutines)
N100   G22 N1
N110   G1 G5 G90 G75 Z-P6 FP9 (Digitizing)
N120   G76 G90 Z
N125   P3=P3F1P31P3=F11K360
N130   G24
N140   G23 N1
N145   P99=K-0.040 (Z successive runs)
N150   P31=K2F3P1 P31=P31F3P22P31=P31F4P4
       P13=F12P31  P31=K360F4P31 P3=K0(Calculation of rotation angle)
N160   P22=P22F1P2 (Calculation of the radius by successive increments)
N170   G24
```

3. Example G76: DIAMETRIC DIGITIZING

Creation of a program by copying the points of a part with a measuring probe (G75).

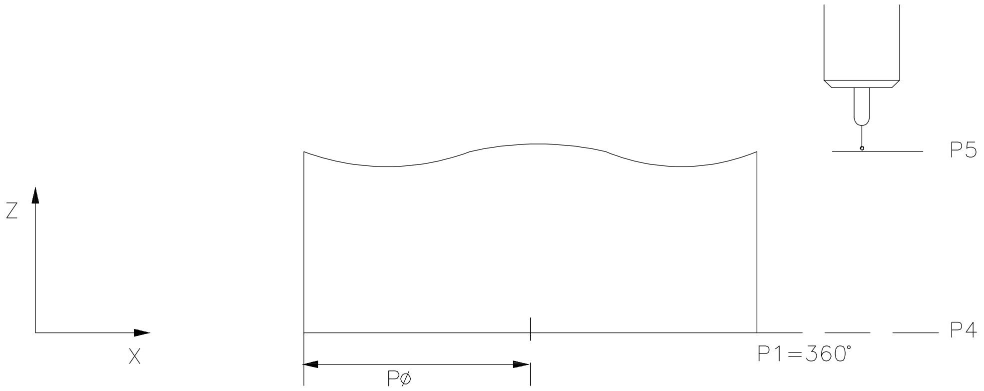
Calling parameters:

P0 = Radius of the part.
P1 = Initial angle fixed at 360 degrees.
P2 = Pitch of radius to sweep.
P3 = Pitch of angle to sweep.
P4 = Minimum Z value to sweep.
P5 = Maximum Z value to sweep.
P8 = Regular movement feedrate.
P9 = Probing movement feedrate.

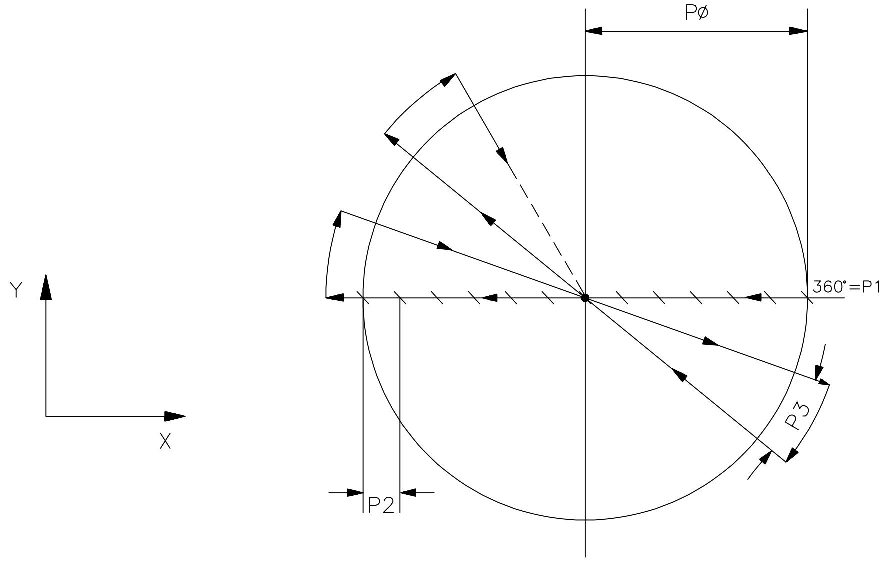
Parameters used for calculations

P10= Accumulated angular increment of the angle.
P11= Distance to travel in angle and absolute value.
P12= Absolute value of the distance to travel in angle.
P20= Accumulated value of radius.
P21= Total absolute radius to travel.
P22= No. of steps in radius
P23= Changed radius sign.
P30= Z limit for G75.
P99= Z increment for successive runs.

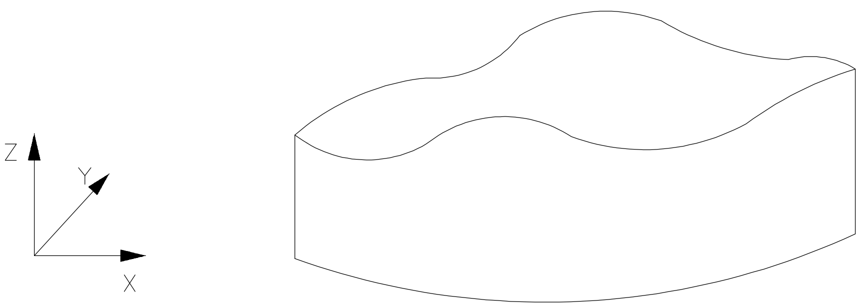
MP04



MP05



MP06



```
%00099

N0    G76 N10000 (Program to be stored in computer)
N5    (Diametric digitizing)
N10   G76 F500 S200 M3 (Machining conditions)
N20   P0=K67 (Radius of part)
N30   P1=K360 (Invariable initial angle)
N40   (radius pitch)
N50   P3=K3 (Angle pitch)
N70   P4=K-50 (minimum Z)
N80   P5=K13 (maximum Z)
N90   P8=K200 (regular movement feedrate)
N100  P9=K100 (probing feedrate)
N105  P99=K-1 (Z successive runs)
N110  P20=P0P21 P20F4P2 P22 =F12P21 P21=F11P22
N112  G26 N118
N114  P21=P22F1K1 P2=P20F4P21 (New radius increment)
N118  P30=P4F2P5 P30=F2K1
N120  P10=P1P11=P10F4P3P12=F12P11 P11=F11P12
N122  G26 N128
N126  P11=P12F1K1P3=P10F4P11(New angular increment)
N127  G1 X Y Z
N128  G93 I J
N130  G76 G93 I J
N140  G G90 Z P5
N150  G76 G1 G90 G5
N155  G76 Z
N156  G76 G91 Z-P99
N157  G76 G92 X P5
N160  G5 G1 G90 RP0 AP1 F500
N170  G76 X Y Z
N180  G1 G91 G75 Z P30 FP9 (Digitizing)
N190  G1 Z1
N200  G1 G91 G75 ZP30 FP9 (Digitizing)
N210  G76 X Y Z
N280  P20=P20F2P2 P23 = F16P0 P20=F11P23 (Compare with R)
N290  G28 N320
N300  G90 G1 RP20 AP10 FP8
N310  G25 N200
N320  P10=P10F2P3 P10=F11K180 (Compare angle)
N322  G28 N400
N325  G90 G5 RP20 AP10 FP8
N340  G1 G91 G75 ZP30 FP9
```

```
N350  G76 X  Y Z
N360  P20=P20F1P2 P20=F11P0 Compare with R)
N370  G29 N374
N372  G28 N380
N374  P10=P10F2P3 P10=F11K180 (Compare angle)
N376  G28 N400
N378  G25 N200
N380  G90 G1 RP20 AP10 FP8
N390  G25 N340
N400  G G90 ZP5
N410  G76 G Z
N420  G1 X Y
N430  G76 G1 X Y M30
N440  M30
```

4-Example G76 : PROFILE DIGITIZING

Creation of a program by copying the points of a part with a measuring probe (G75).

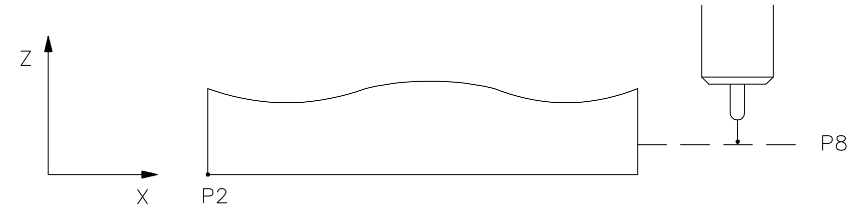
Calling parameters:

P2 = Minimum X value to sweep.
P3 = Minimum Y value to sweep.
P4 = Initial angle
P5 = Angle pitch
P6 = Regular movement feed rate.
P8 = Probing movement feed rate.

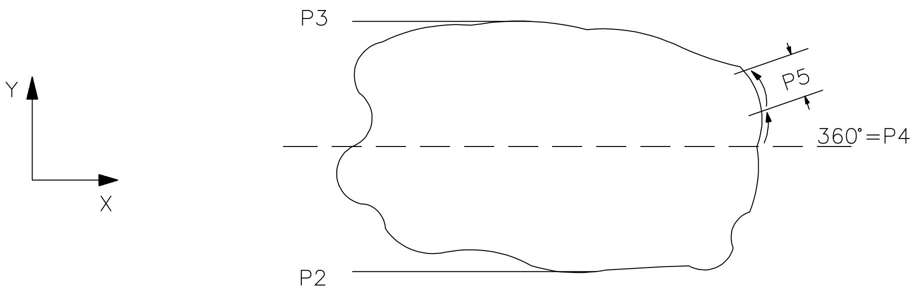
Parameters used for calculations

P10= Accumulated value of the angle.
P11= Distance to travel in absolute value and angle.
P12= Whole part of P11.

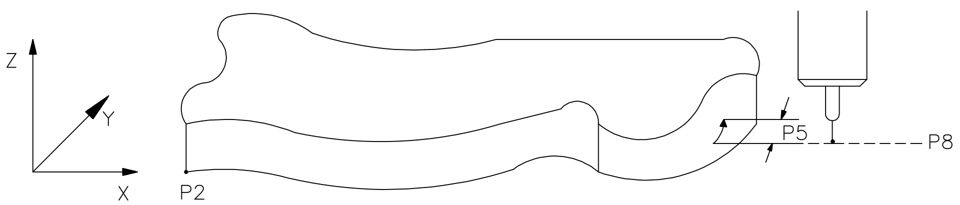
MP07



MP08



MP09



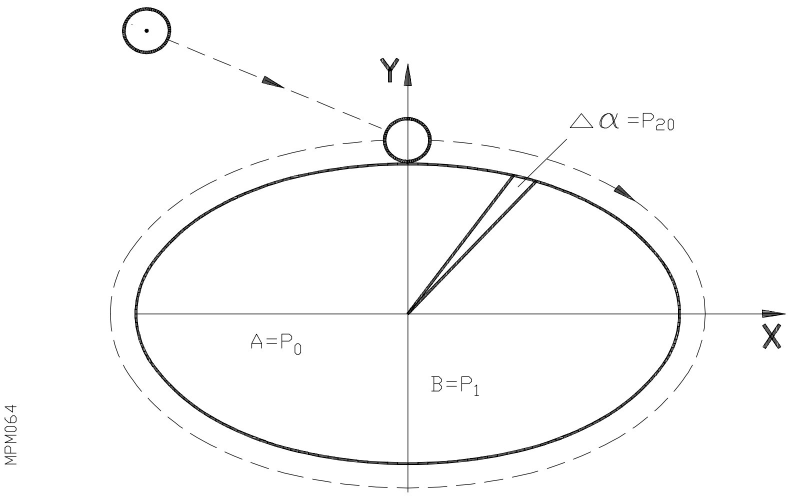
```
%00098

N0     G76 N98765
N10    (Digitizing of profile)
N20    (Machining conditions)
N30    G76 F500 S200 M3
N40    P2=K60 (Minimum X)
N50    P3=K0 (Minimum Y)
N60    P8=K-20(Probing Z)
N70    P4=K360 (Initial angle)
N80    P5=K1 (Angle pitch)
N90    P6=K600 (Regular movement feed rate)
N100   P11=P4F4P5 P12=F12P11 P11=F11P12
N110   G26 N130
N120   P11=P12F1K1 P5=P4F4P11
N130   G G90 X Y
N140   G93 I J
N150   G90 XP2 YP3
N160   G76 G G90 X Y
N170   G1 ZP8 F500
N180   G76 G1 Z FP6
N190   G5 G75 X Y (Digitizing)
N200   G76 X Y
N204   P4=P4F1P5
N210   G90 AP4
N220   P10=P4F1P4=P10 P10=F11K270
N230   G29 N250
N240   G25 N190
N250   G Z0
N260   G76 G Z0 M30
N270   M30
```

5. Example G76. CALCULATION OF THE POINTS OF AN ELLIPSE

This is a parametric program which, when executed, will calculate the different points of an ellipse and load them into a new program by means of G76 for later machining.

The calling parameters are the following:

P0 = Half the long axis (A).
P1 = Half the short axis (B).
P3 = Starting point's angle.
P20= Angular increment.



The XY coordinates of the various points that compose the ellipse are calculated according to the formula:

X = P0 SIN P3
Y = P1 COS P3

Let us suppose that the tool's starting point is X-100 Y100 and the X axis is programmed in radius. The calculation program is **P761**, shown below:

```
N10    G76 P00098
N20    P0=K20 P1=K10 P3=K0 P20=K2
N30    G76 G41 T1.1
N40    P4=F7P3 P5=F8P3 P6=P0F3P4 P7=P1F3P5
N50    G76 G0 G5 XP6 YP7 (ellipse's starting point)
N60    P3=P3F1P20 P4=F7P3 P5=F8P3 P8=P0F3P4 P9=P1F3P5
N70    P3=P3F1P20 P4=F7P3 P5=F8P3 P10=P0F3P4 P11=P1F3P5
N80    G76 G1 G9 XP10 YP11 IP8 JP9 F250
N90    P3=P3F1P20 P4=F7P3 P5=F8P3 P10=P0F3P4 P11=P1F3P5
N100   G76 G8 XP10 YP11
N110   P99=K176
N120   G25 N90.100.P99
N130   G76 G0 G40 X-100 Y100
N140   M30
```

When executing this program in **DRY RUN** program P00098 is generated and loaded into the CNC memory for later machining:

```
N100 G41 T1.1
N101 G0  G5 X— Y—
N102 G1  G9 X— Y— I— J— F250
N103 G8  X— Y—
N104 G8  X— Y—
N105   "   "
N ?   G0 G40 X-100 Y100
```

6.31. G77. SLAVING OF THE 4TH W AXIS (5TH V AXIS) WITH ITS ASSOCIATED AXIS
G78. CANCELLATION OF G77.

In 4 axis machines, after the execution of the **G77** function, the **4th axis (W)** is electronically coupled (slaved) with its associated axis (the axis which is indicated in machine parameter P11), until it is uncoupled (unslaved) by means of the execution of the **G78** function. I.e., when the **G77** function is active, the **4th axis (W)** will carry out the same movements that have been programmed for its associated axis.

While the **G77** function is active, movements of the **4th axis (W)** cannot be programmed. This application is useful in machines which have two spindles mounted on independent shafts.

In **5** axis machines, **G77** couples the **5th axis V** with the one indicated in machine parameter **P11**, equivalent to the indications for the **4th axis W**. The **G77** and **G78** functions are **MODAL**. Under starting conditions, after executing M02, M30, Reset or Emergency, the CNC assumes the **G78** function.

6.32. MACHINING CANNED CYCLES

This CNC features the following canned cycles:

G79 : User defined canned cycle
G81 : Drilling canned cycle
G82 : Drilling canned cycle with dwell
G83 : Deep drilling canned cycle
G84 : Tapping canned cycle
G85 : Reaming canned cycle
G86 : Boring canned cycle with G00 withdrawal
G87 : Rectangular pocket canned cycle
G88 : Circular pocket canned cycle
G89 : Boring canned cycle with G01 withdrawal

The canned cycles can be performed in any plane; so, when programming a canned cycle, it will be performed in the plane selected, the penetration being carried out on the axis perpendicular to that plane.

The **fourth axis (W)**, as well as the **5th axis (V)**, can be a part of the main plane or, if they are linear axes, they can be the perpendicular axis to this plane.

6.32.1. Zone of influence of the canned cycle

Once a canned cycle has been defined as described in the previous section, all the subsequent blocks programmed will be under the influence of that canned cycle until it is cancelled. In other words, each time a block is executed in which any movement of the axes is programmed, the machining corresponding to the canned cycle defined will be carried out automatically.

The structure of the blocks which are within the zone of influence of the canned cycle is normal except that **N2** can be programmed at the end of the block (number of times the block is repeated). If **N0** is programmed, the canned cycle will not be performed after the movement is carried out.

If there as a motionless block within the zone of influence of a canned cycle, the machining of that canned cycle will not be carried out, except in the calling block. If continued performance of the same canned cycle is desired with a change of any of the parameters (Z,I,J,K), the cycle has to be redefined.

6.32.2. Cancellation of canned cycles .

Programming the code **G80** in a block cancels any canned cycle that is active.

. When a canned cycle is defined, it cancels and replaces any others that are active.

. Canned cycles are also cancelled by means of **MO2**, **M30**, **RESET** or **EMERGENCY**.

. All canned cycles except **G79** are also cancelled by **G32**, **G53/G59**, **G74**, **G92**, **M02**, **M30**, **RESET** or **EMERGENCY** or when a new main plane has been selected with **G17**, **G18** or **G19**.

6.32.3. General considerations

. A canned cycle can be defined within a standard or parametric subroutine.

. Calling of standard or parametric subroutines can be performed from a block of the zone of influence of a canned cycle without involving cancellation of the canned cycle.

. The performance of the canned cycle does not affect the sequence of the preceding **G** functions or the direction of rotation of the spindle. A canned cycle may begin with either direction of rotation (M03,M04) and end with the same direction (this is not affected by the stops and reversal involved in the cycle).

. If the canned cycle begins with the spindle not running, the latter starts clockwise (M03) and continues rotating clockwise on completing the cycle.

. Defining a canned cycle cancels radius compensation. It is equivalent to G40.

. The execution of a canned cycle alters the value of the Arithmetic parameters **P70** to **P99**.

. In the block of definition of a canned cycle, the cycle relevant **G** will be cancelled if after it, **G02**,**G03**,**G08**,**G09** or **G33** (one of them) is programmed.

. When defining a canned cycle, except **G79**, either while functions **G02**,**G03** or **G33** are active or programming functions **G08** or **G09** in the same block, the CNC will display error **4**.

. Once a canned cycle has been defined, the functions **G02**,**G03**,**G08** or G09 can be programmed in the subsequent blocks.

6.32.4. G79. Canned cycle definition

By means of the function **G79**, the rank of canned cycle can be given to any parametric subprogram defined by the user (G23 N2); that means the blocks following the calling block (G79 N2 ...) are under the influence of the canned cycle until the function **G79** is cancelled. The calling block format is:

N4 G79 N2 P2=K— P2=K— ...

When reading a block programmed this way, the CNC will execute the **N2** parametric subprogram which will be identified by **G23 N2** either in any part of the program or in another program. In a calling block values may be assigned to the parameters (P2=K— P2=K—). If after this block, any other movement of axes is programmed, **N2** subprogram will be executed.

In the definition of a parametric subroutine (G23 N2) which is going to be called by the function **G79**, no other canned cycle can be programmed. Otherwise the CNC will display error **13**. Nevertheless, **G80** (end of canned cycle) can be programmed (must be by itself) in a block indicating the end of the subroutine. If the subroutine has more than one nesting level, G80 can only be programmed on the first level.

6.32.5. (G81,G82,G84,G85,G86,G89) canned cycle definition

The basic structure of the block in which one of these canned cycles is defined, is as follows:

N4 G8? G(98 or 99) (V+/-4.3) (W+/-4.3) X+/-4.3 Y+/-4.3 Z+/-4.3 I+/-4.3 K2.2 N2

N4 :Block number (0-9999).

G8? :Code of the canned cycle selected.

G98 : Withdrawal of the axis perpendicular to the main plane to the starting plane, after completing the machining of the hole.

G99 : Withdrawal of the axis perpendicular to the main plane up to the reference plane (approach), after completing the machining of the hole. Reference plane identifies a plane that is close to the part's surface.

X+/-4.3 :Their values have different meaning depending on the plane in which we are
Y+6-4.3 :operating.
Z+/-4.3 :
*(W+/-4.3):
*(V+/-4.3) :

<table>
<tr><th>MAIN PLANE</th><th>VALUE</th><th>MEANING</th></tr>
<tr><td>X/Y
G17</td><td>X+/-4.3
Y+/-4.3</td><td rowspan="3">They define the movement of the axes of the main plane necessary to position the tool in the center of the first machining. The values will be either absolute or incremental depending on the function on which we are operating (G90 or G91).
The movements will be carried out either in rapid or in F operation feedrate, depending on the function on which we are operating (G00 or G01).
That point can be also programmed in polar coordinates.</td></tr>
<tr><td>X/Z
G18</td><td>X+/-4.3
Z+/-4.3</td></tr>
<tr><td>Y/Z
G19</td><td>Y+/-4.3
Z+/-4.3</td></tr>
<tr><td>X/Y
G17</td><td>Z+/-4.3</td><td rowspan="3">It identifies the movement of the axis perpendicular to the main plane, from the starting plane up to the reference plane (approach). This movement will be executed in rapid (G00). The values will be either absolute or incremental, depending on the function on which we are operating (G90 or G91). This value must necessarily be programmed.</td></tr>
<tr><td>X/Z
G18</td><td>Y+/-4.3</td></tr>
<tr><td>Y/Z
G19</td><td>X+/-4.3</td></tr>
</table>

* If the 4th W axis or the 5th V axis is perpendicular to the main plane, it must be a linear axis. But if it is one of the axis of the main plane, it may also be a rotary axis.

I+/-4.3 : It defines the depth of the machining. With G90, the values are absolute, in other words, they are related to the origin of the axis perpendicular to the main plane. With G91 the values are incremental, that means, they are related to the reference plane (approach).

K2.2. : It defines the dwell from reaming the full machining depth until starting its withdrawal. A value may be programmed either within K0.00 (0.00 sec.) and K99.99 (99.99) or within 0.00 and 655.35, if it is programmed with a parameter (K P3).

The programming of this parameter is obligatory only in drilling cycle with dwell G82, if it is not programmed, the CNC will display error 44. In the other, canned cycles, if K parameter is not programmed, the CNC assumes the value of K0.

N2 : It defines the number of times that the block's execution is to be repeated.

Any value between N0 and N99 can be programmed, but, if the value is programmed with a parameter (N P3), the latter can have a value between 0 and 255. If parameter N is not programmed, the CNC assumes the value N1. Obviously, the programming of values of N higher than 1 makes sense when operating on G91, in other words, if the axis movement values are incremental, since otherwise the machinings will be repeated at the same point. When programming the same canned cycle a number of times, only F, S and M functions will be executed in the cycle calling block.

A more detailed explanation of the (G81,G82,G84,G85,G86 and G89) canned cycle is subsequently given, supposing that the main plane is the one formed by X and Y axes and that Z is the axis of the tool.

6.32.5.1. G81. Drilling canned cycle

The operations and movements of the tool (Z axis) are as follows:

. If the spindle was previously running, it continues rotating in the same direction. If it was not running, it starts clockwise (M03).

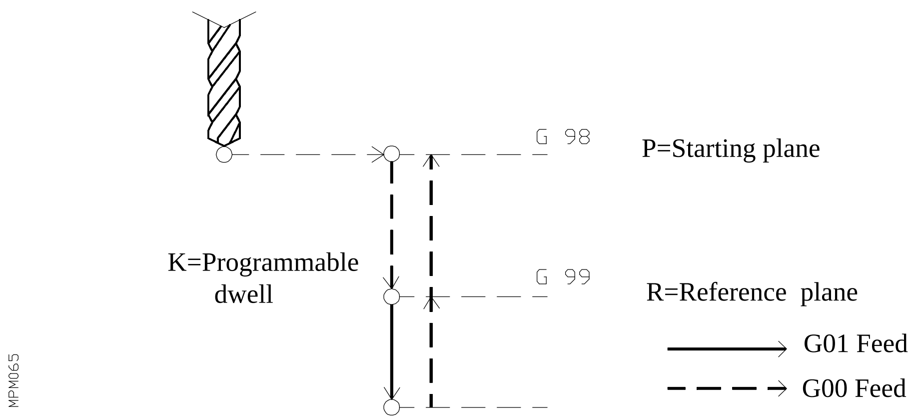
. Rapid movement of the **Z** axis from the starting plane to the reference (approach) plane.

. Movement at the working feedrate of the **Z** axis to the full machining depth. . Dwell, if K has been programmed.

. Rapid withdrawal of the tool (Z axis) to the reference (approach) plane if **G99** is programmed.

. Rapid withdrawal to the starting plane if **G98** is programmed.

(G81) DRILLING



Example G81

Drilling four holes 20 mm deep (polar coordinates).

Let us suppose that:

. The distance between the reference plane and the surface of the part is 2 mm.

. The starting point is X0,Y0,Z0 and the spindle is not running.

```
N0   G81  G98   G00  G91  X250  Y350  Z-98  I-22  F100  S500 N1
N5   G93  I250  J250
N10 A-45 N3
N15  G80 G90  X0    Y0
N20 M30
```

First block (N0)

G81 : Defines the drilling canned cycle.

G98 : Defines the tool withdrawal (Z axis) to the starting plane.

G00 : Defines the movement of the X and Y axes as being rapid.

G91 : Defines the dimensions as being incremental.

X( )
Y( ) : Movement in millimeters on these axes.

Z( ) : Tool movement in millimeters (Z axis) from the starting plane to the reference plane.

I( ) : Movement in millimeters from the reference plane to the full machining depth.

F( ) : Working feedrate in mm/min.

S( ) : Spindle rotation speed in rev/min.

N( ) : Number of times the block is repeated.

Second block (N5)

G93 : Defines the origin of polar coordinates (polar origin).

I( ) : Coordinate values (abscissa, ordinate) of the polar origin.
J( )

Third block (N10)

A( ) : Incremental angular movement referred to the polar origin defined in N5.

N( ) : Number of times the block is repeated.

Fourth block (N15)

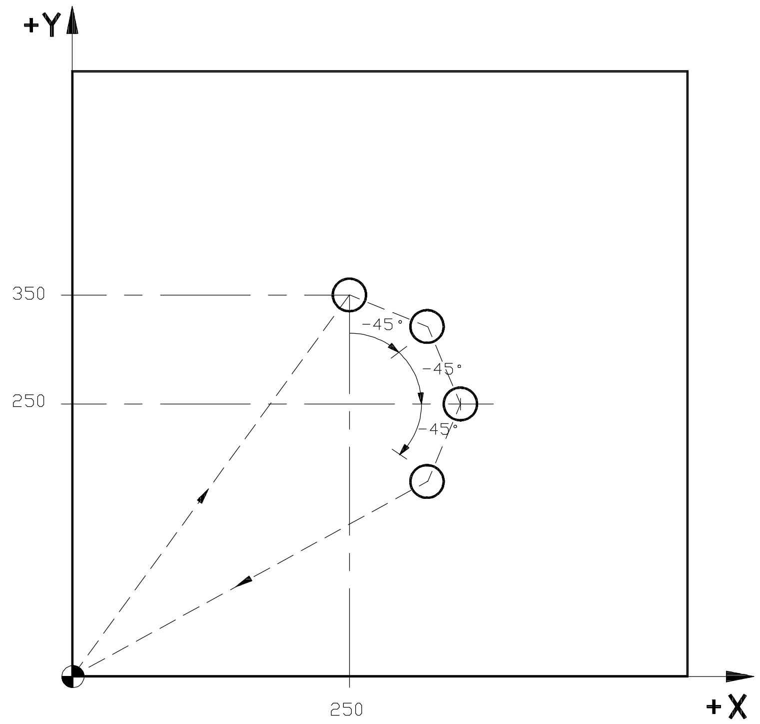
G80 : Cancellation of the canned cycle.

G90 :Defines the X and Y dimensions as being absolute.

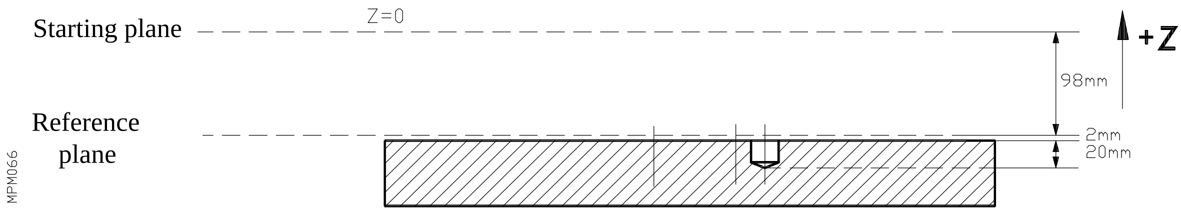
X( ) :Absolute coordinate values of these.
Y( )

Fifth block (N20)

M30 : End of program, with return to the first block.



+Y
350
250
-45°
-45°
-45°
250
+X



Starting plane
Z=0
+Z
98mm
Reference
plane
2mm
20mm
MPM066

Sequence and explanation of operations

1. The X moves in rapid to point X250, and the Y axis to point Y350.
2. The spindle starts rotating clockwise (M03) at 500 rev/min.
3. The Z axis moves 98 mm in rapid to Z-98 (reference plane).
4. The Z axis moves a further 22 mm at the working feedrate (F100) to point Z-120 (full drilling depth).
5. The Z axis withdraws 22 mm in rapid to the starting plane (Z 0).
6. The X,Y axes move in rapid to a point located at 45º from the previous position, along a circle centered on X250 Y250 and radius 100 (distance from the first hole to the polar origin).
7. Operations 3,4 and 5 are repeated.
8. Operations 6 is repeated.
9. Operations 3,4 and 5 are repeated.
10. Operation 6 is repeated.
11. Operation 3,4 and 5 are repeated.
12. The X,Y axes move in rapid to point X0,Y0.
13. End of program. The spindle stops running.

A different program for this example could be the following: Polar origin X0 Y0.

```
N0  G81 G98 G00 G91 R430.116  A54.462 Z-98 I-22 F100 S500 N1
N5  G93 I250 J250
N10 A-45 N3
N15 G80 G90  X0 Y0
N20 M30
```

6.32.5.2. G82. Drilling canned cycle with dwell

The operations and movements of the tool (Z axis) are as follows:

. If the spindle was previously running, it continues rotating in the same direction. If it was not running, it starts clockwise (M03).

. Rapid movement of the Z axis from the starting plane to the reference (approach) plane.

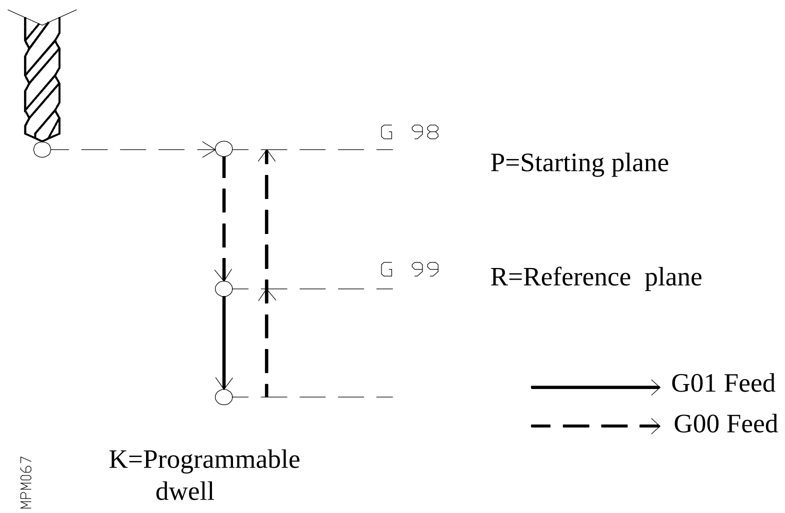
. Movement at the working feedrate of the Z axis to the full machining depth.

. Dwell. Any time between 0.00 and 99.99 seconds may be programmed, unless it is programmed using a parameter (KP3) in which case the limits are 0.00 and 255. **In this cycle the programming of K dwell is obligatory**.

. Rapid withdrawal of the Z axis to the reference plane if G99 is programmed.

. Rapid withdrawal of the Z axis to the starting plane if G98 is programmed.

(G82) DRILLING WITH DWELL



Example G82:

Drilling four holes 20 mm deep.

Let us suppose that:

. The distance between the reference plane and the part's surface is 2 mm.

. The starting point is X0, Y0, Z0 and the spindle is not running.

```
N0   G82  G99 G00  G91 X50  Y50  Z-98 I-22 K1.5 F100 S500 N3
N5   G98  G90 G00  X500 Y500 N1
N10 G80 G00 X0   Y0
N15 M30
```

First block (N0)

G82 : Defines the drilling canned cycle with dwell.

G99 : Defines the withdrawal of the tool (Z axis) to the reference plane.

G00 : Defines the movement of X and Y axes as being rapid.

G91 : Defines the X,Y,Z,I dimensions as being incremental.

X( ) : Movement in millimeters of these axes.
Y( )

Z( ) : Movement in millimeters of the tool (Z axis) from the starting plane to the reference one.

I( ) : Movement in millimeters from the reference plane to the full machining depth.

K( ) : Defines the dwell in seconds.

F( ) : Working feedrate in millimeters/min.

S( ) : Spindle rotation speed in rev/min.

N( ) : Number of times the block is repeated.

Second block (N5)

G98 : Defines the withdrawal of the tool (Z axis) to the starting plane.

G00 : Defines the X and Y axes movement as being in rapid.

G90 : Defines the X and Y dimensions as being absolute.

X( ) : Absolute coordinates of these axes.
Y( )

Third block (N10)

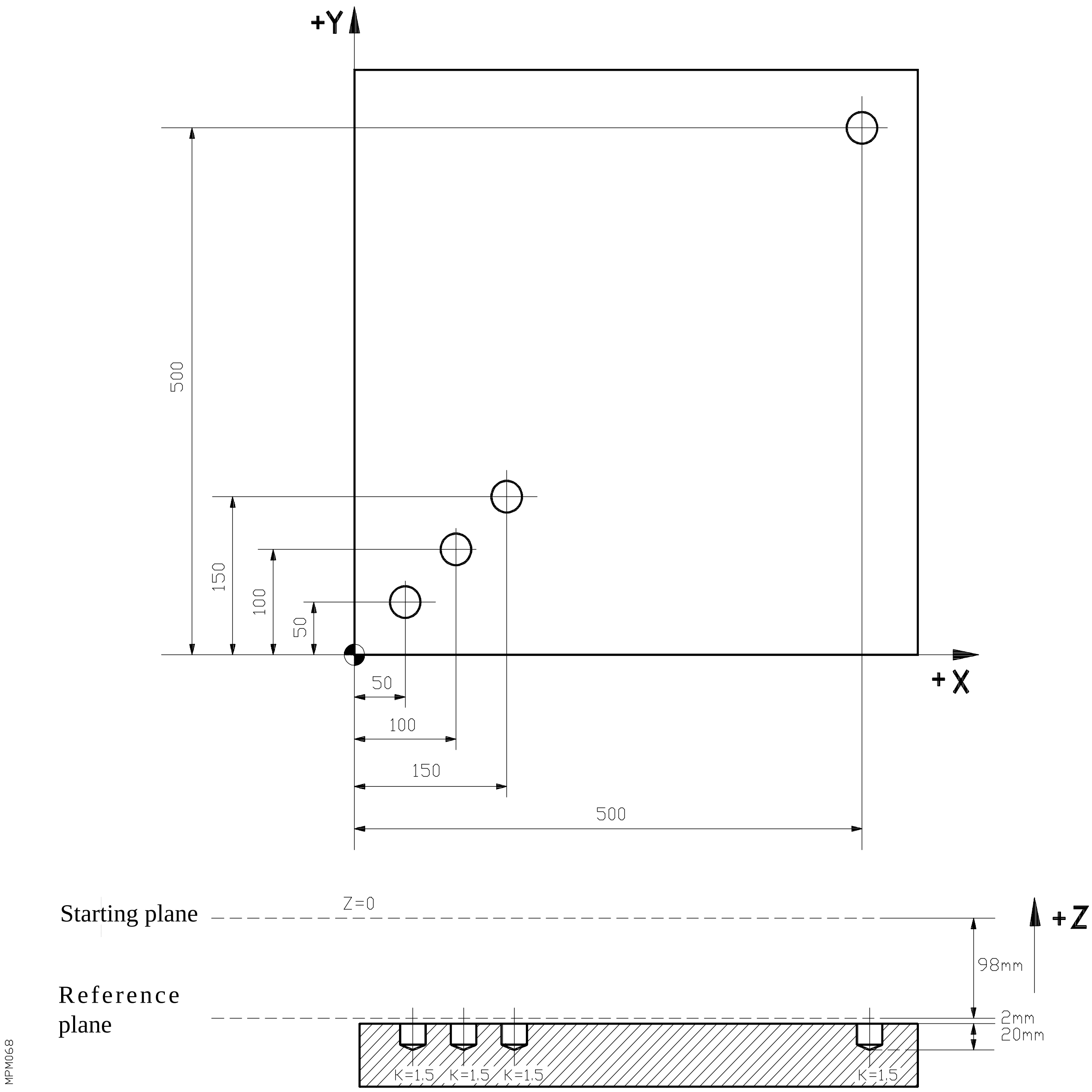
G80 : Canned cycle cancellation.

G00 : Defines the X and Y axes movement as being in rapid.

X( ): Absolute coordinates of these values.
Y( )

Fourth block (N15)

M30 : End of program and return to the first block.


+Y
500
150
100
50
50
100
150
500
+X
Starting plane
Z=0
+Z
98mm
Reference
plane
2mm
20mm
K=1.5
K=1.5
K=1.5
K=1.5
MPM068

Sequence and explanation of the operations

1. The X and Y axes move 50 mm in rapid to point X50,Y50.
2. The spindle starts rotating clockwise (M03) at a speed of 500 rev/min.
3. The Z axis moves 98 mm in rapid to Z-98 (reference plane).
4. The Z axis moves a further 22 mm in working feedrate (F100) to point Z-120 (full drilling depth).
5. Dwell 1.5 seconds.
6. The Z axis withdraws 22 mm in rapid to the reference plane (Z-98).
7. The X and Y axes move 500 mm in rapid to point X100, Y100.
8. Operations 4,5 and 6 are repeated.
9. The X and Y axes move 5 mm in rapid to point X150, Y150.
10. Operations 4,5 and 6 are repeated.
11. The X and Y axes move in rapid to point X500,Y500.
12. Operation 4 repeated.
13. The Z axis withdraws 120 mm in rapid to the starting plane (Z0).
14. The X and Y axes move in rapid to point X0,Y0.
15. End of program. The spindle stops running.

6.32.5.3. G84. Tapping canned cycle

The operations and movements of the tool (Z axis) are as follows:

. If the spindle was previously running, it continues to rotate in the same direction. If it was not running, it starts clockwise (M03).

. Rapid movement of the Z axis from the starting plane to the reference (approach) plane.

. Movement at the working feedrate of the Z axis to the full machining depth.

. Whether the spindle stops running or not (M05), depends on the value given to the machine-parameter P607(2).

. Dwell. Any time between 0.00 and 99.99 seconds may be programmed unless it is programmed using e parameter (KP3) in which case the limits are 0.00 and 655.35 seconds.

. Reversal of spindle rotation.

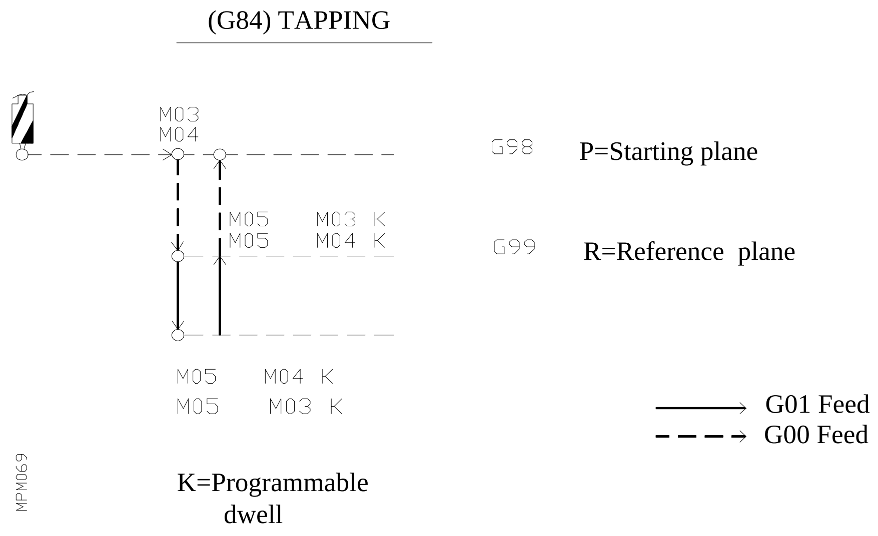
. The Z axis withdraws at the working feedrate to the reference plane.

. The spindle stops running or not (M05), depending on the value given to the machine parameter P607(2).

. Dwell (same value as programmed above).

. Reversal of spindle rotation.

. Rapid withdrawal of the Z axis to the starting plane if G98 is programmed.



Atention:

During the tapping canned cycle (G84), the feedrate is 100% regardless of the position of the **FEEDRATE** knob. also, the spindle speed (S) cannot be changed from the front panel keys during the movement of the axis perpendicular to the main plane.

Example

Tapping four holes, 20 mm deep.

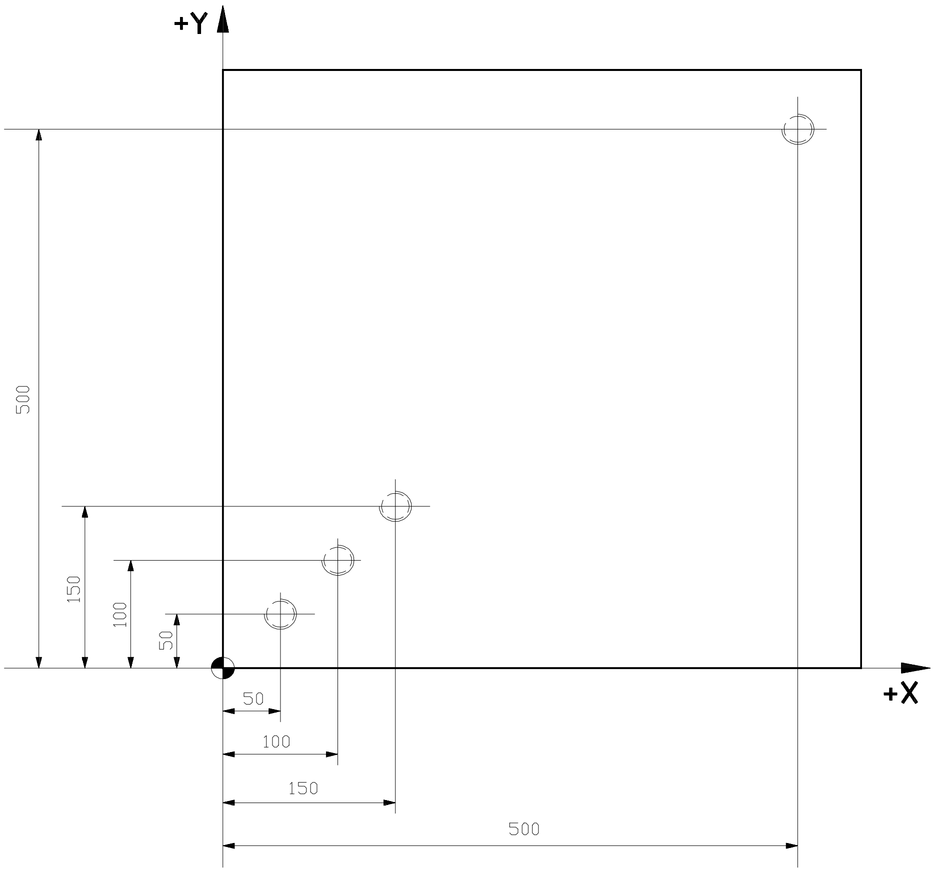
Let us suppose that:

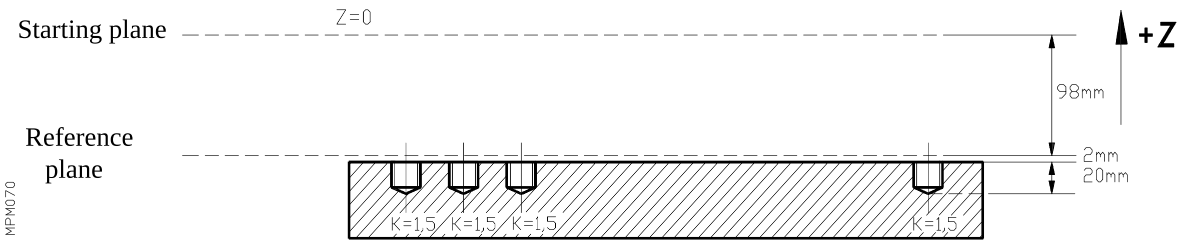
. The working plane is the one formed by X and Y axes.

. The distance between the reference plane and the surface of the part is 2 mm.

. The starting point is X0,Y0,Z0 and the spindle is not running.

```
N0   G84 G99 G00 G91 X50  Y50 Z-98 I-22 K1.5 F350 S500 N3
N5   G98 G90 G00 X500 Y500  N1
N10 G80 G00 X0   Y0
N15 M30
```


+Y
+X
500
150
100
50
50
100
150
500


Starting plane
Z=0
+Z
98mm
Reference plane
2mm
20mm
K=1,5
K=1,5
K=1,5
K=1,5
MPM070

Sequence and explanation of operations

1. The X and Y axes move 50 mm in rapid to point X50,Y50.
2. The spindle starts rotating clockwise (M03) at 500 rev/min.
3. The Z axis moves 98 mm in rapid to the reference plane (Z-98).
4. The Z axis moves at the working feedrate (F350) to point Z-120 (full machining depth).
5. The spindle stops running (M05).
6. Dwell of 1.5 seconds.
7. Reversal of spindle rotation.
8. The Z axis withdraws 22 mm at the working feedrate to the reference plane (Z-98).
9. The spindle stops running.
10. Dwell of 1.5 seconds.
11. Reversal of spindle rotation.
12. The X and Y axes move 50 mm in rapid to point X100,Y100.
13. Operations 4 to 11 are repeated.
14. The X and Y axes move 50 mm in rapid to point X150 Y150.
15. Operations 4 to 11 are repeated.
16. The X and Y axes move in rapid to point X500,Y500.
17. Operations 4 to 11 are repeated.
18. The Z axis withdraws 98 mm in rapid to the starting plane (Z0).
19. The X and Y axes move in rapid to X0,Y0.
20. End of program (spindle stops running).

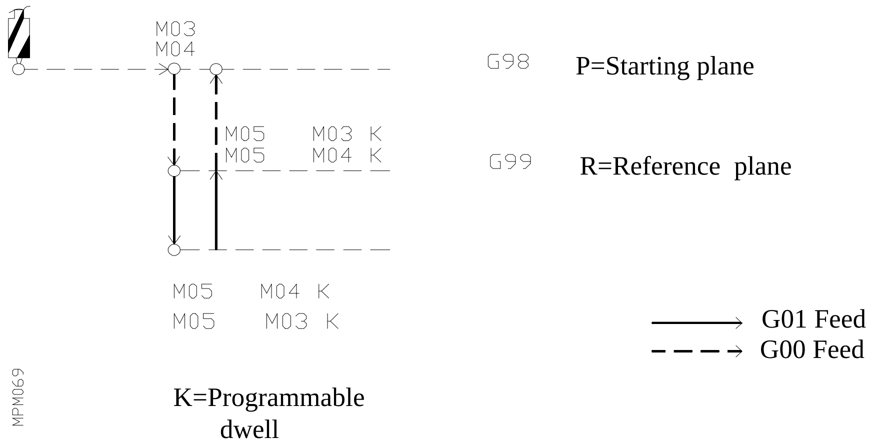
6.32.5.4. G84 R.   *Rigid Tapping canned cycle*

It is similar to the regular tapping canned cycle (G84) except that, in this case, the spindle is interpolated with the tapping axis.

Also, the regular tapping cycle (G84) requires a special tap holder (with a clutching system) while the rigid tapping cycle can be performed with any regular tap.

When programming a rigid tapping cycle (G84 R), the F value must be in mm/minute or inches /minute and the spindle speed in rpm.

(G84) TAPPING



Example and operating method:

We would like to make two taps 90 mm deep with a pitch of 2 mm at positions X10 Y10 and X20 Y20 the reference plane being at Z-10mm.

```
N00 G17 S1000 M3                                ; Main plane XY
N10 G84 R G98 G91 X10 Y10 I-100 K1 F1000 S500 N2 ; Rigid tapping canned cycle
N20 G80                                         ; End of canned cycle
N30 M30                                         ; End of program
```

Sequence of operation:

1. The spindle is turning in open loop at 1000 rpm in the M3 direction.

2. The spindle slows down to 500 rpm still in open loop. If this involves a range change, the CNC executes the corresponding M function.

   If the spindle were no turning, the CNC would execute an M3.

   The X and Y axes move to position X10 Y10 in G00 (rapid).

3. G00 move of the Z axis to the reference plane Z-10. The spindle goes into closed loop.

   If it is the first tap (that is, the spindle goes from open to closed loop) and if parameter "P625(1)=1" for the start of the thread to be synchronized with the spindle marker pulse (Io), the CNC will home the spindle.

   On the rest of the taps, as long as neither G80, M02, M03, M4 nor M30 functions are executed, the CNC will not reference (home) the spindle.

4. Tap-in movement along the Z axis down to Z-110. The spindle is interpolated (G01) with the Z axis at F1000.

5. Dwell at the bottom of the thread.

   The CNC executes an M4 and the spindle starts turning in the opposite direction.

6. Tap-out movement. The Z axis returns to the reference plane Z-10. The spindle is interpolated (G01) with the Z axis at F1000.

7. The CNC executes an M03 and the spindle recovers its turning direction.

   Rapid move (G00) to the starting plane (G98).

8. The X and Y axes move in G00 up to the next tapping position X20 Y20.

9. Same as point 3; but, without homing the spindle.

10. Same as point 4.

11. Same as point 5.

12. Same as point 6.

13. Same as point 7.

When executing function G80, the spindle goes into open loop turning at 500 rpm.

Also, the spindle goes into open loop whenever an M02, M03, M04 or M30 is executed or when a RESET is pressed or an error situation occurs.

6.32.5.5. G85. Reaming canned cycle

Same as G81 except that the withdrawal of the axis perpendicular to the main plane, from the full machining depth to the reference plane, is carried out at the working feedrate.

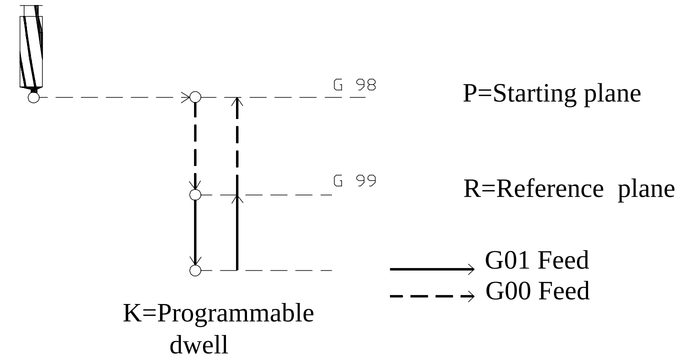
6.32.5.6. G86. Boring canned cycle with G00 withdrawal

Same as G81 except that after reaching the full machining depth the spindle stops running before the axis perpendicular to the main plane withdraws. On completion of the G00 withdrawal, the spindle starts running again in the same direction as before.

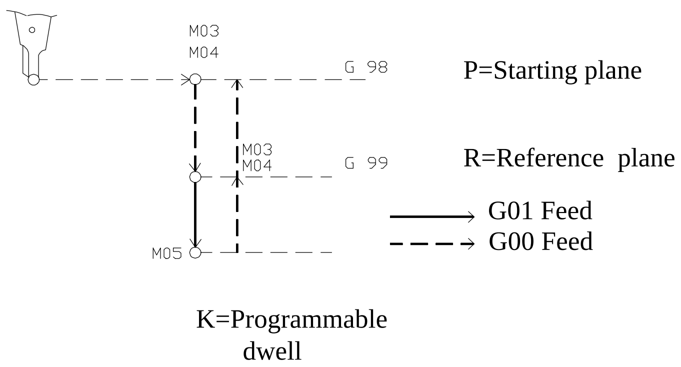
6.32.5.7. G89. Boring canned cycle with G01 withdrawal

Same as G82 except that after reaching the full machining depth the withdrawal to the reference plane is carried out at the working feedrate.

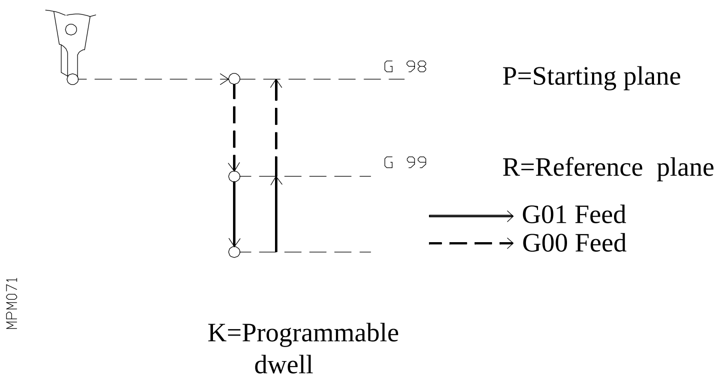
(G85) REAMING



(G86) BORING WITH WITHDRAWAL IN G00



(G89) BORING WITH WITHDRAWAL IN G01



6.32.6. Deep hole drilling canned cycle definition. G83

This canned cycle may be programmed in two different ways:

Format a) **N4 G83 G98/G99 (v+/-4.3)(W+/-4.3) X+/-4.3 Y+/- 4.3 Z+/-4.3 I+/-4.3 J2 N2**

Format b) **N4 G83 G98/G99 (W+/-4.3) X+/-4.3 Y+/-4.3 Z+/- 4.3 I+/-4.3 B+/-4.3 C+/-4.3 D+/-4.3 H4.3 J2 K2.2 L4.3 R(0.000/500) N2**

The meaning of the values of the format a) is as follows:

N4 :Block number (0/9999).
G83 :Code of the deep drilling canned cycle.
G98 :Withdrawal of the axis perpendicular to the main plane, to the starting plane,after completing the machining.
G99 :Withdrawal of the axis perpendicular to the main plane, to the reference plane, after completing the machining.
X+/-4.3 :These values have different meaning, depending on the main plane in which the cycle is being carried out.
Y+/-4.3 :
Z+/-4.3 :
*(W+/-4.3):
*(V+/-4.3) :

<table>
<tr><th>MAIN PLANE</th><th>VALUE</th><th>MEANING</th></tr>
<tr><td>X/Y
G17</td><td>X+/-4.3
Y+/-4.3</td><td rowspan="3">They define the movement of the axes of the main plane necessary to position the tool in the center of the first machining. The values will be either absolute or incremental depending on the function on which we are operating (G90 or G91).
The movements will be carried out either in rapid or in F operation feedrate, depending on the function on which we are operating (G00 or G01).
That point can be also programmed in polar coordinates.</td></tr>
<tr><td>X/Z
G18</td><td>X+/-4.3
Z+/-4.3</td></tr>
<tr><td>Y/Z
G19</td><td>Y+/-4.3
Z+/-4.3</td></tr>
<tr><td>X/Y
G17</td><td>Z+/-4.3</td><td rowspan="3">It identifies the movement of the axis perpendicular to the main plane, from the starting plane up to the reference plane (approach). This movement will be executed in rapid (G00). The values will be either absolute or incremental, depending on the function on which we are operating (G90 or G91).
This value must necessarily be programmed.</td></tr>
<tr><td>X/Z
G18</td><td>Y+/-4.3</td></tr>
<tr><td>Y/Z
G19</td><td>X+/-4.3</td></tr>
</table>

* If the 4th W axis or the 5th V axis are perpendicular to the main plane, it must be a linear axis. But if it is one of the axis of the main plane, it may also be a rotary axis.

I+/-4.3 :Identifies the value of each step of machining and it is always an incremental value.

J2 :Identifies the number of steps required to perform the machining. A value within J00 and **J99** is programmed.

N2 : Indicates the number of times the execution of a block is to be performed. A value between N0 and N99 can be programmed but, if the value is programmed with a parameter (N P3), it can have a value between 0 and 255. If the parameter **N** is not programmed, the CNC assumesthe value N1.

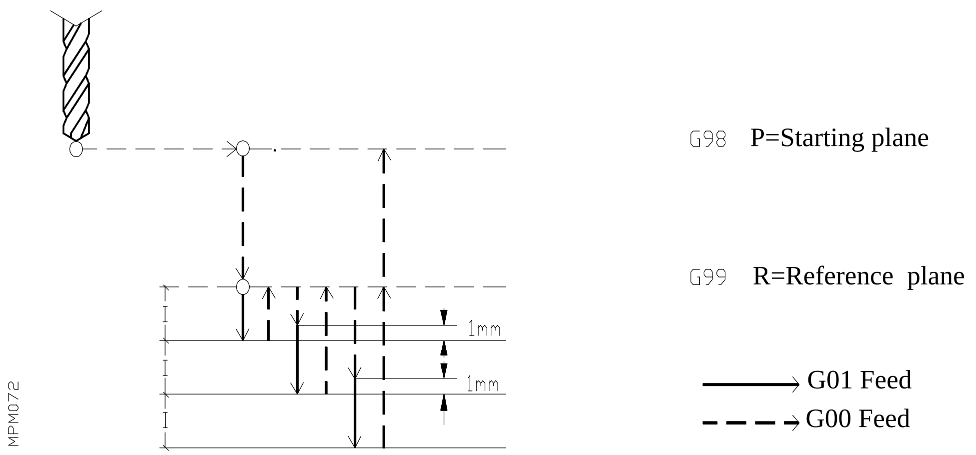
Obviously, the programming of values of **N** higher than 1 makes sense if we are operating on **G91**, that means that the values of the movement of the axes are incremental, since otherwise, the machinings will be repeated at the same point. When programming the same canned cycle a number of times, only the **F**,**S** and **M** functions are executed in the cycle calling block.

The operations and movements of the tool, in the cycle G83 programmed in format identified as a), are as follows:

Let us suppose that the axis of the tool is the Z axis.

1. If the spindle was previously running, it keeps on running in the same direction. If it was not running, it starts clockwise (M03).
2. Movement in rapid of Z axis from the starting plane to the reference one.
3. Movement at the working feedrate to the programmed incremental depth (I).
4. Withdrawal in rapid to the reference plane.
5. Movement in rapid of the Z axis to a point 1 mm higher than the previous incremental depth reached (I).
6. Movement at the working feedrate to 2I.
7. Withdrawal in rapid to the reference plane.
8. Operations 4),5),6) and 7) are repeated as many times as is programmed by J2. The maximum possible is 99 times, reaching the successive depths 3I, 4I ..., up to the total JI.
9. Withdrawal in rapid of the Z axis to the reference plane, if G99 is programmed withdrawal in rapid of the Z axis to the starting plane, if G98 is programmed.

(G83) DEEP HOLE DRILLING



Example:

Drill two holes 64 mm deep.

Let us suppose:

. The main plane is the one formed by X and Y axes.

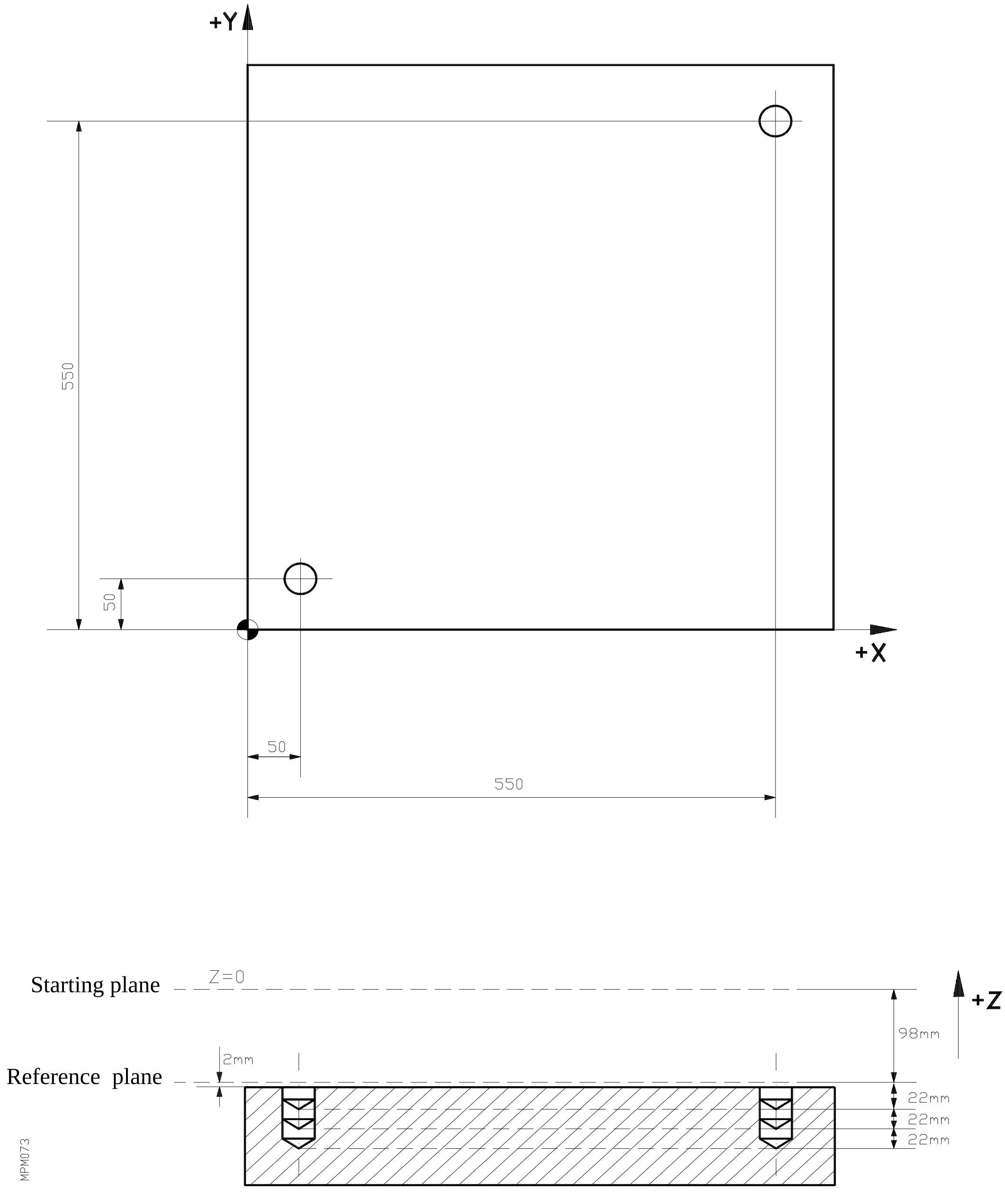
. The distance between the reference plane and the part's surface is 2 mm.

. The starting point of the tool is X0,Y0,Z0 and the spindle rotation direction is c.c.w (M04).

```
N0   G83  G99 G00 G90 X50  Y50  Z-98 I-22 J3 F100 S500 N1
N5   G98  G00 G91 X500 Y500 N1
N10  G00  G80 G90 X0   Y0
N15  M30
```

Sequence and explanation of operations

1. The X and Y axes move 50 mm in rapid to point X50, Y50.
2. The spindle keeps on rotating ccw (M04) and its speed from now on is 500 rev/min.
3. The Z axis moves in rapid to the reference plane (Z- 98).
4. The Z axis moves a further 22 mm at the working feedrate to the point Z-120.
5. The Z axis withdraws in rapid to the reference plane (Z-98).
6. The Z axis moves 21 mm in rapid to point (Z-119).
7. The Z axis moves 23 mm at the working feedrate to the point Z-142.
8. The Z axis withdraws in rapid to the reference plane (Z-98).
9. The Z axis moves 43 mm in rapid to point Z-141.
10. The Z axis moves 23 mm in rapid to point Z-164.
11. The Z axis moves in rapid to the reference plane (Z- 98).
12. The X and Y axes move 500 mm at the rapid feedrate (F100) to point X550, Y550.
13. Operations 4 to 10 are repeated.
14. The Z axis withdraws in rapid to the initial plane (Z0).
15. The X and Y axes move in rapid to point X0,Y0.
16. End of program (the spindle stops running).



+Y
550
50
+X
50
550
Starting plane
Z=0
+Z
98mm
2mm
Reference plane
22mm
22mm
22mm
MPM073

The deep hole drilling canned cycle G83 can also be programmed with the following format:

b) **N4 G83 G98/G99 (V+/-4.3) (W+/-4.3) X+/-4.3 Y+/-4.3 Z+/-4.3 I+/-4.3 B4.3 C4.3 D+/-4.3 H4.3 J2 K2.2 L4.3 R(0.000/500) N2.**

The different parameters have the following meanings:

N4 : Block number (0/9999).
G83 : Code of the deep drilling canned cycle.
G98 : Withdrawal of the axis perpendicular to the main plane, to the starting plane, after completing the machining.
G99 : Withdrawal of the axis perpendicular to the main plane, to the reference plane, after completing the machining.
X+/-4.3 :These values have different meaning depending on the main plane in which
Y+/-4.3 :we are operating.
Z+/-4.3 :
*(W+/-4.3):
*(V+/-4.3):

MAIN PLANE	VALUE	MEANING
X/Y G17	X+/-4.3 Y+/-4.3	They define the movement of the axes of the main planenecessary to position the tool in the center of the first machining. The values will be either absolute or incremental depending on the function on which we are operating (G90 or G91). The movements will be carried out either in rapid or in F operation feedrate, depending on the function on which we are operating (G00 or G01). That point can be also programmed in polar coordinates.
X/Z G18	X+/-4.3 Z+/-4.3	
Y/Z G19	Y+/-4.3 Z+/-4.3	
X/Y G17	Z+/-4.3	It identifies the movement of the axis perpendicular to the main plane, from the starting plane up to the reference plane (approach). This movement will be executed in rapid (G00).The values will be either absolute or incremental, depending on the function on which we are operating (G90 or G91). This value must necessarily be programmed.
X/Z G18	Y+/-4.3	
Y/Z G19	X+/-4.3	

* If the 4th W axis or the 5th V axis are perpendicular to the main plane, it must be a linear axis. But if it is one of the axis of the main plane, it may also be a rotary axis.

I+/-4.3 : Identifies the full machining depth. If operating on G90, the values are absolute, in other words, they are related to the datum point of the axis perpendicular to the main plane. If operating on G91, the values are incremental, that means, they are related to the reference plane.

B4.3 : Incremental penetration. If identifies the value of each step of machining related to the axis perpendicular to the main plane. Only positive numbers are allowed.

C4.3 : It identifies how close to the previous penetration must the move in rapid G00 be performed for the next penetration. If this parameter either is not programmed, or is programmed with value zero, the CNC will consider it as value 1 mm. If a zero value is programmed, the CNC will issue error 44.

D+/-4.3:Identifies the distance between the referenceplane and the part's surface, in other words, it is the value which is added or deducted, depending on the sign, to the incremental penetration B of the first penetration.

H4.3 : Distance that the axis perpendicular to the main plane withdraws after each penetration. If this parameter either is not programmed or is set to 0, the axis perpendicular to the main plane withdraws to the reference plane after each penetration. If zero is programmed, the CNC will issue error 44.

J2 : Value which identifies how often (in number of penetrations) the tool withdraws to the reference plane in G00. It is possible to program either a value within 00 and 99 or, if it is programmed with a parameter (J P2), the latter can have a value within 00 and 225. If this parameter either is not programmed or is set to 0, the CNC will consider it as value 1, in other words, it will withdraw to the reference plane after each penetration.

K2.2 : Dwell in seconds after every penetration. It is possible either to program a time within 0.00 and 99.99 sec. or, if it is programmed with a parameter (K P3), within 0.00 and 655.35 sec.

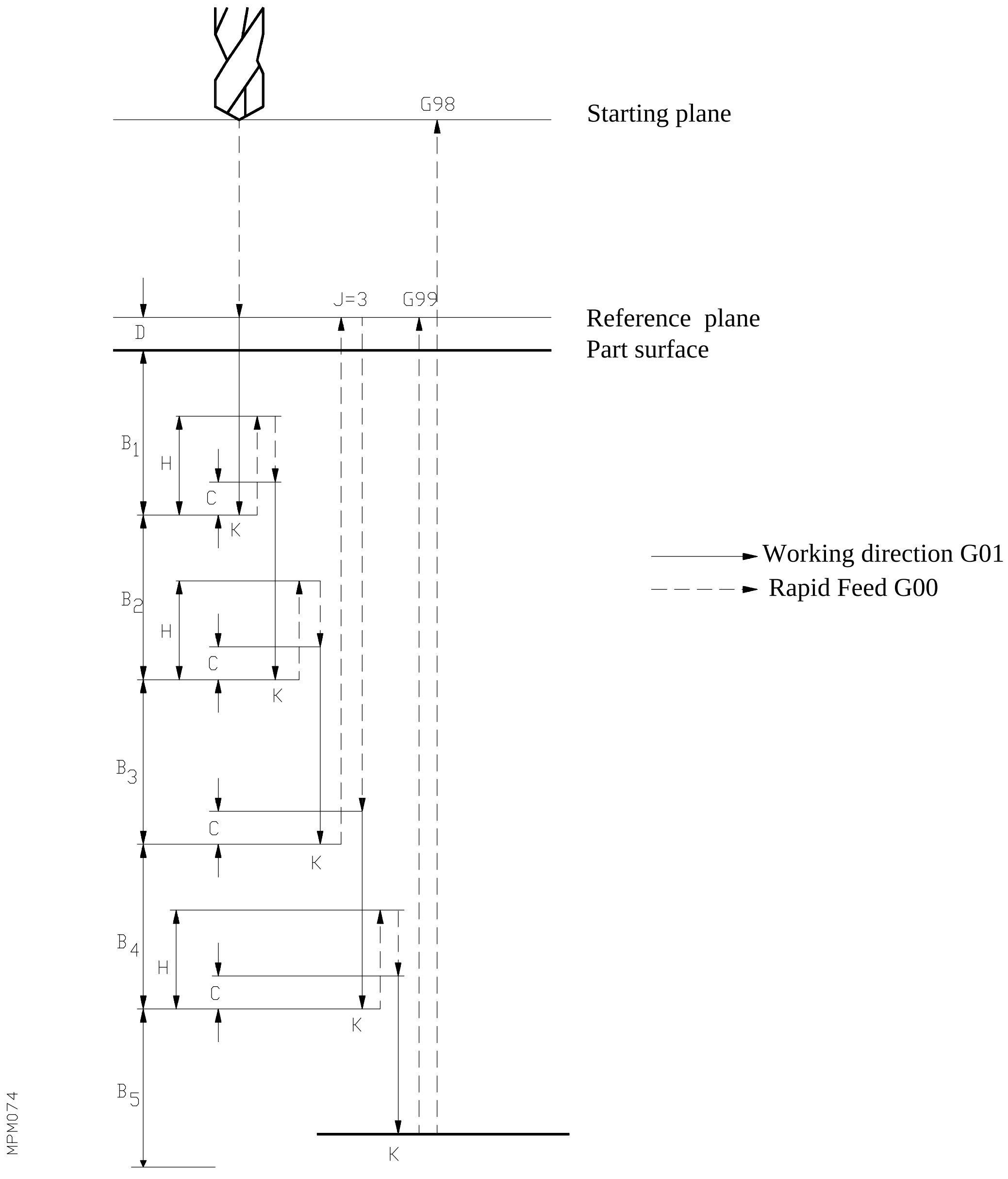
L4.3 : Identifies the minimum value of the incremental penetration. If this parameter either is not programmed or is set to 0, the CNC will consider it as value 1 mm.

R(0.000/500): Factor which decreases or increases the value of incremental penetration B. If R=1, all the penetrations B are equal. If R is different from 1, the first penetration will be B=B, the second B=RB, the third B=R(RB) and so on. If this parameter either is not programmed or is set to 0, the CNC will consider it as value 1.

N2 : Identifies the number of times the block execution is required to be repeated. A value within N0 and N99 can be programmed, although, if it is programmed with a parameter (N P2), the latter can have a value within 0 and 255. If the parameter N is not programmed, CNC assumes the value N1.

Obviously, the programming of values of Nhigher than 1 makes sense if operating on G91, in other words, the values of movement of the axes are incremental, since otherwise, the machinings will be repeated at the same point. When programming the same canned cycle a number of times, only the functions F,S and M will be executed in the cycle calling block.

Movements of the axis perpendicular to the main plane, on the deep drilling cycle G83, programmed in format b).



Sequences and explanation of operation:

1. If the spindle was previously running, it keeps on rotating in the same direction. If it was not running, it start clockwise (M03).
2. Movement from the starting plane to the reference plane in rapid G00.
3. Movement at the working feedrate of a distance equal to B+D.
4. Dwell K in seconds, if it has been programmed.
5. Withdraws in G00 either a distance equal to H or to the reference plane according to the value given to J.
6. Movement in rapid to a distance C, before the previous penetration.
7. Movement at the working feedrate of a distance equal to B+C.
8. Dwell K in seconds if it has been programmed.
9. Operations 5 to 8 are repeated, until reaching the penetration I.
10. Depending on the function programmed G98 or G99, the tool withdraws either to the starting plane or the reference plane in rapid.

<u>Atention:</u>

If the value given to parameter R is equal to 1, all the incremental penetrations B are equal (B1=B2=B3=B4).

If the mentioned parameter is different from 1, the different penetrations will be: B1=B; B2=RB1; B3=RB2; B4=RB3.

In both cases, the last penetration will be determined by the CNC, according to the value of the total penetration I.

If we program for instance, B=12 L=9 R=0.9; the incremental penetrations B will be:

B1=12
B2=0.9X12=10.8
B3=0.9X10.8=9.72
B4=0.9X9.72=8.748

As **B4** is smaller than the minimum penetration **L**, from B4 on, it included, every subsequent penetration will have a value equal to **L**, that means equal to 9.

6.32.7. Pocket milling canned cycle definition (G87,G88)

When operation on cartesian coordinates, the basic structure of the block in which a cycle is defined is:

N4 (G87 or G88) (G98 or G99) (W+/-4.3) (V+/-4.3) X+/- 4.3
Y+/-4.3 Z+/-4.3 I+/-4.3 J+/-4.3 K4.3 (for G87 only)
B4.3 C4.3 D+/-4.3 H4 L4.3 N2

N4 : Block number (0-9999).

G(87 or 88) : Code of the canned cycle selected.

G98 : Withdrawal of the axis perpendicular to the main plane to the starting plane after completing the machining of the pocket.

G99 : Withdrawal of the axis perpendicular to the main plane to the reference (approach) plane after completing the machining of the pocket.

X+/-4.3 :These values have different meaning depending on the plane in whic
Y+/-4.3 we are operating (main plane).
Y+/-4.3
Z+/-4.3
*(W+/-4.3):
*(V+/-4.3) :

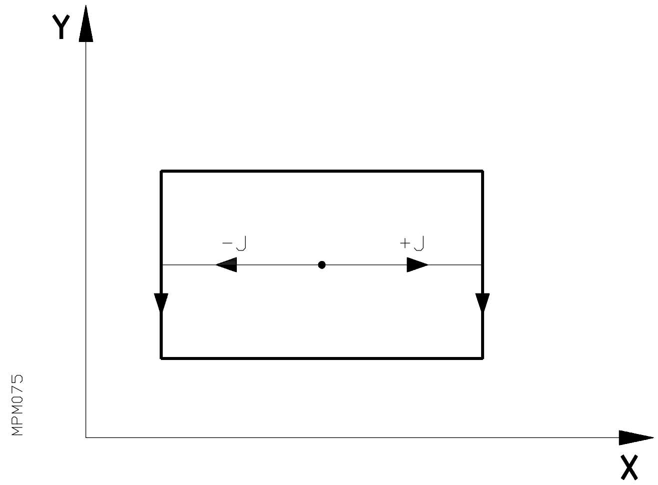
<table>
<tr><th>MAIN PLANE</th><th>VALUE</th><th>MEANING</th></tr>
<tr><td>X/Y
G17</td><td>X+/-4.3
Y+/-4.3</td><td rowspan="3">They define the movement of the axes of the main plane necessary to position the tool in the center of the first machining. The values will be either absolute or incremental depending on the function on which we are operating (G90 or G91).
The movements will be carried out either in rapid or in F operation feedrate, depending on the function on which we are operating (G00 or G01).
That point can be also programmed in polar coordinates.</td></tr>
<tr><td>X/Z
G18</td><td>X+/-4.3
Z+/-4.3</td></tr>
<tr><td>Y/Z
G19</td><td>Y+/-4.3
Z+/-4.3</td></tr>
<tr><td>X/Y
G17</td><td>Z+/-4.3</td><td rowspan="3">It identifies the movement of the axis perpendicular to the main plane, from the starting plane upto the reference plane (approach). This movement will be executed in rapid (G00).The values will be either absolute or incremental, depending on the function on which we are operating (G90 or G91). This value must necessarily be programmed.</td></tr>
<tr><td>X/Z
G18</td><td>Y+/-4.3</td></tr>
<tr><td>Y/Z
G19</td><td>X+/-4.3</td></tr>
</table>

* When machining a pocket, if the 4th W axis or the 5th V axis are perpendicular to the main plane, it must be a linear axis. But if it is one of the axis of the main plane, it may also be a rotary axis.

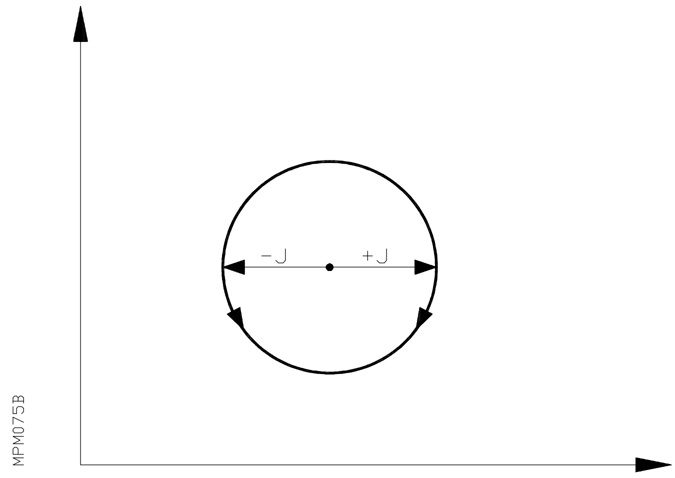
I+/-4.3: Defines the machining depth. When operating on G90, the values are absolute; i.e. they are referred to the origin of the Z axis. When operating on G91, the values are incremental; i.e. they are referred to the reference (approach) plane.

J+/-4.3: In the case of G87 (rectangular pocket), this defines the distance from the center to the edge along the relevant axis.

. Along the X axis in the XY plane (G17)
. Along the X axis in the XZ plane (G18)
. Along the Y axis in the YZ plane (G19)

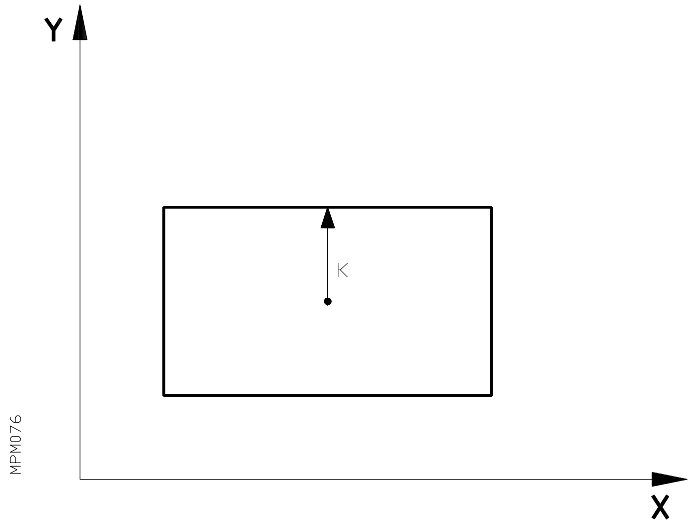


In the case of G88 (circular pocket), it defines the radius of the pocket. The direction of machining depends on whether it isgiven a positive or negative sign.

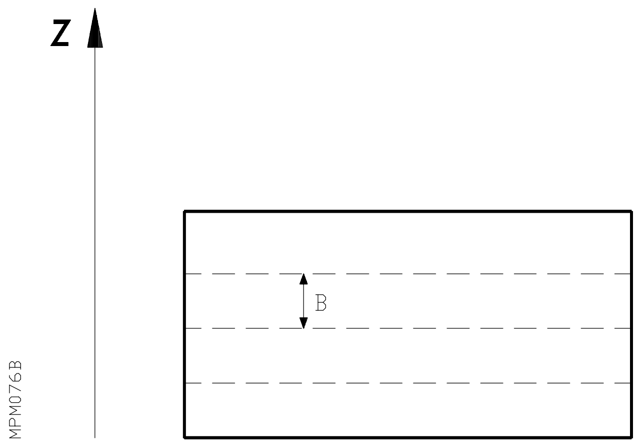


K4.3: Is only used in the case of caned cycle G87 and defines the distance from the center to the edge along the relevant axis. Only positive values may be programmed.

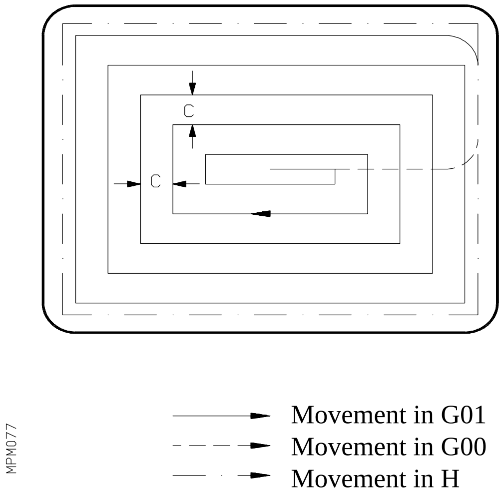
. Along the Y axis in the XY plane (G17)
. Along the Z axis in the XZ plane (G18)
. Along the Z axis in the YZ plane (G19)



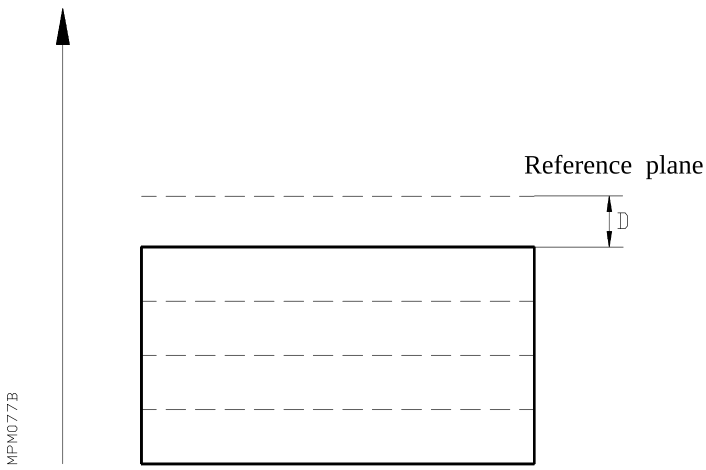
B4.3: Defines the value of each machining step along the Z axis. Only positive values are allowed.



C4.3: Defines the value of each machining step in the plane. Only positive values are allowed. If this parameter is not entered, the CNC assumes that the value of the step is 3/4 D of the active tool. If C=0 is programmed, the CNC will indicate error 44.



D+/-4.3: Defines the distance between the reference (approach) plane and the surface of the part.

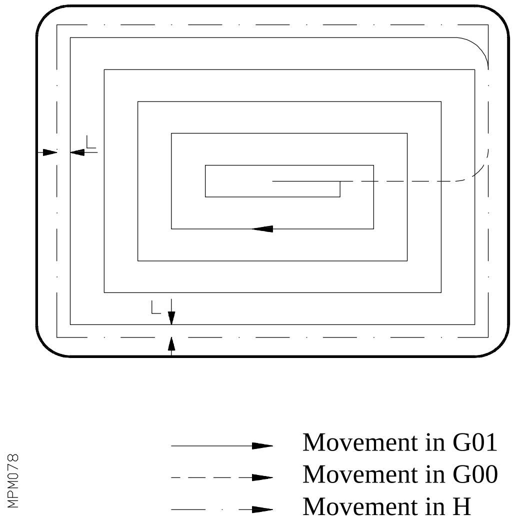


Reference plane D is used to make the axis perpendicular to the main plane travel in rapid to the reference plane and then continue at the machining feedrate for a distance equal to D+B. The other steps of the Z axis will be equal in value to B. If a negative value is given to D, the first penetration will be smaller than B, i.e. will be equal to (-D+B).

H4: Defines the feedrate in the final machining (finishing) pass.

L4.3: Defines the value of the finishing pass, referred to the main plane. Only positive values are allowed.

. If the sign is positive the finish run will be made in G7 (square edge).
. If the sign is negative, the finish run will be made in G5 (rounded edge).



Atention:

The CNC moves the machine in successive passes, according to B and C programmed values, except in the finishing pass when the values are set according to the pocket dimensions.

N2: Defines the number of times the execution of a block defined in the cycle is to be repeated. Any value between N1 and N99 may be programmed. Unless it is programmed using a parameter (N P3) in which case the limits are 0 and 255. If the N parameter is not programmed, the CNC assumes the value N1. The programming of values of N greater than 1 obviously makes sense when operating on G91; i.e. when the values of the pocket's center are incremental, since otherwise the machining operations will be repeated at the same point.

A more detailed explanation of G87 and G88 canned cycles is given next, supposing that the main plane is the one formed by X and Y axes and the tool's axis is Z.

6.32.8. G87. Rectangular pocket milling canned cycle

The operations and movements of the tool are as follows:

- If the spindle was previously running, it continues to rotate in the same direction. If it was not running it starts clockwise (M03).

- Rapid movement of the **Z** axis from the starting plane to the reference (approach) plane.

- Movement a **50%** of the working feedrate (F) of the **Z** axis for a distance equal to (D+B).

  D: Distance between the reference plane and the surface of the part.
  B: Depth value of each machining pass.

- Milling at working feedrate (F) of the pocket surface by steps defined by C, to a distance **L** (finishing pass), of the pocket wall.

- Milling at the working feedrate **H**, of the finishing pass.

- After completing the finishing pass, the tool withdraws in rapid move to the pocket center, positioning the **Z** axis 1 mm higher. In this way, the first penetration is finished.

- Movement at **50%** of the working feedrate (F) of the **Z** axis for a distance equal to **B+1**.

- Milling at working feedrate (F) of the pocket surface (second penetration).

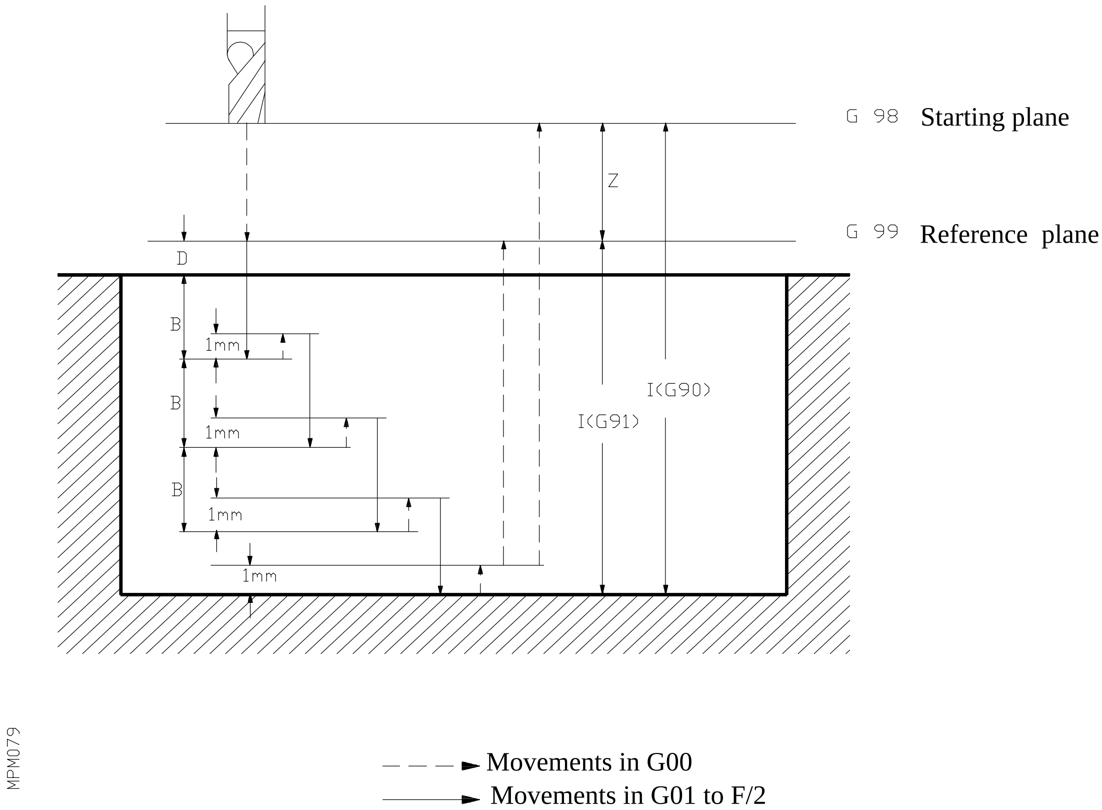
- The above steps are repeated until the full depth of the pocket is reached.

- Once the pocket is completed, the tool withdraws in rapid (Z axis) to the reference plane (if G99 has been programmed) or to the starting plane (if G98 has been programmed).

Atention:

To enable a good finish to be achieved in the machining of pockets the CNC performs a tangential approach and tangential exit in the last pass of each of thepenetrations. For this purpose, the tool has to withdraw to the center of the pocket before beginning the milling of the wall. To avoid problems and possible machining malfunctions, it is mandatory to program the tool code (T2.2) and to enter in the tool table the value of the radius of the tool that is to be used. If the radius value entered in the tool table is **RO**, the final wall pass is performed like all the others; i.e. with neither tangential entry nor tangential exit. **The value or R must never be negative**. If T2.2 is not programmed, the CNC takes as tool radius the value of **R** of the last corrector.

Movement of the axis perpendicular to the main plane in **G87** canned cycle (e.g. Z axis).



Example:

Machining of a rectangular pocket of 105x75 mm surface and 40 mm depth.

Let us suppose that:

. The distance between the reference plane and the surface of the part is 2 mm.

. The starting point is X0,Y0,Z0 and the spindle is not running.

. The tool has 7,5 mm radius and it is number 1 (T1.1).

```
N0  G87 G98 G00 G90 X90 Y60 Z-48 I-90 J52.5 K37.5 B12 C10 D2 H100 L5 F300 S1000
T1.1 M03
N5   G80 X0   Y0
N10  M30
```

First block N0

G87 : Defines the rectangular pocket canned cycle.

G98 : Defines the withdrawal of the tool (Z axis) to the starting plane, after completing the machining of the pocket.

G00 : Defines the movement of the axes XY as being rapid.

G90 : Defines the X,Y,Z,I dimensions as absolute coordinate values.

X,Y : Movement of those axes to the center of the pocket.

Z : Movement of the tool (Z axis) from the starting plane to the reference plane (always in rapid).

I : Movement to the bottom of the pocket (absolute coordinate referred to Z0).

J : Defines the value of 1/2 the pocket's length; i.e. the distance between the center and the wall, following X axis. The direction of the milling will depend on the sign (positive or negative) programmed.

K : Defines the value of 1/2 the pocket's width; i.e. the distance between the center and the wall, following Y axis (always positive).

B : Defines the penetration of each milling step (always positive).

C : Defines the value of each machining step in the XY plane. Only positive values are allowed. If C either is not programmed, or is set to 0 , the CNC assumes that the value of the step is 3/4 D of the active tool.

D : Defines the distance between the reference (approach) plane and the surface of the part. The depth of the first machining step is : (D+B).

H : Defines the feedrate in the final machining (finishing pass).

L : Value of the final machining (finishing pass).

F : Defines the machining feedrate.

S : Revolutions per minute of the spindle.

T : Tool number.

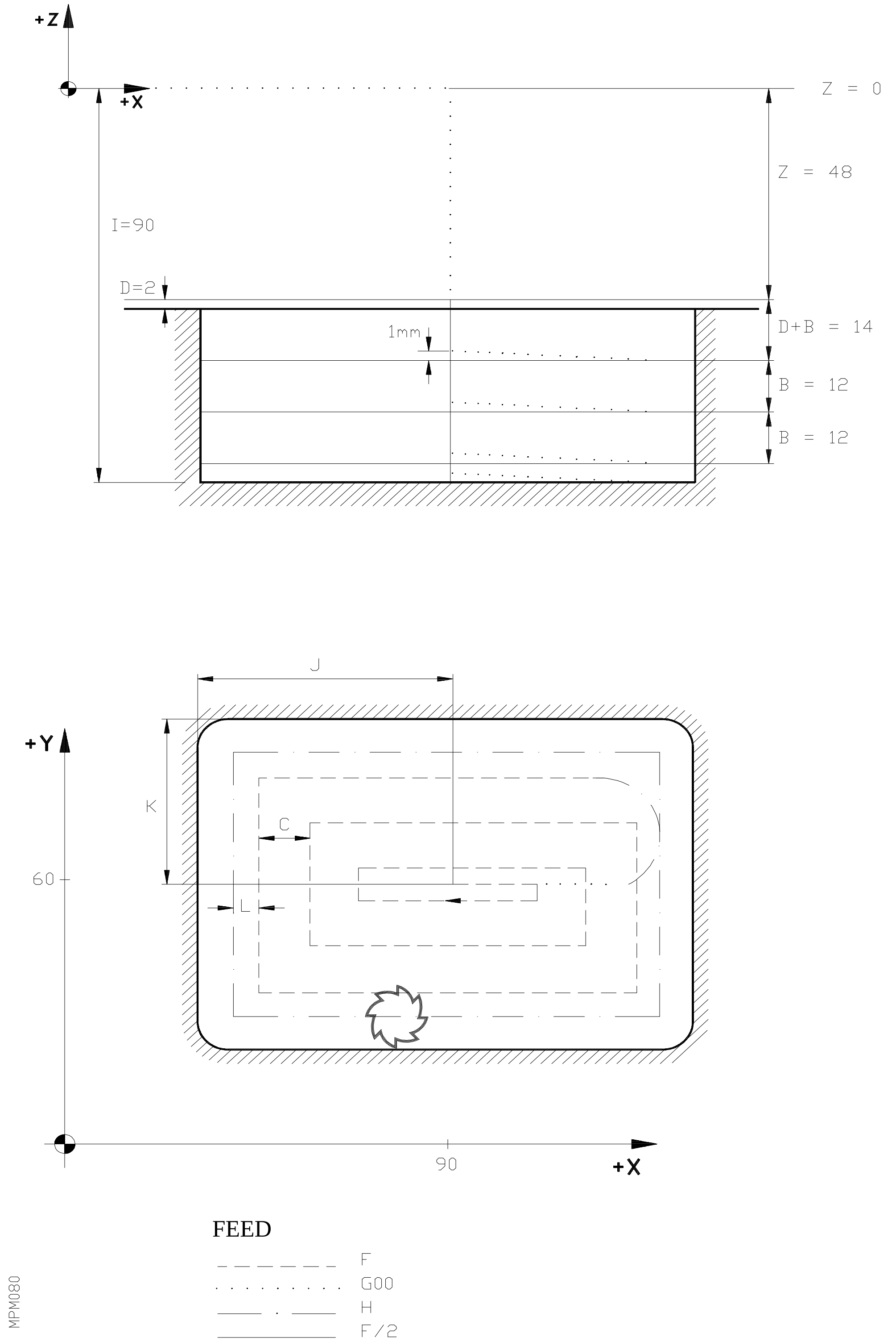
M03 : Clockwise spindle rotation.

Second block (N5)

G80 X0 Y0 : Cancellation of canned cycle and return in rapid feedrate to the starting point.

Third block (N10)

M30 : End of program.


+Z
+X
Z = 0
Z = 48
I=90
D=2
1mm
D+B = 14
B = 12
B = 12
J
+Y
K
C
60
L
90
+X
FEED
F
G00
H
F/2
MPM080

Sequence and explanation of operations

1) The X and Y axes move in rapid from point X0,Y0,Z0 to point X90 Y60 Z0.

2) The spindle will start running clockwise at 1000 rev/min.

3) The Z axis will move in rapid 48 mm to the reference plane Z-48.

4) The Z axis moves a further 14 mm (D+B) at F/2 (half the programmed F value) to Z-62.

5) The X and Y axes move until completing the pocket's final dimensions, as it is shown in the drawing, at the working feedrate F, except in the last pass (machining of the pocket wall), which is carried out at the finishing feedrate H, and with a tangential entry and tangential exit. The mentioned for the finishing pass, is always carried out even when the finishing pass L has not been programmed.

6) The tool will withdraw in rapid feedrate to the pocket center and position the Z axis at a point 1 mm higher (X90 Y60 Z-61).

7) The Z axis will move 13 mm (B+1), at F/2 (half the working feedrate F) to Z-74.

8) Operations 5 and 6 are repeated.

9) The Z axis will move 13 mm at F/2 feedrate, to Z-86.

10) Operations 5 and 6 are repeated.

11) The Z axis will move 5 mm at F/2 feedrate to Z-90.

12) Operations 5 and 6 are repeated.

13) The Z axis will withdraw in rapid feedrate 89 mm to Z0.

14) The X and Y axes will withdraw in rapid feedrate to X0 Y0.

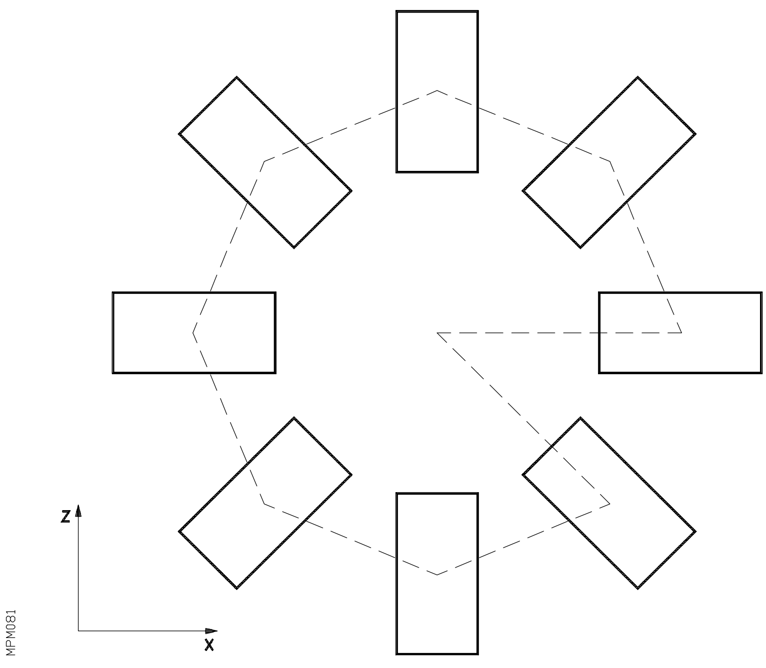
15) End of program.

The possibility of performing pockets whose sides are not parallel to the coordinate axes, by applying the function G73 (Coordinates system rotation) must be emphasized.

This service enables a rapid pocket programming in any point of any plane.

Example: The initial point is X0,Y0,Z0 and the pocket is performed in (X Z) plane.

```
N5  G18
N10 G87 G98 G00 G90 X200 Y-48 Z0 I-90 J52.5 K37.5 B12 C10 D2 H100 L5 F300
N20 G73 A45 N30 G25 N10.20.7
N40 M30
```



6.32.9. G88. Circular pocket milling canned cycle

The operations and movements of the tool are as follows:

- If the spindle was previously running, it continues to rotate in the same direction. If it was not running, it starts clockwise (M03).

- Rapid movement of the **Z** axis from the starting plane to the reference (approach) plane. As mentioned earlier,reference plane means a plane which must be situated close to the surface of the part to be machined.

- Movement at 50% of the working feedrate (F) of the **Z** axis for a distance equal to (D+B).

  D: Distance between the reference plane and the surface of the part.

  B: Deep value of each machining pass.

- Milling at working feedrate (F) of the surface of the pocket by steps defined by **C**, to a distance **L** (finishing pass), of the pocket wall.

- Milling at the working feedrate **H**, of the final machining finishing pass.

- After completing the finishing pass, the tool withdraws in **G00** to the pocket center, positioning the **Z** axis at a point 1 mm higher, in this way the first penetration finishes.

- Movement at 50% of the working feedrate (F) of the **Z** axis for a distance equal to **B+1.**

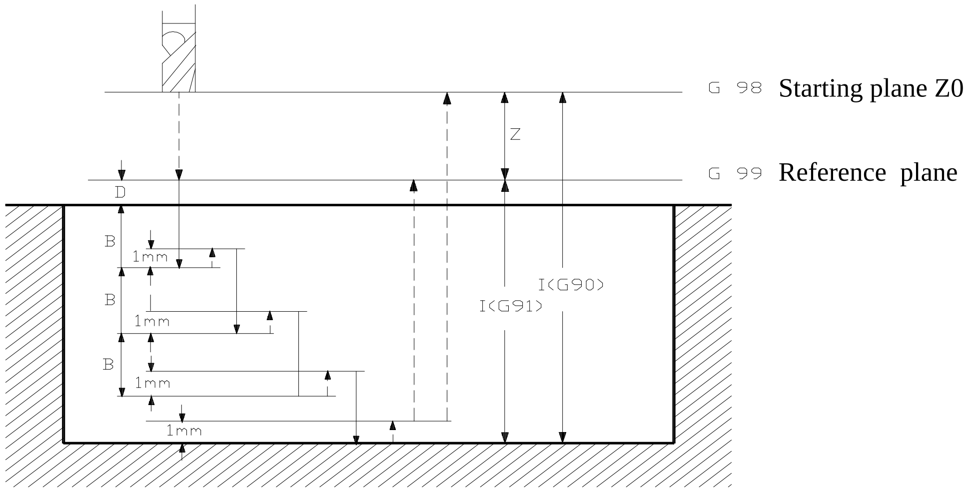
- Milling at working feedrate (F) of the surface of the pocket (second penetration).

- The above steps are repeated until the full depth of the pocket is reached.

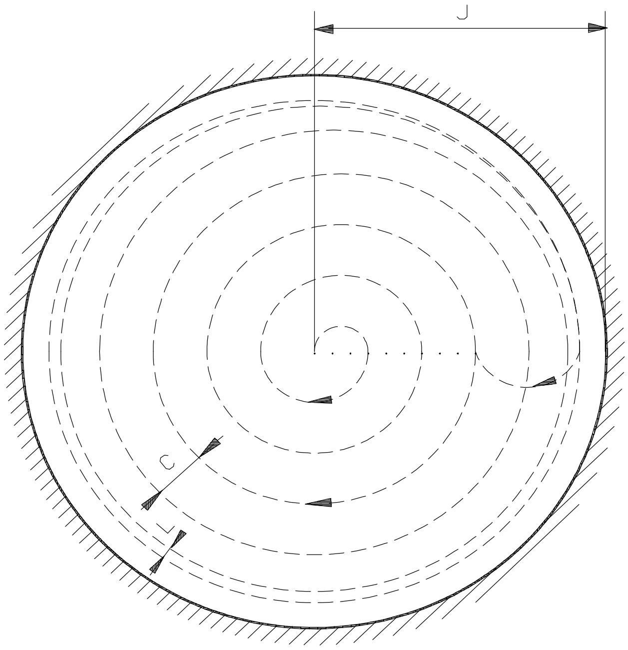
- Once the pocket is completed, the tool withdraws in rapid (Z axis) to the reference plane (if G99 has been programmed) or to the starting plane (if G98 has been programmed).

Atention:

To enable a good finish to be achieved in the machining of pockets the CNC performs a tangential approach and tangential exit in the last pass of each of the penetrations. For this purpose, the tool has to withdraw to the center of the pocket before beginning the milling of the wall. To avoid problems and possible machining malfunctions, it is mandatory to program the tool code (T2.2) and to enter in the tool table the value of the radius of the tool that is to be used. If the radius value entered in the tool table is **R0**, the final wall pass is performed like all the others; i.e. with neithertangential entry nor tangential exit. The value or **R must never be negative**. If T2.2 is not programmed, the CNC takes as tool radius the value of **R** of the last corrector used.



Movements in G00
Movements in G01 to F/2



Movement from the center of the tool in G00
Movement from the center of the tool in G01
Wall of pocket

MPM082

Example:

Machining of a circular pocket of radius 70 mm and depth 40 mm.

Let us suppose that:

. The distance between the reference plane and the part's surface is 2 mm.

. The tool starting point is X0 Y0 Z0 and the spindle is not running.

. The tool has 7.5 mm radius and it is number 1 (T.1).

```
N0  G88 G98  G00 G90 X90 Y80 Z-48 I-90 J70 B12 C10 D2 H100 L5  F300 S1000 T.1
M3
N5  G80 X0   Y0
N10 M30
```

Block N0

G88 : Identifies the circular pocket cycle.

G98 : Identifies the withdrawal of the tool (Z axis) to the starting plane after completing the pocket machining.

G00 : Identifies the X and Y axes movement as being in rapid move.

G90 : Identifies the X,Y,Z and I dimensions as being absolute coordinate values.

X,Y : Movement of the mentioned axes to the center of the pocket.

Z : Movement of the tool (Z axis), from the starting plane to the reference one (always in rapid move).

I : Movement to the pocket bottom (Absolute coordinate value referred to Z0).

J : Identifies the pocket radius. The milling direction will depend on the sign (positive or negative) programmed.

B : Depth of each milling pass (always positive).

C : Identifies the value of each pass in the plane (X,Y) always positive. If the value of C is either not programm or set to 0, the CNC takes 3/4 D of the tool.

D : Distance between the reference plane and the part's surface. The depth of the first machining step is D+B.

H : Identifies the feedrate speed in the final machining (finishing pass).

L : Value of the final machining.

S : Revolutions per minute of the S spindle rotation.

T : Tool number (code).

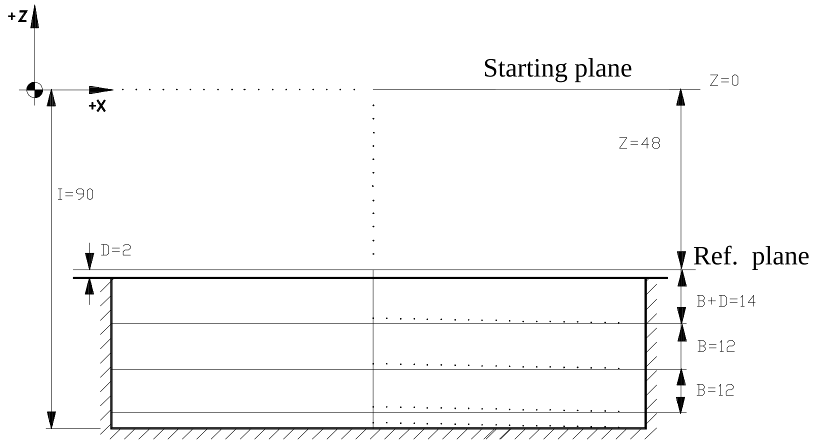
M03 : Clockwise spindle rotation.

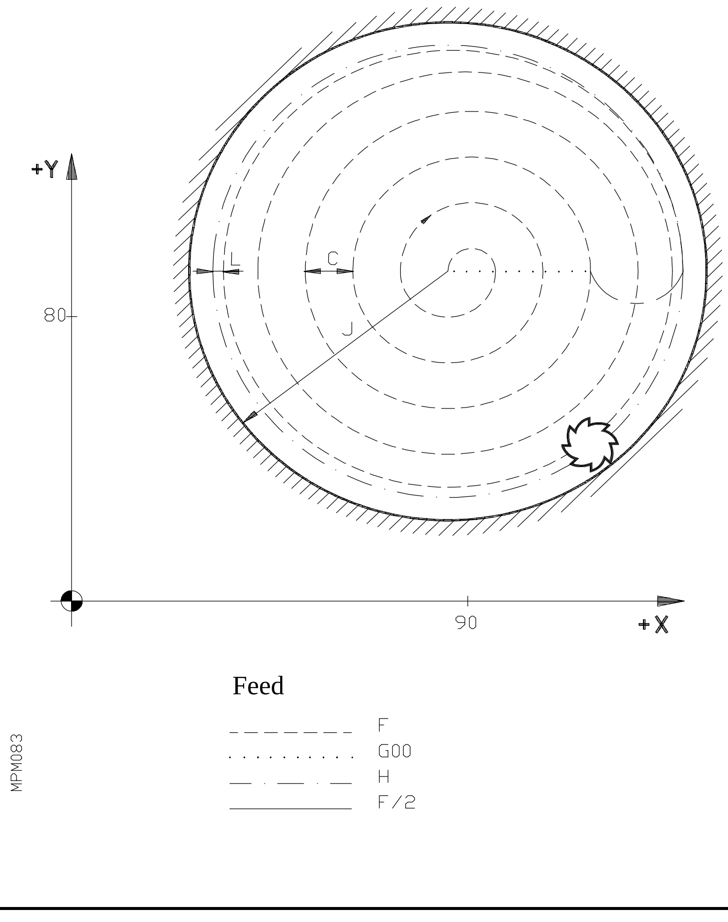
Block N5

G80 X0 Y0 : Cancellation of the canned cycle and return in the rapid move to the starting point.

Block N10

M30 : End of program.


+Z
+X
Starting plane
Z=0
Z=48
I=90
D=2
Ref. plane
B+D=14
B=12
B=12


+Y
80
L
C
J
T
90
+X
Feed
F
G00
H
F/2
MPM083

Sequence and explanation of operations

1) The **X** and **Y** axes will move in rapid from point X0 Y0 Z0 to point X90 Y80 Z0.

2) The spindle will start clockwise at 1000 rpm.

3) The **Z** axis will move 48 mm in rapid to the reference plane (Z-48).

4) The **Z** axis will move a further 14 mm (D+B) at **F/2** (half the working feedrate F) to Z-62.

5) The **X** and **Y** axes will move until completing the pocket final dimensions, as it is shown in the drawing, at the feedrate **F**, except in the final pass (machining of the pocket wall), that will be performed at the feedrate **H** and with a tangential exit. The mentioned for the final pass is always performed, even if the finishing pass **L** has not been programmed.

6) The tool will withdraws in rapid feedrate to the pocket center and position the **Z** axis at a point 1 mm higher (X90 Y60 Z-61).

7) The **Z** axis will move 13 mm (B+1) at feedrate F/2, up to Z-74.

8) Operations **5** and **6** are repeated.

9) The **Z** axis will move 13 mm at feedrate F/2 up to Z- 86.

10) Operations **5** and **6** are repeated.

11) The **Z** axis will move 5 mm at feedrate F/2 up to Z- 90.

12) Operations **5** and **6** are repeated.

13) The **Z** axis will withdraws 89 mm in rapid, up to Z0.

14) The **X** and **Y** axes will withdraw in rapid up to X0 Y0.

15) End of program.

6.33. G90 G91. ABSOLUTE AND INCREMENTAL PROGRAMMING

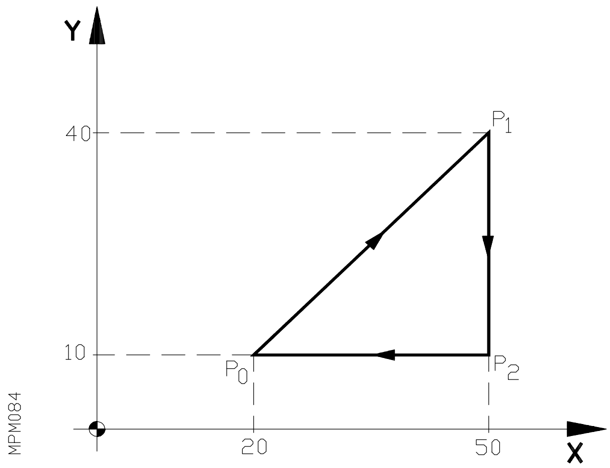
The programming of the coordinates of a point, may be carried out, either in absolute coordinates **G90** or in incremental coordinates **G91**.

When operating on **G90**, the coordinates of a point programmed, are referred to the point of the coordinate origin.

When operating on **G91**, the coordinates of the point programmed, are referred to the path's previous point; i.e. the programmed values identify the distance to go along the relevant axis.

When turning on and after executing **M02**,**M30**, **EMERGENCY** or **RESET**, the CNC assumes the function **G90**.

The functions **G90** and **G91** are incompatible with each other when being in the same block.



Let us suppose that the starting point is P0(20,10).

Absolute programming G90

N20 G90 X50 Y40   P0 —> P1
N30 Y10   P1 —> P2
N40 X20   P2 —> P0

Incremental programming G91

N20 G91 X30 Y30 P0 —> P1
N30 Y-30   P1 —> P2
N40 X-30   P2 —> P0

6.34. G92. COORDINATE PRESET

Function **G92** can be used to preset any value on the axes of the CNC, which involves being able to shift the coordinate origin.

Block format: N4 G92 V+/-4.3 W+/-4.3 X+/-4.3 Y+/-4.3 Z+/-4.3.

When function **G92** is programmed, there is no movement of the axes, and the CNC accepts the values of the axes programmed after **G92** as the new coordinate values of those axes.

Example:

Let us suppose that the tool is at the coordinate origin (X0,Y0).

The program for describing the path drawn will be:

N10 G00 G90 X100 Y100
N20 X400

If we use **G92**, the following will occur:

N10 G92 X500 Y500

The coordinate origin (X0,Y0) is now point X500,Y500.

N20 G00 G90 X600 Y600
N30 X900

No other function can be programmed in the block in which **G92** is programmed.

Coordinate values preselection by **G92** is referred always to the theoretical position in which the axes are.

6.35. G93. PRESELECTION OF POLAR ORIGIN

Function **G93** can be used to preselect any point in a plane (XY,XZ,YZ) as the origin of polar coordinates.

There are two ways of preselecting an origin of polar coordinates:

a) G93 I+/-4.3 J+/-4.3 (always absolute coordinate values).

or, G93 I+/-3.4 J+/-3.4

I+/-4.3: I+/-3.4: Indicates the value of the abscissa of the polar coordinate origin; i.e. the value of X in the XY plane, the value of X in the XZ plane and the Y in the YZ plane.

J+/-4.3: J+/-3.4: Indicates the value of the ordinate of the polar coordinate origin; i.e. the value of Y in the XY plane, the value of Z in the XZ plane and the value of Z in the YZ plane.

In **four-axis** machines, if the **fourth axis (W)** is linear and is a part of the main plane, the values of **I**,**J** will define the value either of the fourth axis or of its associated one.

This will also happen with the **5th axisV** in five axis machines. No more information can be programmed in this block.

b) The programming of **G93** in a block determines that, prior to the movement programmed, the actual position of the tool becomes the polar origin.

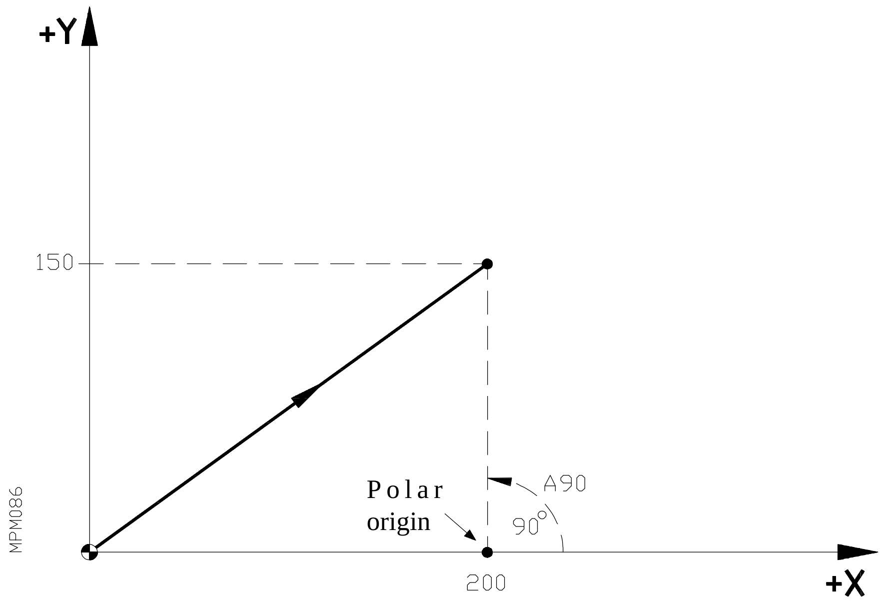
Atention:

When a circular (helical) interpolation is programmed with **G02** or **G03**, the CNC takes the arc's center as the new polar origin.

Examples:

1) Let us suppose that the tool is situated at the cartesian coordinate origin.

N0 G93 I200 J0
N5 G01 R150 A90 F500

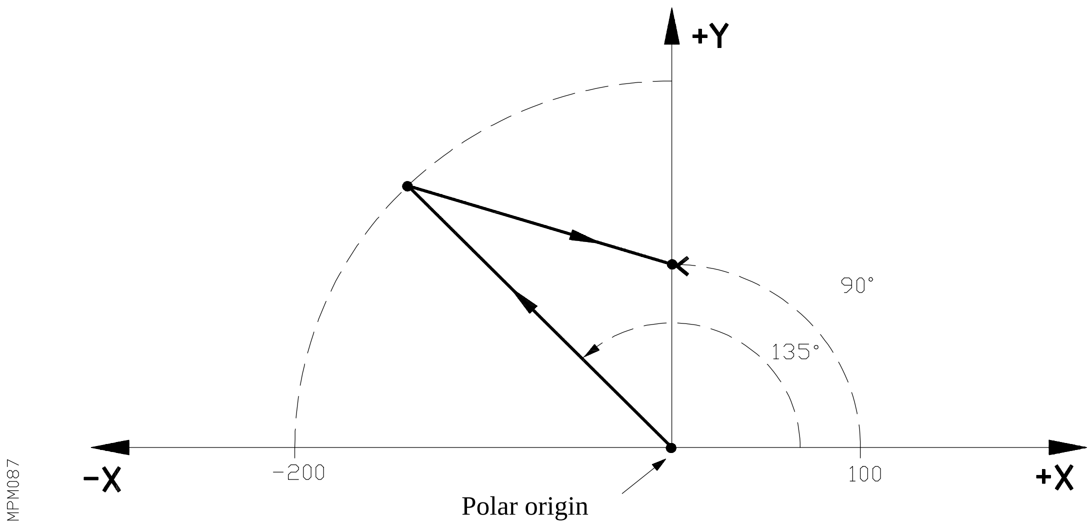


In block **N0**, the point X200 Y0 has been defined as polar origin.

In block **N5** a linear interpolation (G01) up to point R150 A90 (X200 Y150) has been programmed.

2) Let us again suppose that the tool is at X0 Y0.

N0 G93 G01 R200 A135 F500
N5 R100 A90



On reading block **N0**, the CNC takes the point where the tool is located at that moment (X0,Y0) as the polar origin in order to continue by executing a linear interpolation movement (G01) to the point defined by R200 A135.

N5 then defines another linear interpolation movement to R100 A90.

Atention:

When turned on or after **M02**,**M30**, **EMERGENCY** or **RESET**, the CNC takes the point (X0,Y0) as the polar origin.

When changing of main plane, it takes the cartesian coordinate origin of that plane as the polar origin.

When changing to G18 it takes X0 Z0.
When changing to G19 it takes Y0 Z0.
When changing to G17 it takes X0 Y0.

6.36. G94. FEEDRATE F IN mm/min. (inches/min.)

When the code **G94** is programmed the CNC assumes that the values entered by **F** are in mm/min.(0.1 inches/min) or 0.1mm/min (0.01 inch/min) depending on the value of machine parameter P611(5).

G94 is modal, i.e. it remains active until **G95**, is programmed when turning on or after **M02**,**M30**, **EMERGENCY** or **RESET** the CNC assumes **G94**.

6.37. G95. FEEDRATE F IN mm/rev. (inches/rev.)

When the code **G95** is programmed the CNC assumes that the values entered by **F3.4** are in mm/rev. the maximum value in mm is F500 (500mm/rev). In inches the format is **F2.4** (F1=1inch/rev) and the maximum value is 19.6850 inch/rev. **G95** is modal, i.e. it remains active until **G94**,**M02** or **M30** are programmed. This feature requires an encoder on the spindle.

Atention:

The meaning of **F** (feed programming) differs, according to whether we are working in **G94** or **G95**, from the value of machine parameter P611 (5) when we are working in **G94** and from the system used in programming either in mm or inches. All this will be dealt with later in the section **FEED PROGRAMMING.**

6.38. G96. CONSTANT SURFACE SPEED

When **G96** is programmed, the CNC assumes that values **F** refer to the feed at the tool's cutting edge. The feed at the center of the tool will vary when machining around corners so that the feed at the cutting edge remains constant.

This feature enables a better finishing of the part especially on inside corners. **G96** is modal and is cancelled by **G97**,**M02** or **M30**.

When operating in **G96** the tool center's speed will varyaround corners, so that the cutting edge's speed remains constant.

6.39. G97. CONSTANT TOOL CENTER SPEED

When **G97** is programmed the values **F4** and **F3.4** are assumed as being the feed at the tool's center. The feed at the cutting edge will vary when machining around corners so that the feed at the center of the tool remains constant.

Function **G97** is modal and incompatible with **G96**. The CNC assumes **G97** when turning ON and after **M02**,**M30**, **EMERGENCY** or **RESET**.

7. COORDINATE PROGRAMMING

A point can be programmed in the CNC by using:

. Cartesian coordinates
. Polar coordinates
. Cylindrical coordinates
. Two angles
. One angle and one cartesian value

7.1. CARTESIAN COORDINATES

7.1.1. Axis coordinates

The format of the axis coordinate values is as follows:

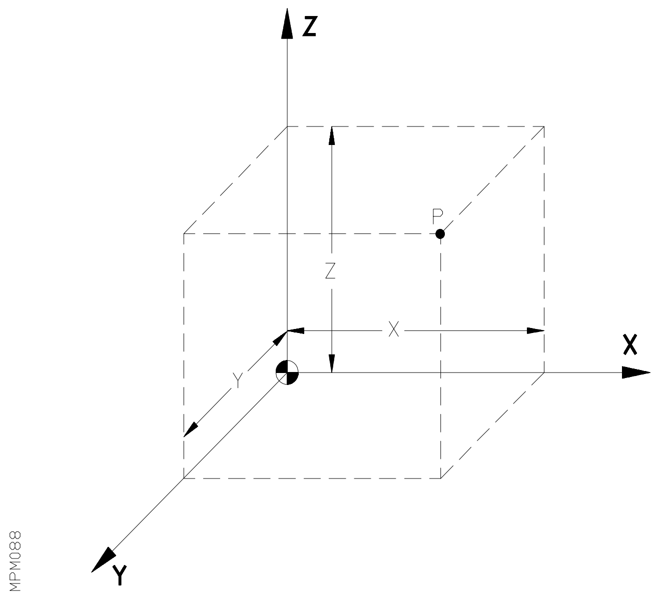
. In mm (V+/-4.3) (W+/-4.3), X+/-4.3, Y+/-4.3, Z+/- 4.3
. In inches (V+/-4.3) (W+/-3.4), X+/-3.4, Y+/-3.4, Z+/- 3.4

In other words, the axis coordinate values are programmed by the letters (V), (W), X,Y,Z followed by the coordinate value.

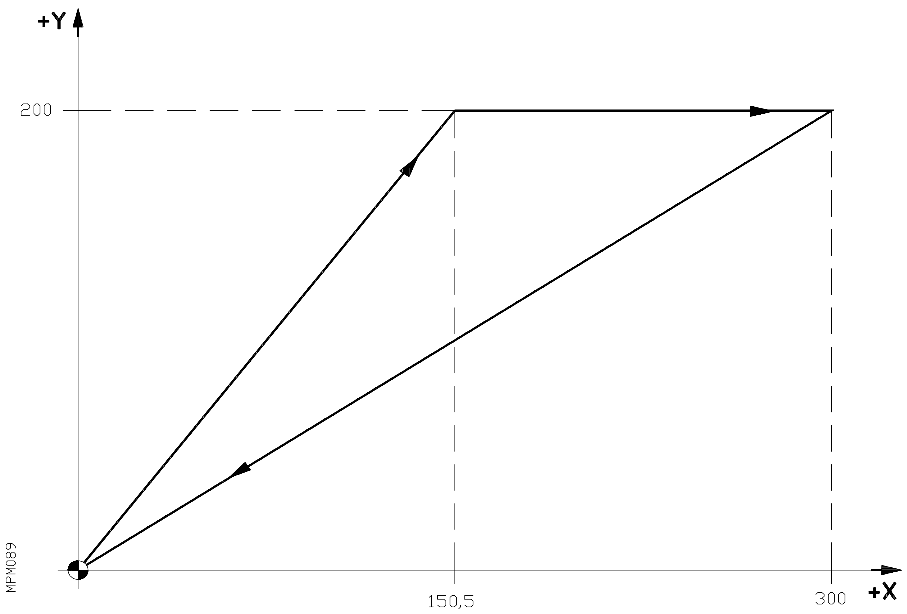
The **V**, **W** axes and the axis associated to both cannot be programmed in the same block.

The coordinate values programmed will be absolute or incremental depending on whether **G90** or **G91** is programmed.

There is no need to write the **+** sign in the case of positive coordinate values. The leading and trailing zeros of coordinate values may be omitted.



Example:



Absolute coordinate values

N10 G90 G01 X150.5 Y200
N20 X300
N30 X0 Y0

Incremental coordinate values

N10 G91 G01 X150.5 Y200
N20 X149.5
N30 X-300   Y-200

If the **4th axis (W)** or the **5th axis (V)** are rotary, the format will be:

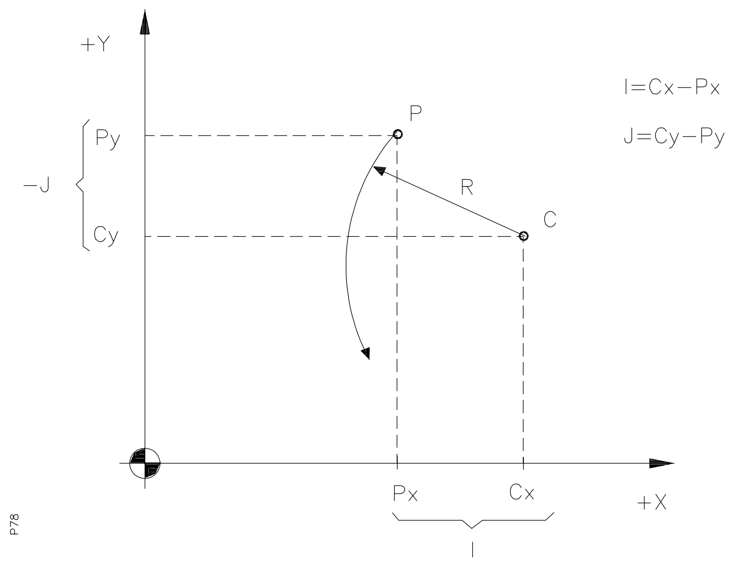
W +/-4.3
V +/-4.3

and will be programmed in degrees.

7.1.2. Center coordinates.

When working in circular interpolation the coordinates of center **I**,**J** must be programmed.

The values of **I** and **J** represent the distance from the starting point of the arc to the center of the circumference, according to axes **X**, **Y**.



The values of **I**,**J** are programmed with their sign. It is always necessary to program them, even though their value is zero.

7.1.3. Rotary axis

By means of machine parameters it is possible to determine whether the **4th axis W** or the **5th axis V** or both, are Rotary or Linear.

Likewise, should they be a Rotary axis it is possible to define if **HIRTH** toothing is available or not (only integer programming values are allowed), as well as whether the **4th axis W** is a Rollover Axis or not (programming between ± 360 degrees).

Type	4th axis W	5th axis V
ROTARY	P 600(1) = 1	P 616(1) = 1
ROLLOVER	P 606(1) = 1	Always
HIRTH	P 600(2) = 1	P 616(2) = 1

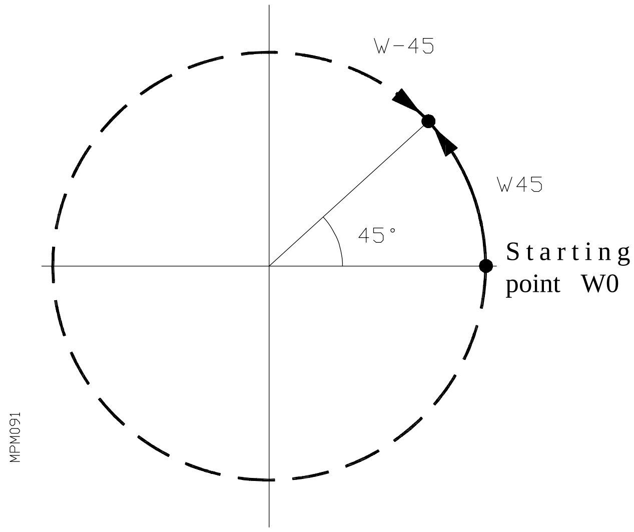
<u>4th axis W</u>

If the 4th axis (W) is rotary P600(1)=1 and parameter P606(1) is set to 0 a max. value of +/-8388.607 degrees can be programmed both in absolute coordinates G90 and relative coordinates G91. Lower limits can be set by P407 and P408.

Programming is identical to linear axes.

If P606(1)=1, rotary axis ROLLOVER, the counting will be reset to zero every time it rotates over 360 degrees.

When operating in G90 the sign identifies the direction of the rotation i.e. if the same value is programmed with different signs, the axis will rotate to the same point in both cases but following opposite directions.



Working in G90 the sign will be disregarded if P606(1)=1(ROLLOVER) and P600(2)=1 (Hirth toothing) and the CNC will rotate the axis to position by the shortest turn. This will also happen even when it is not a HIRTH rotary axis, as long as a value of 1 is assigned to machine parameter P619(8).

5th axis V

Similar to the indications for the 4th axis W, except that if it is ROTARY [P616(1) = 1] this implies that the axis is ROLLOVER.

If P620(6) = 1 the V axis will travel along the shortest path even though it is not HIRTH.

7.2. POLAR COORDINATES

Only movements in a plane (2 axes simultaneously) can be carried out when operating with polar coordinates.

If **3D** movements (in the space) are desired, they must be programmed in cartesian or cylindrical coordinates.

The format to identify a particular point of the plane with polar coordinates is:

In mm R+/-4.3 A+/-3.3

In inches R+/-3.4 A+/-3.3

R being the radius value and **A** the value of the angle (A in degrees), referred to the polar center.

When turning on and after **M02**,**M30** ,**EMERGENCY** or **RESET**, the CNC takes the point X0,Y0 as polar origin.

Every time there is a change of main plane during a program execution, the polar origin assumes the point of the coordinate origin of that plane.

When G18 Is programmed, the polar origin assumes the point X0 Z0.

When G19 is programmed, the polar origin assumes the point Y0 Z0.

When a circular interpolation is programmed with **G02**,**G03** the CNC takes the arc's center as the new polar origin.

With the function **G93**, any point of the plane can be preset as polar origin.

The values of **R** and **A** will be absolute or incrementaldepending on whether G90 or G91 are active.

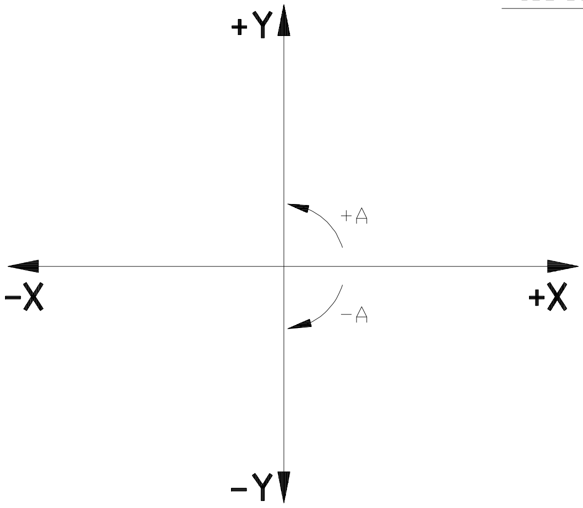
When a circular interpolation (G02,G03) is programmed the values of the angle A+/-3.3 and the values of the center referred to the arc's starting point must be entered.

Atention:

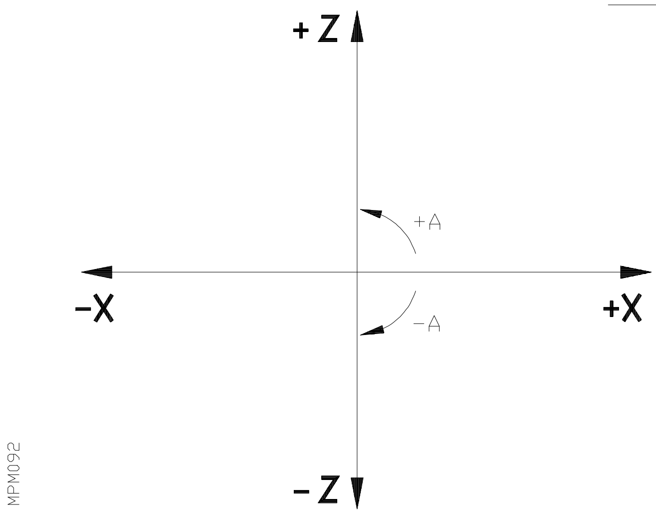
There is no need to write the **I**,**J**,**K** coordinates of the center referred to the starting point, if the arc's center is the polar origin point. Only the angle must be programmed.

DIRECTION AND SIGN OF THE ANGLES

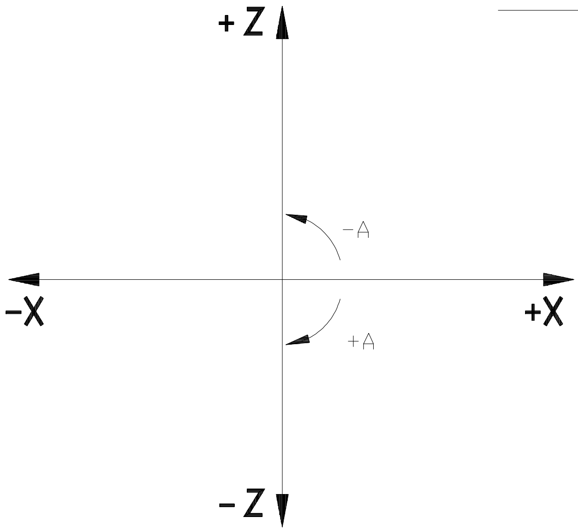
XY Plane



XZ Plane

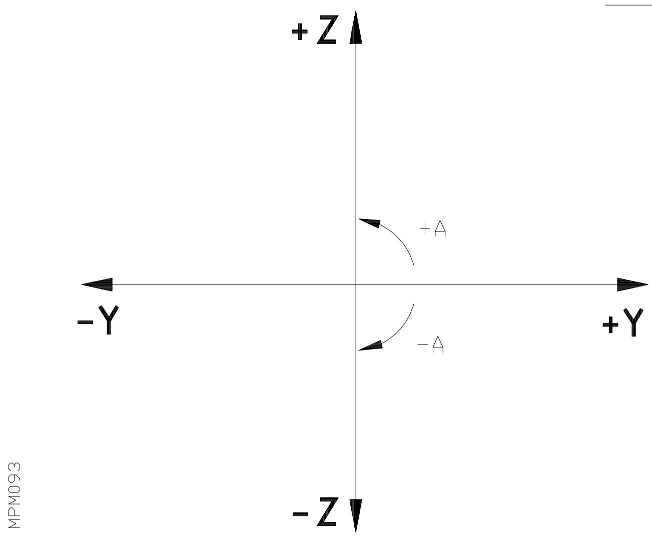


XZ Plane



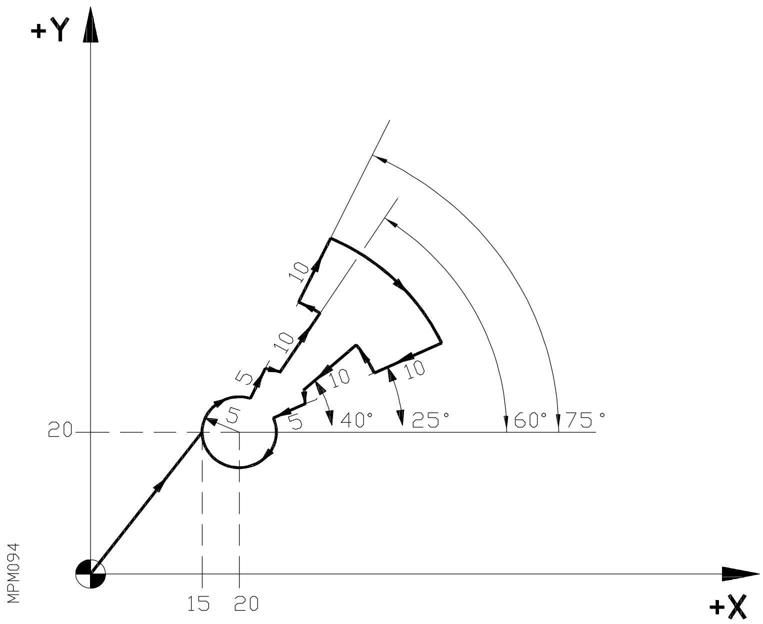
XZ PLANE with machine parameter P605(4)=0

YZ Plane



After the definition of the center of the circle (I,J) or the polar origin (G93 I,J) the angles counter-clockwise will be considered positive and the angles clockwise negative, except in the XZ plane when P605(4)=1

XZ PLANE with machine parameter P605(4)=0
Example:



The tool starts at point X0 Y0

N0 G93 I20   Y20 F150
N5 G01 G90 R5 A180 F150
N10 G02 A75 N15 G01 G91 R5
N20 G02 A-15
N25 G01 R10
N30 G03 A15
N35 G01 R10
N40 G02 A-50
N45 G01 R-10
N50 G03 A15
N55 G01 R-10
N60 G02 A-15
N65 G01 R-5
N70 G02 G90 A180
N75 G01   X0 Y0

7.3. CYLINDRICAL COORDINATES

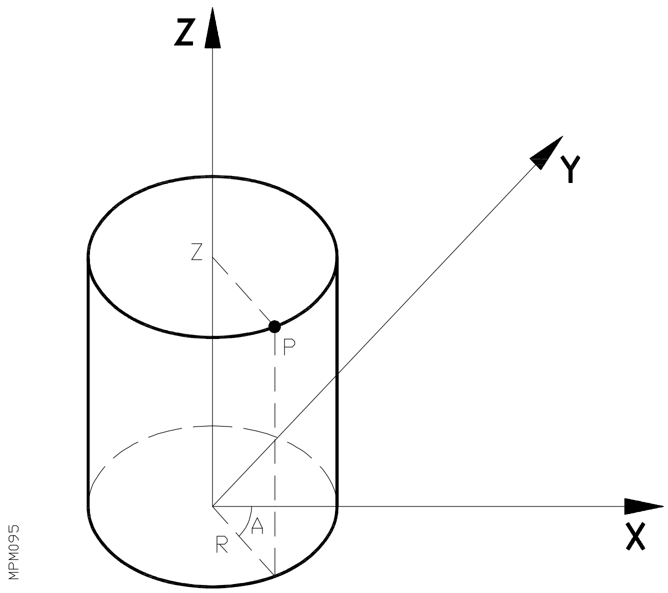
A point in the space can be defined by:

X Y Z cartesian coordinate values or in cylindrical coordinates.

The format to define cylindrical coordinates of a point is as follows:

Operating in G17 (plane XY): N10 G01 R.. A.. Z.

Where R,A define the projection of the point on the main plane in polar coordinates and Z is the value of the coordinate Z at that point.



The format for G18 (plane XZ) is: N10 G01 R... A...Y

and for G19 (plane YZ) is: N10 G01 R...A...X

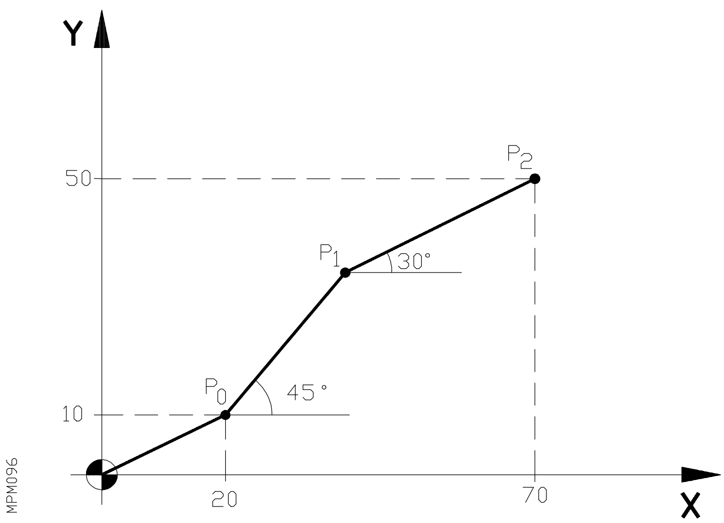
7.4. TWO ANGLES (A1,A2)

One point on the main plane whose coordinate values are not known can be identified by means of two angles if the coordinate values of the previous and next points along the path are known by using: A1, A2 XY (YZ) (XZ).

Where A1 is the angle of the exit path from the starting point (P0). A2 is the angle of the exit path from the intermediate point (P1). XY (YZ) (XZ) are the coordinates of the final point P2 according to the working plane.

The CNC calculates automatically the coordinates of P1.

Example:

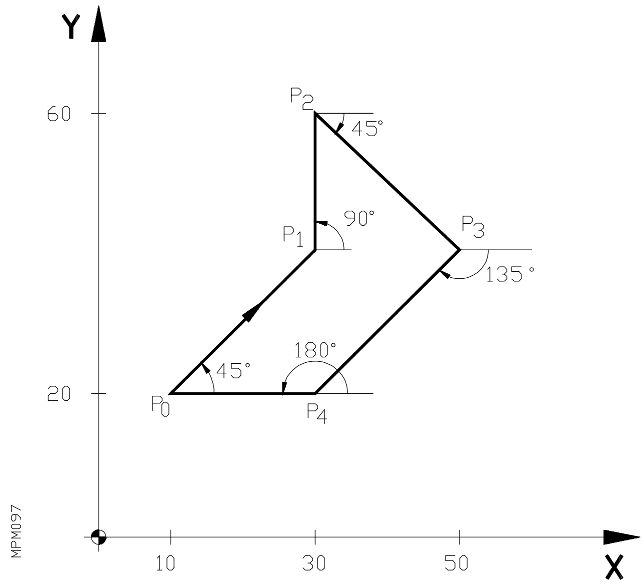


Starting point is X0 Y0

N10 X20 Y10 (Coordinates of P0)
N20 A45 A30 (Exit angle of P0 and P1)
N30 X70 Y50 (Coordinates of P2)

7.5. ANGLE AND ONE CARTESIAN COORDINATE

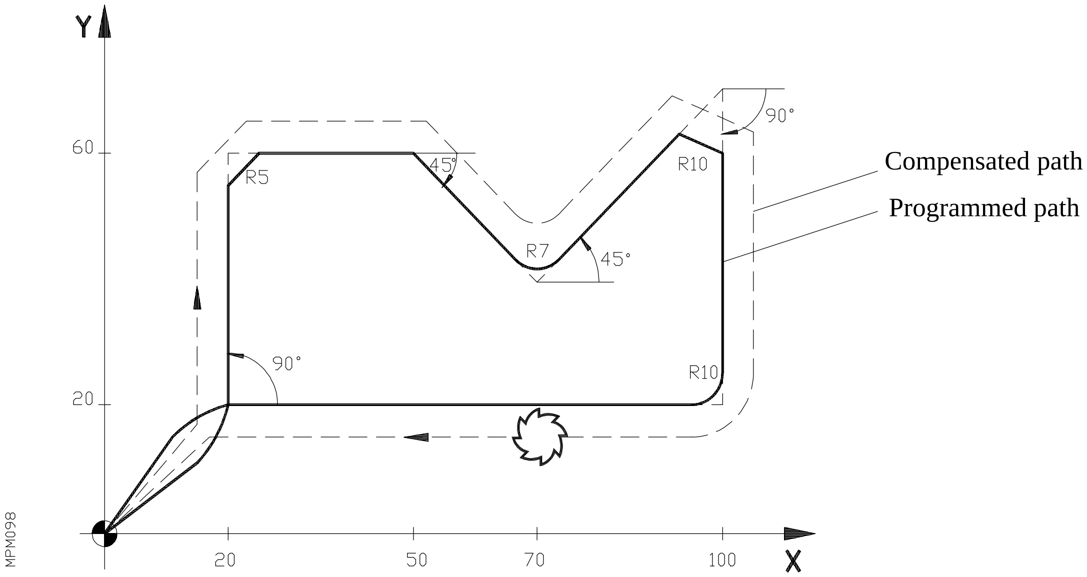
A point on the main plane can also be defined by the exit angle of the path in the previous point and one cartesian coordinate of the point which is to be defined.



Starting point P0 (X10 Y20)

```
N10 A45 X30      ; (Point P1)
N20 A90 Y60      ; (Point P2)
N30 A-45 X50     ; (Point P3)
N40 A-135 Y20    ; (Point P4)
N50 A180 X10     ; (Point P0)
```

When defining the points of a path, with two angles or one angle and one coordinate, roundings, tangential approaches and exists can be inserted.



Starting point X0 Y0 and tool's radius T1=5 mm.

N100 T1.1
N110 G37 R10 G41 X20 Y20
N120 G39 R5 A90 A0
N130 X50 Y60
N140 G36 R7 A-45 X70
N150 G39 R10 A45 A-90
N160 G36 R10 X100 Y20
N170 G38 R10 X20
N180 G40 X0 Y0 N190 M30

8. F. FEEDRATE PROGRAMMING

The axis feedrate is programmed with the letter "F" and its value depends on the currently selected work units, millimeters or inches, and type of feedrate, G94 or G95.

Metric programming:

	Format	Programming units	Minimum value	Maximum value
G94	F 5.4	F1= 1mm/min	F0.0001 (0.0001 mm/min)	F65535.000 (63535 mm/min)
G95	F3.4	F1= 1mm/rev.	F0.0001 (0.0001 mm/rev.)	F500.000 (500 mm/rev.)

When operating in inches, we recommend setting machine parameter P615(6) to "1" so the programming units in G94 are in inches/minute.

P615(6) = 0 Programming format F1 = 0.1 inch/min.
Maintaining compatibility with older versions which did not accept decimal feedrate values.

P615(6) = 1 Programming format F1 = 1 inch/min.

	P615(6)	Format	Programming units	Minimum value	Maximum value
G94	P615(6)=0	F 5.4	F1= 0.1inch/min	F0.001 (0.0001 inch/min)	F25801.1810 (2580.1181 inch/min)
	P615(6)=1	F 5.4	F1= 1 inch/min	F0.0001 (0.0001 inch/min)	F25801.1810 (25801.1810 inch/min)
G95	------	F2.4	F1= 1 inch/rev.	F0.0001 (0.0001inch/rev.)	F19.6849 (19.6849 inch/rev.)

By the same token, when operating in inches and with rotary axes, we recommend setting machine parameter P615(7) to "1" so the programming units in G94 are in degrees/minute.

	P615(6)=1 Programming units: Inches/min		
	P615(7)	**Only rotary axis**	**Rotary axis interpolated with a linear axis**
G94	P615(7)=0	F1= 2.54°/min	F1= 1 inch/min
	P615(7)=1	F1= 1°/min	F1= 1 inch/min

The machine's actual maximum feedrate may be limited to a lower value (see instruction book of the machine).

The machine's maximum working feedrate can be programmed directly or by using code F0.

Example :

On a machine with a maximum programmable working feedrate of 10,000 mm/min. it makes no difference whether F10.000 or F0 is programmed.

The programmed feedrate F is effective when operating on linear interpolation (G01) or circular interpolation (G02/G03). When operating on positioning (G00), the machine will move in rapid regardless of the F programmed.

The rapid speed is set for each axis during the final adjustment of the machine, the maximum possible value being 65.535 m/min. (See instruction book of the machine).

The programmed feedrate can be varied between 0% and 120% or between 0% and 100% according to P606(2) by means of the knob on the front panel of the CNC. When carrying out the tapping canned cycle G84 or when G33, G47 are activated or during probing movements (G75), this knob is cancelled and operation is at 100% of the programmed F.

9. (S) SPINDLE SPEED AND SPINDLE ORIENTATION

Code S can have three different meanings:

a) Spindle speed.

The spindle speed is programmed directly in rev/min. by means of code S4.

Any value may be programmed between S0 and S9999; i.e. between 0 and 9999 rev/min. This value is limited by the max. speed permitted by the machine; this limit is set by a machine parameter.

The instruction book of the machine must be consulted in each particular case. The controls + − on the front panel of the CNC may be used to achieve between 50 and 120% variation in programmed spindle speed.

These controls do not operate when carrying out the tapping canned cycle G84 or when function G33 is activated the speed being fixed at 100% of the programmed value.

b) Spindle orientation

If S4.3 is programmed after M19, it identifies the spindle's stopping point in degrees, referred to the encoder zero marker. The spindle will rotate according to parameters P601(7) and P700 until the point identified by S4.3 is reached.

The spindle must have an encoder in order to use this function.

c) Analogue output S proportional to F.

The CNC permits a special function applicable for example for controlling the BEAM in LASER machines, for which a value of 1 should be introduced into the machine parameter P619(3).

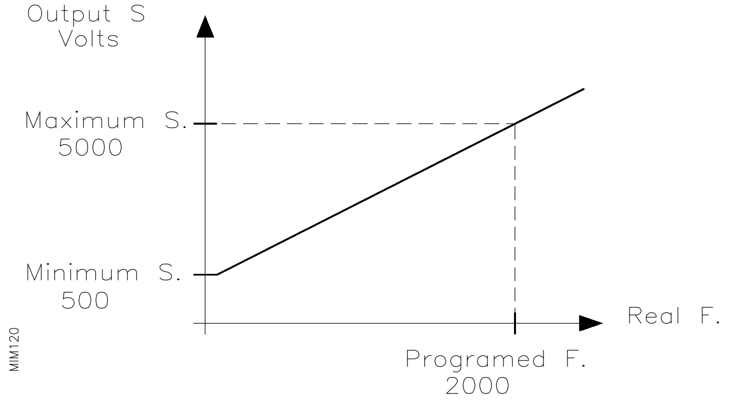
The function consists of sending an order proportional to the real speed of the machine axes through the analogue S output.

In this case, the format of the programme will be:

N4 G1 X ___ Y ___ F ___ S(minimum). (maximum) M3(M4)

Example:

N1234 G1 X100 Y80 F2000 S500.5000 M3



When there is no movement or when it is in G00, the CNC sends the minimum S, when the movement is the programmed F, it sends the maximum S and between these the CNC will send an S command proportional to the speed of the Real Feed.

10. <u>(T) TOOL PROGRAMMING</u>

The CNC has a table of **100** tools (00-99) for tool radius and length compensation.

The tool to be used is programmed by means of codes T2./T.2/T2.2

- Tool number.The two digits of code **T2.** or the two digits to the left of the decimal point of code **T2.2** may have any value between **00** and **99**. This value is used for selecting the required tool in the case of a machine with automatic tool changer, and may be limited to a value lower than 99 according to parameter **P701**.

- Tool compensation (table). The two digits to the right of the decimal point in codes T.2 and T2.2 may have any value between 00 and 99.

  When **G41** or **G42** is programmed, the CNC applies the addition of **R** and **I** stored at the programmed **T** address (00-99) as radius compensation value.

  If G43 is programmed, the CNC applies the addition of **L** and **K** stored, at the programmed **T** address (00-99) as length compensation value.

  If no **T** has been programmed, the CNC applies the address 00.00.

  The maximum compensation values are:

  R and L +/-1000.000 mm (+/-39.3699 inches)
  I and K +/-32.766 mm (+/-1.2900 inches)

  The tool's radius and length values are recorded in operation mode 8, TOOL TABLE. The values of **I**,**K** can also be checked and modified without stopping the execution of a cycle (see Operation Manual).

The values can be loaded as well by program, using code **G50**.

10.1. HOW TO USE CODES T2.2/T2/T.2

10.1.1. Machines without automatic tool changer

In the case of machines with manual tool change, the two digits of code **T2.** or the two digits to the left of the decimal point of code **T2.2 (00-98)** have no significance and any value between **0** and the value given to parameter **P701** may be programmed. It is recommended to assign the max. possible value (**99**) to this parameter.

The two digits to the right of the decimal point of codes **T.2** or **T2.2** (00-99) are used for selecting the required compensation value.

As soon as the CNC reads a code **T.2** or **T2.2**, it applies the new compensation values.

The CNC assumes that the machine does not have automatic tool changer if parameters **P601(1)** and P601(5) are set to 0.

10.1.2. Machines with automatic tool changer

The two digits of the code **T2.** or the two digits to the left of the decimal point of code **T2.2 (00-98)** are used for selecting the required tool.

When the CNC reads a **T** value **(00-98)** which is different from that previously programmed, it sends it to the interface, in **BCD** code. If the **T** value is the same as that programmed, it is ignored. If two different tools are programmed in succession without code **M06** between them, the CNC will show error code **53**, unless the second tool programmed is the one engaged in the spindle. In this case, the CNC will apply the new values.

Error code **53** will also be generated if two **M06** are programmed without a **T** code between them, unless **P139** is set to **0**. To resume the operation:

- Select **JOG** mode
- Key-in the number of the tool located in the spindle **T2**.
- Key-in **P00**
- Press **ENTER**

This way the tool located in the spindle has been confirmed to the CNC.

The two digits to the right of the decimal point are used for selecting the required compensation value in the tool table.

If the figure to the left of the decimal point (external selection of tool) is the same as the last figure programmed, as soon as the CNC reads code **T2.2**, it applies the compensation values corresponding to the new code (**00-99**). If it is only desired to change the compensation value without changing the working tool, the same external tool number as that programmed previously to the left of the decimal point plus the new compensation value to the right must be programmed. Only **T.2** can also be programmed. This way there is no external output of any code but the CNC assumes the newcompensation value for the previous tool.

If **T2.2** is programmed and the external tool selection code (figure to the left of the decimal point) is different from that previously programmed, the CNC does not assume the new compensation values until the actual tool change takes place; i.e. until **M06** is executed.

To set the **RANDOM** tool magazine in initial conditions, the following has to be done in **TEACH-IN** mode:

- **T99.xx**
- **START CYCLE**

This way, tool **1** is in position **1**, tool **2** in position **2**, etc. Even if the CNC is turned off, when being turned on again it remembers the actual position of the tools in the magazine.

11. <u>(M) MISCELLANEOUS FUNCTIONS</u>

The miscellaneous functions are programmed by means of code **M**.

The miscellaneous functions are output in **BCD** code (M00/M99) or in binary code (M00/M254) depending on the value assigned to the machine parameter **P617(8)**.

Miscellaneous functions M41, M42, M43, M44 implicit with **S** cannot be programmed. The CNC also has **15** decoded outputs for miscellaneous functions.

These outputs will be assigned to the required functions during the final adjustment of the CNC to the machine.

The miscellaneous functions that are not assigned a decoded output are always performed at the beginning of the block in which they are programmed.

In assigning a decoded output to any miscellaneous functions, a decision is also made as to whether it is to be performed at the beginning or at the end of the block in which it is programmed.

Up to a maximum of **seven** miscellaneous functions may be programmed in one block.

When more than one miscellaneous function is programmed in a block, the CNC performs them consecutively in the order in which they are programmed.

Some of the miscellaneous functions have an internal meaning assigned to them in the CNC.

11.1. M00. PROGRAM STOP

When the CNC reads code **M00** in a block, it halts the program. The **CYCLE START** key must be pressed for it to resume.

It is recommended that this function be set in the table of decoded **M** functions so that it is performed at the end of the block in which it is programmed (see Installation and Start-up Manual).

11.2. M01. CONDITIONAL STOP OF PROGRAM

Same as **M00** except that the CNC only takes it into account if the “Optional stop” input is activated.

11.3. M02. END OF PROGRAM

This code indicates end of program and performs a general reset function of the CNC (reversion to initial state). It also acts as an **M05**. As in the case of M00, it is recommended that this function be set so that it is executed at the end of the block in which it is programmed.

11.4. M30. END OF PROGRAM WITH RETURN TO BEGINNING

Same as **M02** except that the CNC goes back to the first block at the beginning of the program. It also acts as an **M05**. If parameter P609(3)=0, when a CNC **RESET** is carried out, the CNC will send out code **M30**.

11.5. M03. CLOCKWISE START OF THE SPINDLE

This code means that the spindle starts running clockwise. As explained in the correspon-ding section, the CNC executes this code automatically in the machining canned cycles. It is recommended that this function be set so that it is executed at the beginning of the block in which it is programmed.

11.6. M04. COUNTER CLOCKWISE START OF THE SPINDLE

Same as **M03** except that the spindle rotates in the opposite direction.

11.7. M05. SPINDLE STOP

It is recommended that this be set so that the CNC executes it at the end of the block in which it is programmed.

11.8. M06. TOOL CHANGE CODE

a) MACHINE WITHOUT AUTOMATIC TOOL CHANGER

- If P601(1) and P601(5) are set to zero(MACHINE WITHOUT AUTOMATIC TOOL CHANGE), the CNC sends out codes M05 and M06 when code M06 is read.

  According to the value assigned to P601(8), it will stop or not the program (like M00).

  P601(8)= 1 It stops the program
  P601(8)= 0 It does not stop the program

b) MACHINE WITH AUTOMATIC TOOL CHANGER

- If P601(1) or P601(5) are set to 1, the code M06 has to be programmed alone in a block. When reading this code the CNC, in modes: AUTOMATIC, SINGLE BLOCK, TEACH- IN, will execute the following sequence:

- It will send out the code M19 and will apply a residual **S** analog output (defined by P601(7) and P700) to the spindle.

- It will move the axes X,Y,Z,(W) to the positions fixed by P900, P901, P902 and P903, according to the order defined by P702, P703, P704 and P705.

- It will send out the code M06, cancelling the residual S analog output afterwards.

- If parameter P709 has a value between 1 and 99, the CNC will automatically jump to a subroutine identified by P709. If P709 is zero, there is no jump.

  In SINGLE BLOCK mode, the CYCLE START key must be pressed as many times as different functions are involved with M06. In JOG mode, the CNC will send the code M19, will apply to the spindle the residual analog output and will send M06, previously cancelling the residual analog output if the axes have been correctly positioned for the tool change. If any axis is out of position, the CNC will give error code 51.

- Same as in the previous section, it will stop the program or not, according to status of parameter P601(8).

Atention:

It is recommended that M06 be set so that the CNC executes it at the end of the block in which it is programmed.

11.9. M19. ANALOG S OUTPUT (CREEP) FOR TOOL CHANGE AND SPINDLE ORIENTATION

There are different possibilities for operating with M19.

a) If **M19** is not followed by **S4.3**, when this function is executed the CNC will send out code M19 and will apply to the spindle an analog output defined by P601(7) and P700. The remaining analog output is deleted when execution any other **M** or **programmed S4**.

b) If **S4.3** is programmed right after **M19** the CNC will send out code **M19** and will apply to the spindle an analog output defined by **P601(7)** and **P700** until the spindle reaches the position identified in degrees by **S4.3**, referred to the zero marker. In this case, no other information is allowed in the same block.

c) If the machine parameter **P615(8)=1**, on executing function **M19**, the CNC will search for machine reference in the spindle at the same time as the movement of the axes.

d) The machine parameter **P916** determines the position of the spindle stop when the functions **M06** are executed (tool change) or M19 without programming **S**, in both cases, whenever the machine is fitted with an encoder on the spindle, and due to this, parameter P800 must have a different value from 0. If a value of 0 is given to it, the CNC ignores all positions. The remaining assignable values go from 0.001 to 360.

e) Machine parameters **P917** and **P918** determine the lower and upper limits of the range of travel of the spindle, respectively, with **M19**. When a movement is executed into the forbidden area, the CNC will indicate error **74**, this being the area which exists between both limits.

More information on the use of **M19** is to be found in the 8025/30 CNC START UP MANUAL.The application of this feature involves the installation of rotary encoder on to the spindle.

11.10. M22.M23,M24,M25. OPERATION WITH PALLETS

If parameter **P603(3)** is set to **1**, the CNC can manage the operation of pallets in the machine. This means that M22,M23,M24 and M25 acquire a precise meaning.

M22 - Code to load the part in one end of the table (X axis)
M23 - Code to unload the part in the same point as M22
M24 - Code to load the part in the other end of the table
M25 - Code to unload the part in the same point as M24

When the CNC reads any of these four codes, it executes the following sequence:

1. The CNC sends out the code **M21** if parameter P605(3) is 1.

2. Shifts the fourth axis **(W)** to the position identified by parameter P904 if P605(1) is 0.

3. Shifts the **X** axis to the position identified by P905 for **M22** and **M23** or **P906** for M24 and **M25**, as long as **P611(7)=0**.

4. Shifts the **Z** axis to the position identified by **P907** if **P605(2)** is **1**.

5. When all the axes are in position, the CNC sends out the relevant code (M22,M23,M24 or M25). This codes are used by the cabinet to load and unload the part. During this process the **FEED-HOLD** signal must be applied to the CNC.

6. If parameters related to this function (P710,P711,P712,P713) have a value between 1 and **99** the CNC will automatically jump to the subroutine identified by these parameters after the **M** function has been executed. If they are set to **0**, there is no jump.

Example:

N5   M23
N10 M24

Block N5. The CNC sends out **M21** if P605(3) and will place the just machined part in unloading position by moving the axes **W**,**X** and **Z** to the positions set by **P904**, **P905** and **P906**. Then it will send out the code **M23** so that the interface executes the unloading sequence.

If parameter **P711**, corresponding to M23, has a value of **5**, the CNC will execute subroutine number **5**.

Block N10. The CNC will place the machine ready to load the new part by moving the axis W,X and Z to the position set by **P904**,**P906** and **P907**. Then it will send out the code **M24** so that the interface executes the loading sequence.

If parameter **P712**, corresponding to **M24**, has a value **0**, the sequence is now finished.

The sequence described is executed in modes **AUTOMATIC**, **SINGLE BLOCK**, and **TEACH-IN**. In **SINGLE BLOCK** mode, the key must be pressed as many times as different operations there are.

In **JOG** mode, the CNC will move the last axis **X**, or **Z** and then it will send the relevant code (M22,M23,M24,M25) if the axis **W**, or **W** and **X** were previously positioned. Otherwise if will give error code **51**.

No other information can be programmed in a block containing **M22**,**M23**,**M24** or **M25**.

12. STANDARD AND PARAMETRIC SUBROUTINES

A subroutine is a part of a program which is suitably identified and can be called in for execution from any position in a program.

A subroutine may be called in several times from different positions in the program or from different programs.

A single call can be used to repeat the execution of a subroutine up to 255 times.

A subroutine may be stored in the memory of the CNC as an independent program or as part of a program. Standard and parametric subroutines are basically identical. The difference between them is that up to 10 arithmetic parameters can be defined in the call block (G21 N2.2) of parametric subroutines.

In standard subroutines the parameters cannot be defined in the call block (G20 N2.2).

The max. number of parameters of a subroutine (standard or parametric) is 255 (P0-P254).

12.1. IDENTIFICATION OF A STANDARD SUBROUTINE

A standard (non-parametric) subroutine always begins with a block which contains function G22. The structure of the subroutine opening block is: N4 G22 N2

N4 : Block number
G22: Defines the beginning of a subroutine
N2 : Identifies the subroutine (may be any number between N0 and N99)

This block cannot contain additional information.

Atention:

Two standard subroutines having the same identification number but belonging to different programs cannot be present at the same time in the memory of the CNC, although a standard subroutine and a parametric subroutine may be identified by the same number.

The subroutine opening block is followed by programming the blocks required.

A standard subroutine can contain parametric blocks.

Example:

```
N0  G22 N25
N10 X20
N15 P0=P0 F1 P1
N20 G24
```

A subroutine must always end with a block of the form: N4 G24.

N4 : Block number
G24: End of subroutine

No other additional information can be programmed in that block.

12.2. CALLING IN A STANDARD SUBROUTINE

A standard subroutine may be called in from any program or other subroutine (standard or parametric). Calling in a standard subroutine is achieved by function G20.

The structure of a call block is: N4 G20 N2.2

N4 : Block number

G20 : Subroutine call

N2.2 : The two figures to the left of the decimal point identify the number of the subroutine called in (00-99).The two figures on the right of the decimal point indicate the number of times the subroutine is to be repeated (00- 99). Unless it is programmed by a parameter, in which case the limits are 0 and 255.

However, when no number is indicated, the subroutine will be executed only once.

No other additional information can be programmed in the block calling in a standard subroutine.

12.3. IDENTIFICATION OF A PARAMETRIC SUBROUTINE

A parametric subroutine always begins with function G23.

The structure of the first block of a parametricsubroutine is: N4 G23 N2

N4 : Block number
G23 : Defines the beginning of a parametric subroutine.
N2 : Identifies the parametric subroutine (may be any number between N00 and N99).

Atention:

Two parametric subroutines having the same number but belonging to different programs cannot be present at the same time in the memory of the CNC, although it is possible for a normal subroutine and a parametric subroutine to be identified by the same number.

The above block is followed by programming the blocks required. A parametric subroutine must always end with a block of the form N4 G24.

N4 : Block number
G24 : Defines the end of a subroutine (standard or parametric).

No other additional information can be programmed in that block.

12.4. CALLING IN A PARAMETRIC SUBROUTINE

A parametric subroutine may be called in from a main program or from another subroutine (standard or parametric).

The calling of a parametric subroutine is achieved by function G21. The structure of the call block is:

N4 G21 N2.2 P2=K+/-5.5 P2=K+/-5.5 P2=K+/-5.5 ......

N4 : Block number

G21 : Call for parametric subroutine

N2.2 : The two figures to the left of the decimal point identify the number of the parametric subroutine called in (00-99).

The two figures to the right of the decimal point indicate the number of times the parametric subroutine is to be repeated (00-99).

If a parameter is programmed instead of the two figures on the right of the decimal point,the former can have a value between 0 and 255.

When no number is indicated, the subroutine will be executed only once.

P3 :Value of the parameter.

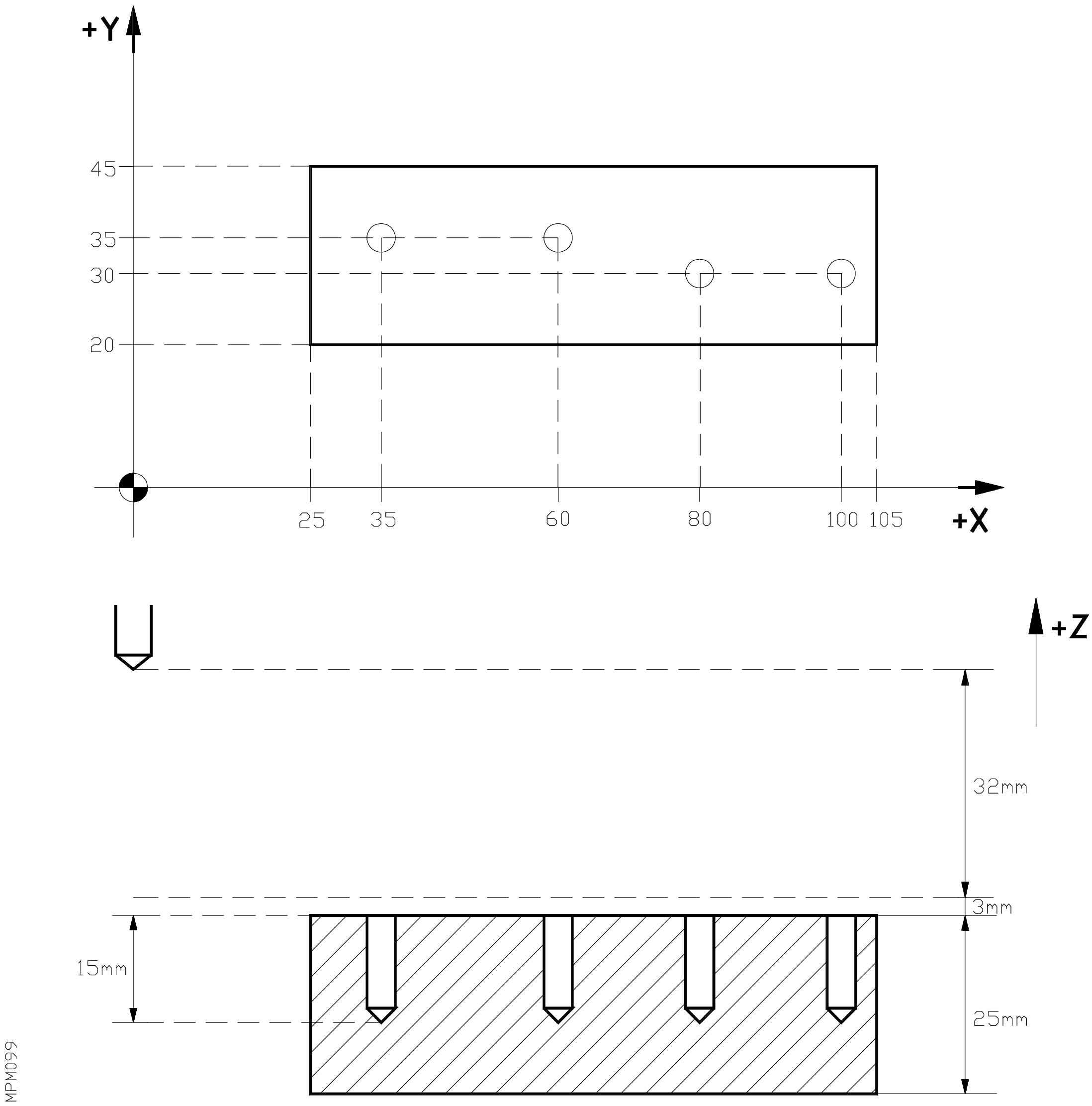
K+/-5.5 :Value assigned to the arithmetic parameter. Write K after the sign = when a constant must be assigned to a parameter. In this block values can be assigned to a maximum of 10 parameters.

When the same parametric subroutine is executed several times in succession, for example:

G21 N2.12 P2=K5 P4=K15 P6=K25

Once each repetition has been completed, except the last one, the values of the arithmetic parameters assigned in the call block are recovered, even though they have been assigned different values throughout the subroutine.

Example of use of standard subroutines without parameters.



This example concerns the drilling of four holes 15 mm deep.

```
N0    G90  G00 X35  Y35  M03
N5    G22  N1
N10  Z-32
N15  G01  Z-50  F100
N20  G04   K1.0
N25  G00  Z0
N30  G24
N35  X60
N40  G20  N1.1
N45  X80  Y30
N50  G20  N1.1
N55  X100
N60  G20  N1.1
N65  X0 Y0  M05
N70  M30
```

This same example can be programmed so that subroutine N1 is not part of the main program.

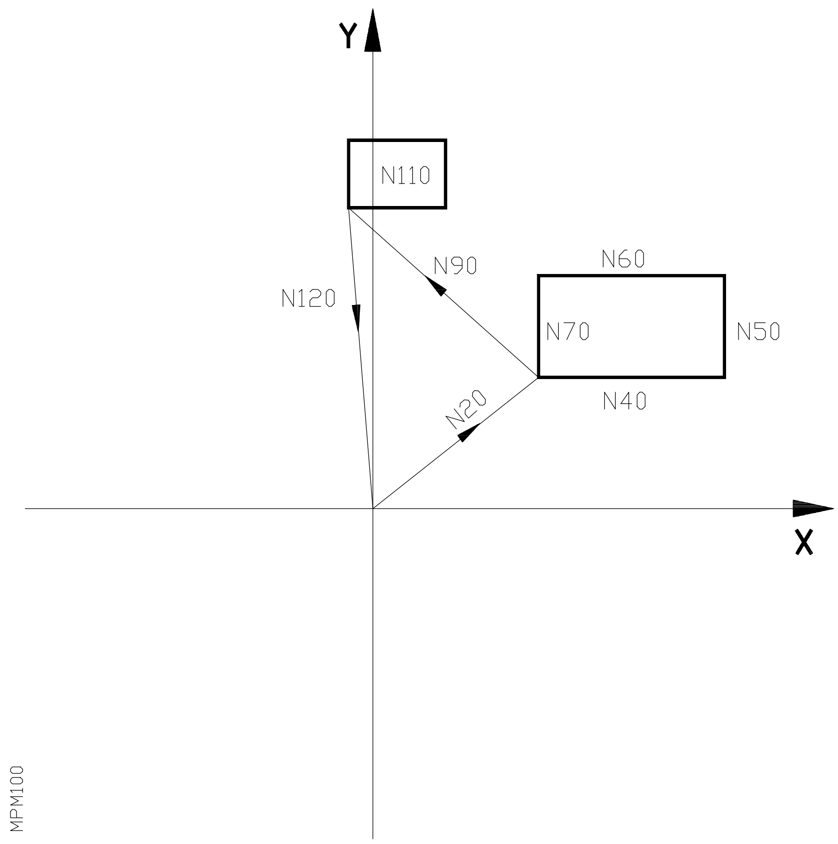
```
P 0 0 0 0 1

N0     G90  G00  X35  Y35 M03
N5     G20  N1.1
N10   X60
N15   G20  N1.1
N20   X80  Y30
N25   G20  N1.1
N30   X100
N35   G20  N1.1
N40   X0    Y0 M05
N45   M30
```

```
P 0 0 0 0 2

N100 G22 N1
N105 Z-32
N110 G01 Z-50  F100
N115 G04 K1.0
N120 G00 Z0
N125 G24
```

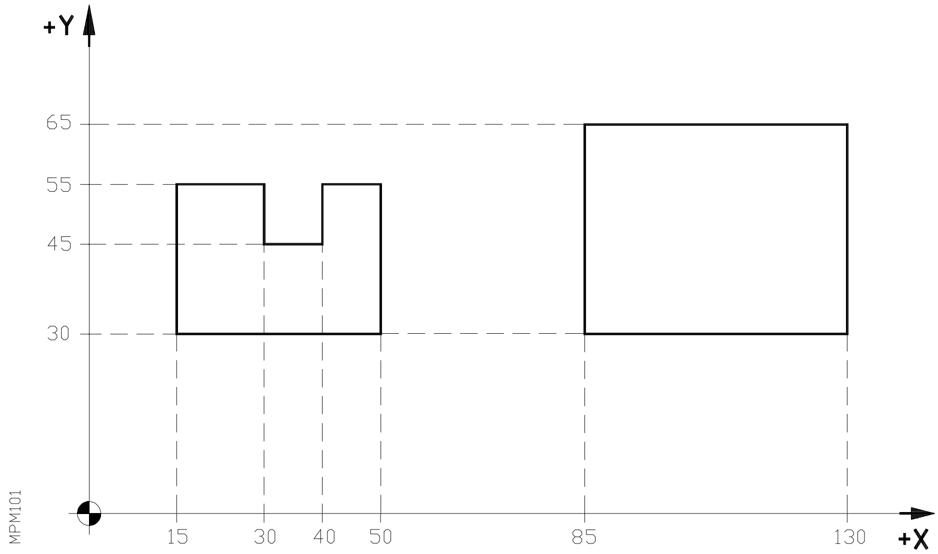
Example of use of standard subroutines with parameters .

Theoretical path without taking into account the tool diameter.

N10 P0=K48 P1=K24
N20 G1 X40 Y32 F0
N30 G22 N10 .... (Definition of standard subroutine)
N40 G91 XP0 F500
N50 YP1
N60 X-P0
N70 Y-P1
N80 G24 .... (End of subroutines)
N90 G90 X-6 Y72
N100 P0=K24 P1=K16
N110 G20 N10.1 .... (Call of standard subroutine)
N120 G1 G90 X0 Y0 F0
N130 M30 .... (End of program)



Example parametric subroutines using parameters

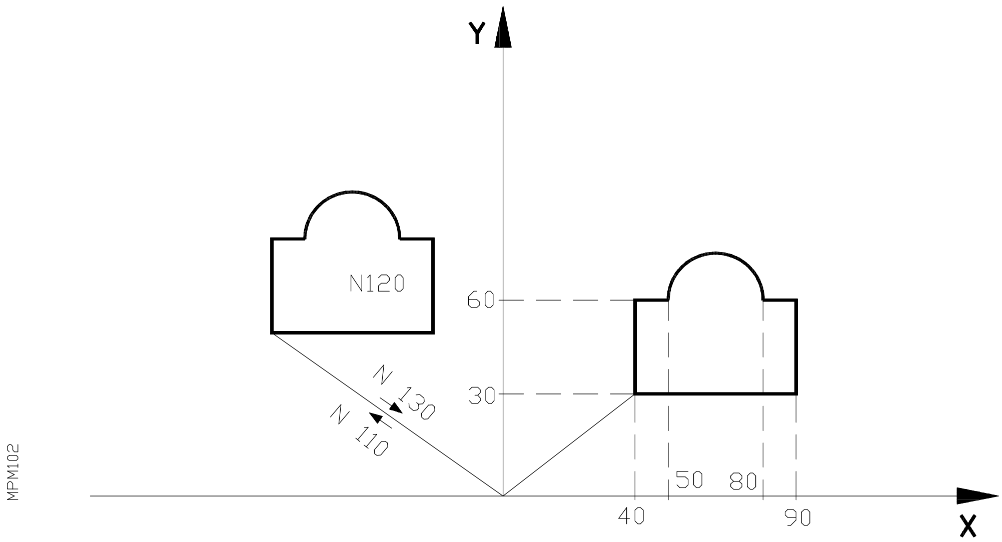


This example involves carrying out the two machining tasks illustrated, using the same parametric subroutine. The tool is supposed to be 100 mm above the surface of the part and the machining depth to be 10 mm.

```
P 0 0 0 0 1

N0    G90  G00  X15  Y30  M03
N5    Z-97
N10  G01  Z-110  F100
N15  G21   N1.1 P0=K25 P6=K15 P30=K-10 P13=K10 P14=K10 P15=K10
                  P50=K-25 P99=K-35
N20  G90   G00  Z0
N25  X85   Y30
N30  Z-97
N35  G01   Z-110
N40  G21   N1.1  P0=K35 P6=K45 P30=K0  P13=K0 P14=K0 P15=K0
                   P50=K-35 P99=K-45
N45  G90  G00  Z0
N50  X0 Y0 M05
N55  M30


P 0 0 0 0 2

N100  G23  N1
N105  G01  G91 YP0  F100
N110 XP6
N115 YP30
N120 XP13
N125 YP14
N130 XP15
N135 YP50
N140 XP99
N145 G24
```

Example of parametric subroutine without using parameters

Starting point X0 Y0



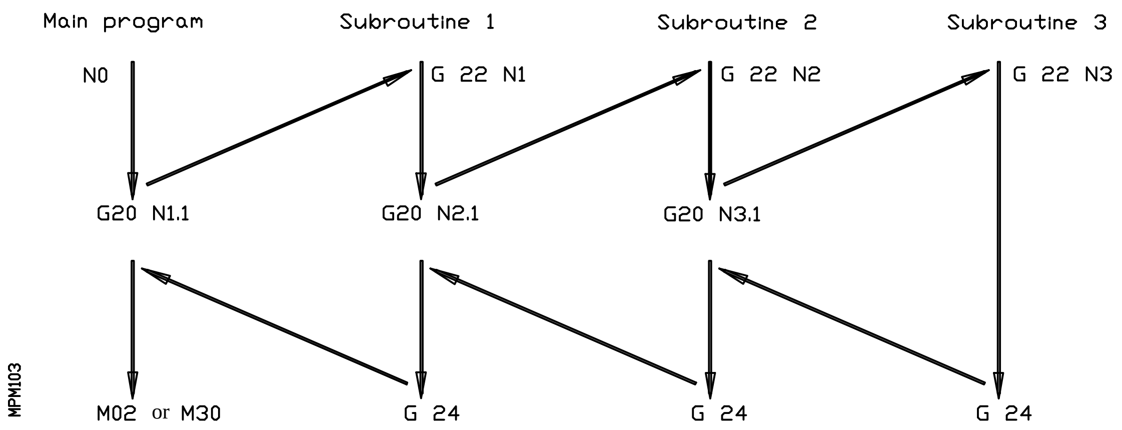

```
N10  G90 G01 X40 Y30 F0
N20  G23 N8 .........................................      (Definition of parametric subroutine)
N30  G01 G91 X50 F500
N40  Y30
N50  X-10
N60  G03 X-30 Y0 I-15 J0
N70  G01 X-10
N80  Y-30
N90  G24 ................................................   (End of subroutine)
N100 G01 G90 X0 Y0 F0
N110 X-70 Y50
N120 G21 N8.1 .....................................       (Call for subroutine)
N130 G01 G90 X0 Y0 F0
N140 M30 .............................................      (End of program)
```

When Block 120 is read, the CNC will execute once subprogram N8 which is defined between blocks 30 and 80.

12.5. NESTING LEVELS

From a main program or from a subroutine (standard or parametric) it is possible to call in a subroutine, from this a second subroutine, from the second a third, and so on up to a maximum of 15 levels of nesting. Each level may be repeated 255 times.

Subroutine linking diagram



12.6. EMERGENCY SUBROUTINE

If machine parameter P727 has a value between 1 and 99 and the external STOP input is activated while running a program, the CNC will interrupt the program and jump to execute the subroutine whose number has been assigned to P727.

13. PARAMETRIC PROGRAMMING. OPERATIONS WITH PARAMETERS

The CNC has 255 parameters (P0-P254) with which the following operation can be performed.

- Programming of parametric blocks
- Different operating
- Jumps within a program

The parametric blocks can be written in any part of the program. By means of machine parameters, it is possible to determine if the range of arithmetic parameters, included between P150 and P254 are only for **READING** or not.

The operations which can be made between parameters are as follows:

F1 : Addition
F2 : Subtraction
F3 : Multiplication
F4 : Division
F5 : Square root
F6 : Square root of the addition of the squares
F7 : Sine
F8 : Cosine
F9 : Tangent
F10 : Arc tangent
F11 : Comparison
F12 : Entire part
F13 : Entire part plus one
F14 : Entire part minus one
F15 : Absolute value
F16 : Complementation
F17 : Special functions
F18 : Special functions
F19 : Special functions
F20 : Special functions
F21 : Special functions
F22 : Special functions
F23 : Special functions
F24 : Special functions
F25 : Special functions
F26 : Special functions
F27 : Special functions
F28 : Special functions
F30 : AND
F31 : OR
F32 : XOR
F33 : NOR

The use of the parameter is described below:

PREDEFINED ARITHMETIC PARAMETERS

There are parameters whose value depends on the status of the CNC.

P100. PARAMETER INDICATING THE FIRST TIME

This parameter takes the value of 0, every time a program is run for the first time.

P101. PARAMETER INDICATING OPERATING MODE

The value of this parameter, is defined by operating mode active in the CNC.

Active mode	**Submode**	**Value taken P101**
Automatic		0
Single block		1
Teach in		3
Dry run	0	4
	1	5
	2	6
	3	7
	4	8

Assignments

Any value can be assigned to a parameter.

a) N4 P1 = P2

The indicates that P1 takes the value of P2, while P2 keeps the value it had.

b) N4 P1 = K1.5

P1 takes the value 1.5 K identifies a constant. Constants can have values comprised between +/-99999.999.

c) N4 P1 = X

P1 takes the theoretical value of the actual position of the X axis.

d) N4 P1 = Y

P1 takes the theoretical value of the actual position of the Y axis.

e) N4 P1 = Z

P1 takes the theoretical value of the actual position of the Z axis.

f) N4 P1 =W

P1 takes the theoretical value of the actual position of the W axis.

g) N4 P1 = T

P1 takes the actual value of the clock (execution time) in hundredths of a second. This assignation entails the cancellation of the radius compensation (G41 or G42).

h) N4 P1 - 0X

P1 takes the theoretical coordinate of the X axis, with respect to the machine zero where the CNC is situated.

i) N4P1 = 0Y

P1 takes the theoretical coordinate of the Y axis, with respect to the machine zero where the CNC is situated.

j) N4P1 = 0Z

P1 takes the theoretical coordinate of the Z axis, with respect to the machine zero where the CNC is situated.

k) N4P1 = 0W

P1 takes the theoretical coordinate of the 4th W axis, with respect to the machine zero where the CNC is situated.

l) N4P1 = 0V

P1 takes the theoretical coordinate of the 5th V axis,with respect to the machine zero where the CNC is situated.

In these last assignments, the measuring units taken by arithmetic parameter, depend upon the value assigned to the machine parameter P618(8).

If we assign the value 1 to this machine parameter, when the assignment parameter block is executed, of the P1 = 0X type: P1 takes the value of the X coordinate, with respect to the machine zero point, either in millimeters or inches, depending on the units of measure used.

Nevertheless, if we assign the value 0, when P1=0X is executed, P1 takes the value of the X coordinate with respect to the machine zero, always in millimeters, without taking into consideration the units of measure which are being used (mm or inches).

If any of the axes are **ROTARY**, the value taken by the parameter will always be in degrees.

m)N4P1 = H (Value in HEXADECIMAL)

P1 takes the value in HEXADECIMAL indicated after H.
Possible values of H: 0/FFFFFFFF.

Operations

F1 Addition

Example: N4 P1 = P2 F1 P3

P1 takes the value of the addition of P2 and P3, i.e. P1= P2 + P3. N4 P1 = P2 F1 K2 can also be programmed, i.e. P1 takes the value of P2 + 2. The letter K identifies a constant for instance:

K1 means value 1
K1000 means value 1000

The same parameter can be as an addend and as the result i.e., N4 P1 = P1 F1 K2. This means that P1 = P1 + 2

F2 Subtraction

N4 P10 = P2 F2 P3 —> P10 = P2 - P3
N4 P10 = P2 F2 K3 —> P10 = P2 - 3
N4 P10 = P10 F2 K1 --> P10 = P10 - 1

F3 Multiplication

N4 P17 = P2 F3 P30 —> P17 = P2 x P30
N4 P17 = P2 F3 K4 —> P17 = P2 x 4
N4 P17 = P17 F3 K8 —> P17 = P17x 8

F4 Division

N4 P8 = P7 F4 P35 —> P8 = P7 : P35
N4 P8 = P2 F4 K5 —> P8 = P2 : 5
N4 P8 = P8 F4 K2 —> P8 = P8 : 2

F5 Square root

N4 P15 = F5 P23 —> P15 = $\sqrt{P23}$

N4 P14 = F5 K9 —> P14 = $\sqrt{9}$

N4 P18 = F5 P18 —> P18 = $\sqrt{P18}$

F6 Square root of the addition of the square

N4 P60 = P2 F6 P3   —> P60 = $\sqrt{P2\ 2 + P3\ 2}$

N4 P50 = P40 F6 K5 —> P50 = $\sqrt{P40\ 2+ 5\ 2}$

N4 P1 = P1 F6 K4   —> P1   = $\sqrt{P1\ 2 + 4\ 2}$

F7 Sinus

N4 P1 = F7 P2 —> P1 = Sen P2

The angle has to be programmed in degrees.

N4 P1 = F7 K5 —> P1 = Sen 5 degrees

F8 Cosinus

N4 P1 = F8 P2   —> P1 = Cos P2
N4 P1 = F8 K75 —> P1 = Cos 75 degrees

F9 Tangent

N4 P1 = F9 P2   —> P1 = tg P2
N5 P1 = F9 K30 —> P1 = tg 30 degrees

F10 Arc tangent

N4 P1 = F10 P2   —> P1 = arc tg P2 (result in degrees)
N4 P1 = F10 K0.5 —> P1 = arc tg 0.5

F11 Comparison

It compares different parameters, or a parameter and aconstant, and activates the conditional jumps flags. Its application will be described in the conditional jumps section.

N4 P1 = F11 P2

If P1 = P2 the if zero jump flag is activated. If P1 => P2 the if => jump flag is activated. If P1 < P2 the if < jump flag is activated N4 P1 = F11 K6 can also be programmed.

F12 Entire part

N4 P1 = F12 P2 —> P1 takes the entire part of P2 as its value
N4 P1 = F12 K5,4 —> P1 = 5

F13 Entire part plus one

N4 P1 = F13 P2 —> P1 takes the entire part of P2 plus one as its value
N4 P1 = F13 K5,4 —> P1 = 5 + 1 = 6

F14 Entire part minus one

N4 P1 = F14 P27 —> P1 takes the entire part of P2 minus one as its value
N4 P5 = F14 K5,4 —> P5 = 5 - 1 = 4

F15 Absolute value

N4 P1 = F15 P2 —> P1 takes the absolute value of P2
N4 P1 = F15 K-8 —> P1 = 8

F16 Complementation

N4 P7 = F16 P20 —> P7 takes the complemented value of P20, i.e. P7 =-P20
N4 P7 = F16 K10 —> P7 = -10

F17-F28 Special functions

They do not affect the jump flags.

F17

N4 P1 = F17 P2

P1 takes the value of the memory address in which the P2 block is located

Example N4 P1 = F17 K12

P1 takes the value of the memory address in which the block N12 is located.

F18

N4 P1 = F18 P2

P1 takes the value of the X coordinate value in the block located at P2.

F18 does not accept a constant as operand.

Example: P1 = F18 K2 is not valid.

F19

N4 P1 = F19 P2

P1 takes the value of the Y coordinate value in the block located at P2.

F19 does not accept a constant as operand.

Example: P1 = F19 K3 is not valid.

F20

N4 P1 = F20 P2

P1 takes the value of the Z coordinate in the block located at P2.

F20 does not accept a constant as operand.

Example: P1 = F20 K5 is not valid.

F21

N4 P1 = F21 P2

P1 takes the value of the W coordinate in the block located at P2.

F21 does not accept a constant as operand.

Example: P1 = F21 K6 is not valid.

F22

N4 P1 = F22 P2

P1 takes the value of the memory address in the block previous to the one defined by P2.

F22 does not accept a constant as operand.

Example: P1 = F22 K4. is not valid.

F23

N4 P1 = F23

P1 takes the value of the tool table number being used at this moment.

F24

This function can be programmed in two different ways:

Example a) N4 P9 = F24 K2

Parameter P9 takes the R value of the tool table in the position 2.

Example b) N4 P8 = F24 P12

Parameter P8 takes the R value of the tool table in the position indicated by parameter P12.

F25

This function can be programmed in two different ways:

Example a) N4 P15 = F25 K16

Parameter P15 takes the L value of the tool table in the position 16.

Example b) N4 P13 = F25 P34

Parameter P13 takes the L value of the tool table in the position indicated by parameter P34.

F26

This function can be programmed in two different ways:

Example a) N4 P17 = F26 K10

Parameter P17 takes the I value of the tool table in the position 10.

Example b) N4 P19 = F26 P63

Parameter P19 takes the I value of the tool table in the position indicated by parameter P63.

F27

This function can be programmed in two different ways:

Example a) N4 P15 = F27 K27

Parameter P15 takes the K value of the tool table in the position 27.

Example b) N4 P13 = F27 P25

Parameter P13 takes the K value of the tool table in the position indicated by parameter P25.

F28

N4P1 = F28 P2

P1 takes the value of coordinate V in the block with direction P2.

F28 does not accept constant operand.

Example: P1 = F28 K6. Invalid.

Any number of assignments and operations can be programmed in a block provided, however, that no more than 10 parameters are modified.

F29

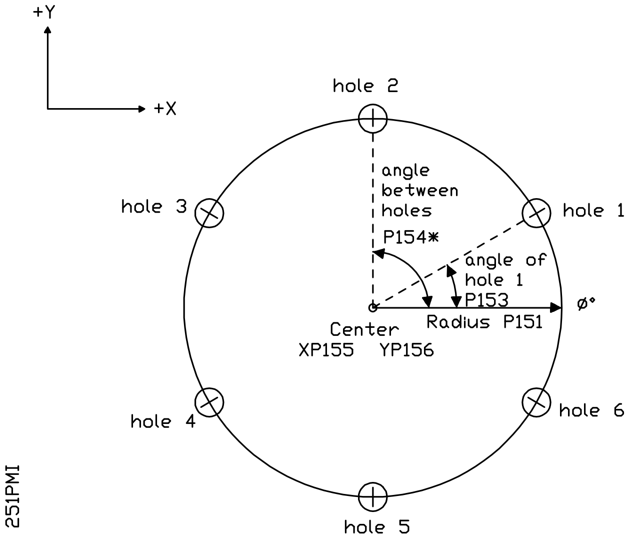
N4 P27 = F29

Parameter P27 takes the value of the selected tool number.

Any number of assignments and operations can be programmed in a block provided, however, that no more than 10 parameters are modified.

BOLT HOLE CIRCLE EXAMPLE:

Six holes equally spaced, 2.5 inch radius, first hole 30° from zero.



P151 = RADIUS
P152 = NUMBER OF HOLES
P153 = ANGLE OF FIRST HOLE
P154 = ± ANGLE BETWEEN HOLES
P155 = X CENTER COORDINATE
P156 = Y CENTER COORDINATE

* If P154 = K0, the holes will be equally spaced around 360°. A positive angle for P154 moves counter-clockwise around the circle. A negative angle moves clockwise.

PARAMETRIC SUBROUTINE N97 - BOLT HOLE CIRCLE

```
N10  G23 N97
N20  P172 = K0
N30  P154 = F11K0
N40  G27N60
N50  P154 = K360 F4 P152
N60  P170 = F8 P153 P170 = P170 F3 P151 P170 = P170 F1 P155
N70  P171 = F7 P153 P171 = P171 F3 P151 P171 = P171 F1 P156
N80  G90 G0 XP170 YP171
N90  P172 = P172 F1 K1 P153 = P153 F1 P154
N100 P172 =F11 P152
N110 G26 N130
N120 G25 N60
N130 G24
```

EXAMPLE PROGRAM USING BOLT-HOLE CIRCLE SUBROUTINE:

```
P9997

N0    (BOLT HOLE CIRCLE EXAMPLE)
N10   G90 G0 X6 Y0 Z0 (SAFE POSITION-OPTIONAL)
N20   G81 G98 G91 Z-0.5 I-0.5 I-3 F10 S500 N0 (DRILLING CANNED CYCLE)
N30   G21 N97 P151=K2.5 P152=-K6 P153=K30 P154 =K0
      P155=K3 P156=K0 (CALL SUBROUTINE)
N40   G80 (CANCEL CANNED CYCLE)
N50   G90 G0 X0 Y0 Z0 (SAFE POSITION-OPTIONAL)
N60   G81 G98 G91 Z-0.5 I-3 F10 S500 N0
N70   G21 N97 P155=K-3 (NEW X CENTER COORDINATE)
N80   G80 (CANCEL CANNED CYCLE)
N90   G90 G0 X0 Y0 Z0
N100  M30
```

NOTES:

. When calling a drilling canned cycle (lines 20 and 60) N0 (repeat zero times) must be specified.

. It is possible to move directly from the end of one circle to the beginning of the next by omitting lines 40 thru 60. If direct movement by the shortest path is not possible due to the interference with the part or fixture, then the above procedure to cancel the drilling canned cycle, position to a safe zone, and re-calling the canned cycle must be used.

. The first time parametric subroutine N97 is called in the program, all six parameters must be specified.
Thereafter, only the parameters which are different from subsequent bolt hole circle patterns need be specified.
However, safe programming practice dictates including all six parameters each time the routine is called.

BINARY OPERATIONS

F30 — AND
F31 — OR
F32 — XOR
F33 — NOT

These BINARY operations, also activate the internal indicators (FLAGS) depending on the value of their result, to use later in the programming of CONDITIONAL JUMPS,CALLS (G26,G27,G28,G29). The binary operations can be made between:

- Parameters : P1 = P2F30P3
- Parameters and constants : P11=P25F31H(8)
- Constants : P19=K2F32K5

The value of constant H must be given in hexadecimal code, integer, positive and with 8 characters maximum, i.e., from 0 to FFFFFFFF and cannot form part of the first operand.

F30 - AND
Example: N4 P1 = P2 F30 P3

Value of P2	Value P3	Value of P1
A5C631F	C883D	C001D

F31 - OR
Example: N4 P11 = P25 F31 H35AF9D01

Value of P25	Value of H	Value of P11
48BE6	35AF9D01	35AF9FE7

F32 - XOR
Example: N4 P19 = P72 F32 H91C6EF

Value of P72	Value of H	Value of P19
AB456	91C6EF	9B72B9

F33 - NOT
Example: N4 P154 = F33 P88
P154 takes the value of P88 complementary to 1.

Value of P88	Value of P154
4A52D63F	B5AD29C0

Jumps/calls within a program

Functions G25,G26,G27,G28 and G29 can be used to jump to another block of the current program.

No more information can be programmed into the same block in which some of the functions G25,G26,G27,G28 or G29 are programmed.

There are two formats:

Format a) JUMP

N4 (G25,G26,G27,G28,G29) N4

N4 : Block number

G25,G26,G27,G28,G29 : Codes for different jumps

N4 : Number of the block the jump is aimed at

When the CNC reads this block it jumps to the targeted block and the program continues.

Example:

```
N0   G00  X100
N5  Y50
N10 G25  N50
N15 X50
N20 Y70
N50 G01  X20
```

When the block 10 is reached, the CNC jumps to block 50 and then the program continues until it is finished.

Format b) CALL:

N4 (G25,G26,G27,G28,G29) N4.4.2.

N4 : Block number
G25,G26,G27,G28,G29 : Codes for different jumps

```
N4.4.2 —> Number of repetitions
 |   └———>Number of the last block to be executed
 └————————> Number of the block the jump is aimed at
```

When the CNC reads such a block it jumps to the block identified between the N and the first decimal point. It then executes the section of the program between this block and the one identified between the two decimal points as many times as set by the last number. This last number can take a value within 0 and 99, unless it is programmed using a parameter in which case the limits are 0 and 255.

If only N4.4 is written the CNC will assume N4.4.1.

When the execution of this section is finished the CNC goes to the block next to the one in which G25 N4.4.2. was programmed.

Example:

```
N0   G00  X10
N5   Y20
N10 G01  X50  M3
N15 G00  Y0
N20 X0
N25 G25 N0.20.8
N30 M30
```

When block 25 is reached the CNC will jump to block 0 and will execute 8 times the section N0-N20. On completion of this, it will go to block 30.

G25 Unconditional jump/call

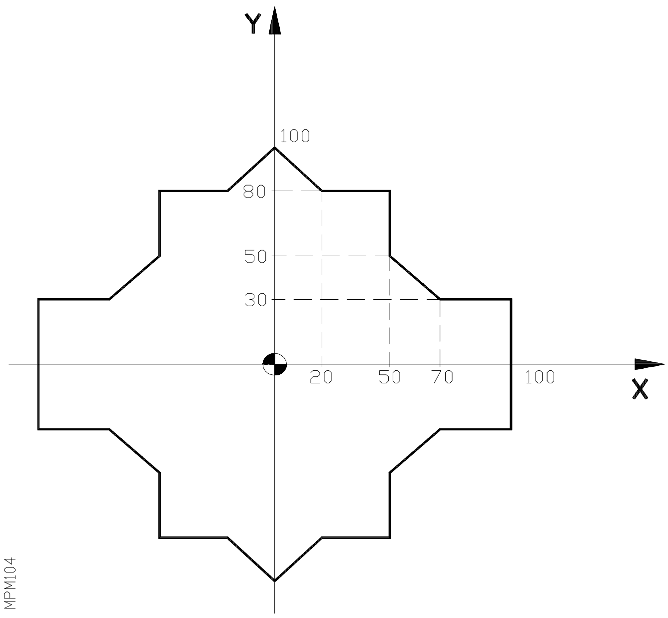
As soon as the CNC reads code G25, it jumps to the block identified by N4 or N4.4.2.

Programming

N4 G25 N4 or N4 G25 N4.4.2

G25 must stand alone in a block.

Example:



Starting point X100 Y0

```
N10   G90  G01  Y30  F500
N20   X70
N30   X50 Y50
N40   Y80
N50   X20
N60   X0 Y100
N70   X-20 Y80
N80   X-50
N90   Y50
N100 X-70 Y30
N110 X-100
N120 Y0
N130 G11  G12
N140 G25  N10.120.1
N150 M30
```

Two flags can be activated according to the result of the following operations:

F1,F2,F3,F4,F5,F6,F7,F8,F9,F10,F11,F12,F13,F14,F15,F16,F30,F31,F32,F33.

The assignments do not affect the state of these flags.

Flag 1 (zero, equal)

If the result of an operation is zero, flag 1 is activated.

If the result of an operation is not zero, flag 1 is not activated.

If the result of a comparison is equal, flag 1 is activated.

If the result of a comparison is different, flag 1 is not activated.

Flag 2 (negative, smaller)

If the result of an operation is smaller than zero, flag 2 is activated.

If the result of an operation is greater than or equal to zero, flag 2 is not activated.

If, in a comparison, the first operand is smaller than the second, flag 2 is activated.

If, in a comparison, the first operand is greater than or equal to the second flag 2 is not activated.

The conditions for the program to jump to the targeted block, after reading G26,G27,G28 or G29 are:

With G26 the program will jump if flag 1 is activated.
With G27 the program will jump if flag 1 is not activated.
With G28 the program will jump if flag 2 is activated.
With G29 the program will jump if flag 2 is not activated.

G26 Conditional jump/call if = 0

When the CNC reads a block with the code G26, if the condition = 0 is met, it jumps to the block indicated by N4 or N4.4.2; if the condition = 0 is not met the CNC will disregard this block.

Programming: N4 G26 N4 or N4 G26 N4.4.2

G26 must stand alone in a block.

Examples:

a)
```
N0  G00  X10
N5  P2 = K3
N10 P1 = P2 F1 K5
N15 G01  Z5
N20 G26  N50
N25
“
“
“
N50 G1  Z10
```

The last operation with parameters being P1 = P2+K5=3+5=8 (result = 0) the = 0 flag will not be activated and the CNC will disregard block N20.

b)
```
N0  G00  X10
N5  P2 = K3
N10 P1 = P2 F1 K5
N15 G01  Z5
N20 P3 = K7
N25 P4 = P3 F2 K7
N30 G26  N50
“
“
“
N50 M30
```

The last operation with parameters being P4 = P3+K2 K7= 7-7=0, the =0 flag will be activated and the CNC will jump to block 50 when reading block 30.

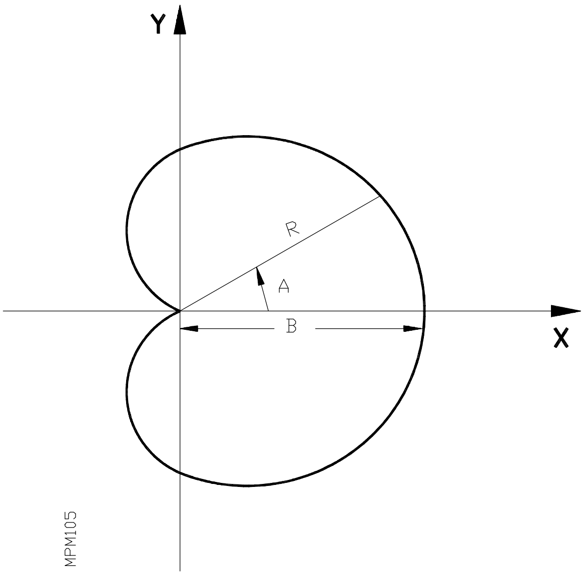
G27 Conditional jump/call if ≠ 0

When the CNC reads a block with G27, if the condition ≠ 0 is met, it jumps to the block identified by N4 or N4.4.2, if the condition ≠ 0 is not met the CNC will disregard this block.

Programming N4 G27 N4 or N4 G27 N4.4.2

G27 must stand alone in a block.

Example:



For example, to program a cardioid R = B |cos A/2 |
being A=P0 (Angle) and
B=P1 (value 30 mm)

Starting point X0 Y0:

```
N10 G93  G01 F500
N20 P0=K0
N30 P1=K30 P2=P0 F4 K2 P3=F8 P2 P4=F15 P3 P5=P1 F3 P4
N40 G01  G05  R P5  A P0 ........................................... (Movement block)
N50 P0=P0 F1 K5 ..........................................................  (5 degrees added to the angle)
N60 P0=F11 K365 .........................................................  (If not equal to 365 degrees)
N70 G27 N30 ..................................................................  (It jumps to block N30)
N80 X0   Y0
N90 M30
```

G28 Conditional jump/call if smaller

When the CNC reads a block with the code G28, if the condition < is met, it jumps to the block identified by N4 or N4.4.2. If the condition < is not met, the CNC will disregard this block.

Programming: N4 G28 N4 or N4 G28 N4.4.2

G28 must stand alone in a block.

G29 Conditional jump/call if equal or greater

When the CNC reads a block with G29, if the condition => is met, it jumps to the block identified by N4 or N4.4.2. If the condition => is not met, the CNC will disregard this block.

Programming: N4 G29 N4 or N4 G29 N4.4.2

G29 must stand alone in a block.

G30 Display of error code defined by K

When the CNC reads a block with G30, it stops the program and displays the contents of this block.

Programming: N4 G30 K2

N4 : Block number
G30 : Code identifying programming of an error
K2(0-99) : Programmed error code

Any code between 0 and 99 can be programmed unless the error code K is defined by a parameter (N4 G30 K P2) because then the possible values are 0-255.

This code, combined with G26,G27,G28,G29 enables the stopping of the program and the detection of possible measuring error, for instance.

G30 must stand alone in a block.

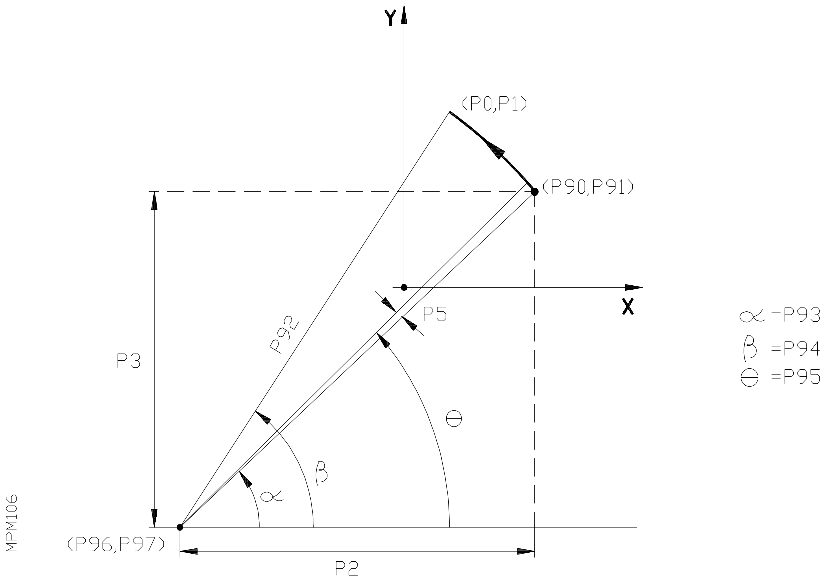
Atention:

If it is not required for the CNC error code comment to be displayed, the number of the code after G30 must be greater than those used by the CNC.

Remember also that the user can write comments in the program which will be displayed when the corresponding block is executed.

EXAMPLE OF PROGRAM OF AN ARC WHOSE RADIUS IS GREATER THAN 8388.607 mm

If starting point is X3000 Y2000 and the following arc is programmed: G03 X1000 Y3774.964 I-8000 J-7000 The CNC will generate error 33 because the radius is greater than 8388 mm. Parametric programming can be used to overcome this limitation.



MEANING OF PARAMETERS

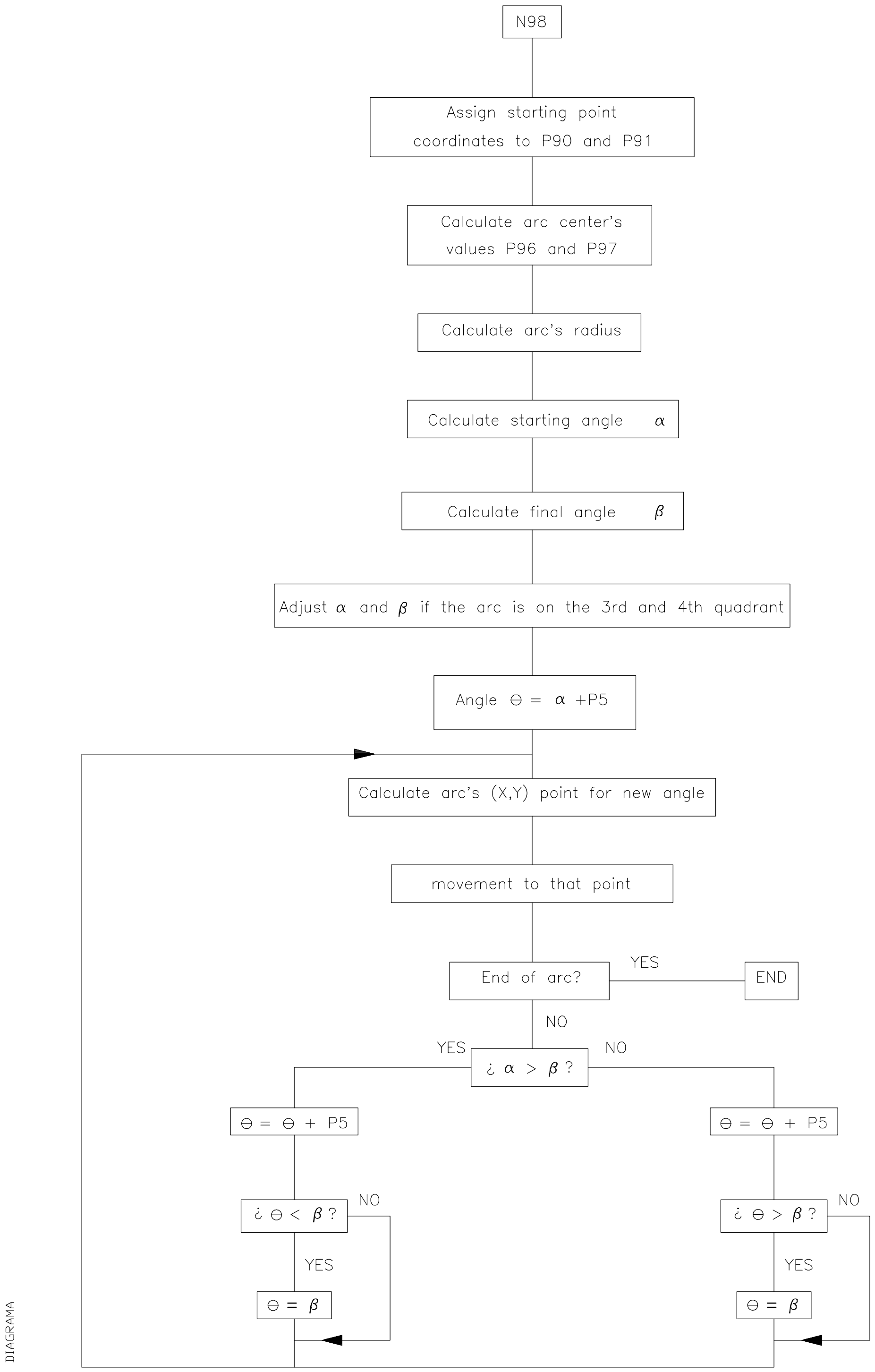
Call parameters

P0: X value of the final point
P1: Y value of the final point
P2: Distance from the starting point to the center, along X axis
P3: Distance from the starting point to the center, along Y axis
P4: Feedrate
P5: Increment of the angle with its sign (Clockwise = negative, counterclockwise = positive).

Parameters used in the subroutine

P90: X value of the starting point
P91: Y value of the starting point
P92: Radius
P93: Initial angle α
P94: Final angle β
P95: Working or movement angle θ
P96: Arc's center's X value
P97: Arc's center's Y value
P98: Calculations
P99: Calculations

Subroutines flow chart:



SUBROUTINE N98

```
N00 G23 N98
N01 P90=X P91=Y ...............................................(Takes point values)
        P96=P90 F1 P2  P97=P91 F1 P3...........................(Calculates center)
         P92=P2 F6 P3 .........................................(Calculates radius)
        P98=P3 F4 P2  P93=F10 P98 .............................(Calculates angle α)
        P98=P90 F2 P96  P98=F11 K0
N02 G29 N4
N03 P93=P93 F1 K180
N04 P98=P0 F2 P96  P99=P1 F2 P97 ........................ (Calculates angle β)
N05 P94=P99 F4 P98 P94=F10 P94 P98=F11 K0
N06 G29 N8
N07 P94=P94 F1 K180
N08 P5=F11 K0
N09 G29 N16
N10 P93=F11 K0
N11 G29 N2 ....................................................(Adjusts values of α
N12 P94=F11 K0                                                 and β if the
N13 G28 N21                                                    arc spares 3rd and 4th
N14 P93=P93 F1 K360                                            quadrants)
N15 G25 N21
N16 P94=F11 K0
N17 G29 N21
N18 P93=F11 K0
N19 G28 N2
N20 P94=P94 F1 K360
N21 P95=P93 F1 P5..............................................(angle θ = α + P5)
N22 P98=F8 P95 P98=P98 F3 P92 P98=P98 F1 P96 .....X value of point)
       P99=F7 P95 P99=P99 F3 P92 P99=P99 F1 P97 .....(Y value of point)
N23 G1 XP98 YP99 FP4...........................................(Go to point)
N24 P95=F11 P94 ...............................................(End of arc?)
N25 G26 N37
N26 P94=F11 P93 ...............................................(Compare α and β )
N27 G26 N37 ...................................................(If α = β end )
N28 G28 N33
N29 P95=P95 F1 P5  P95=F11 P94.................................(If β > α increment of θ
        .......................................................and check whether = β )
N30 G28 N32
N31 P95=P94....................................................(If θ = β has been reached or
                                                               surpassed)
N32 G25 N22 ...................................................(calculates new point)
N33 P95=P95 F1 P5  P94=F11 P95.................................(If α > β decrement θ of and
        .......................................................check whether = β )
N34 G28 N36
N35 P95=P94....................................................(If θ = β has been reached or
                                                               surpassed)
N36 G25 N22 ...................................................(calculates new point)
N37 G24
```

This subroutine can be used to perform any arc with radius greater than 8388.607 mm both clockwise and counterclockwise.

The program to execute the arc previously defined will be:

N10 P0=K1000 P1=K3774.964 P2=K-8000 P3=K-7000 P4=K100 P5=K0.5
N20 G1 G41 X3000 Y2000 T1.1
N30 G21 N98.01

Atention:

If tool offset is to be used the following programming order must be strictly followed.

1. Definition of call parameter.
2. Positioning in the arc's initial point.
3. Calling the subroutine.

ERROR CODES

001 This error occurs in the following cases:

> When the first character of the block to be executed is not an "N".

> When while BACKGROUND editing, the program in execution calls a subroutine located in the program being edited or in a later program.

The order in which the part-programs are stored in memory are shown in the part-program directory. If during the execution of a program, a new one is edited, this new one will be placed at the end of the list.

002 Too many digits when defining a function in general.

003 A negative value has been assigned to a function which does not accept the (-) sign or an incorrect value has been given to a canned cycle parameter.

004 A canned cycle has been defined while function G02, G03 or G33 was active.

005 Parametric block programmed wrong.

006 There are more than 10 parameters affected in a block.

007 Division by zero.

008 Square root of a negative number.

009 Parameter value too large

010 M41, M42, M43 or M44 has been programmed.

011 More than 7 "M" functions in a block.

012 This error occurs in the following cases:

> Function G50 is programmed wrong

> Tool dimension values too large.

> Zero offset values (G53/G59) too large.

013 Cycle defined incorrectly.

014 A block has been programmed which is incorrect either by itself or in relation with the program history up to that instant.

015 Functions G20, G21, G22, G23, G24, G25, G26, G27, G28, G29, G30, G31, G32, G50, G52, G53, G54, G55, G56, G57, G58, G59, G72, G73, G74, G92 and G93 must be programmed alone in a block.

016 The called subroutine or block does not exist or the block searched by means of special function F17 does not exist.

017 This error is issued in the following cases:

> Negative or too large thread pitch value.

> Function G95 or M19 has been used with machine parameter "P800=0".

018 Error in blocks where the points are defined by means of angle-angle or angle-coordinate.

019 This error is issued in the following cases:

> After defining G20, G21, G22 or G23, the number of the subroutine it refers to is missing.

> The "N" character has not been programmed after function G25, G26, G27, G28 or G29.

> Too many nesting levels.

020 The axes of the circular interpolation are not programmed correctly.

021 There is no block at the address defined by the parameter assigned to F18, F19, F20, F21, F22.

022 An axis is repeated when programming G74.

023 K has not been programmed after G04.

024 The decimal point is missing when programming T2.2 or N2.2.

025 Error in a definition block or subroutine call, or when defining either conditional or unconditional jumps.

026 This error is issued in the following cases:

> Memory overflow.

> Not enough free tape or CNC memory to store the part-program.

027 I/J/K has not been defined for a circular interpolation or thread.

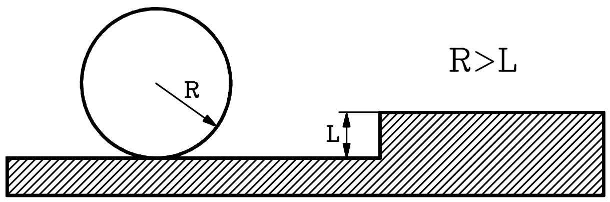
028 An attempt has been made to select a tool offset at the tool table or a non-existent external tool (the number of tools is set by machine parameter).

029 Too large a value assigned to a function.

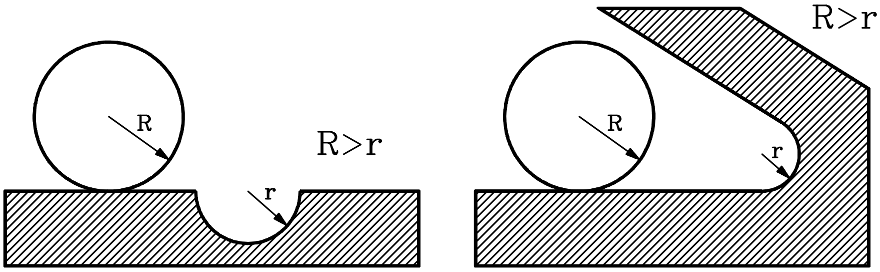
This error is often issued when programming an F value in mm/min (inch/min) and, then, switching to work in mm/rev (inch/rev) without changing the F value.

030 The programmed G function does not exist.

031 Tool radius value too large.



032 Tool radius value too large.



033 A movement of over 8388 mm or 330.26 inches has been programmed.

Example: Being the X axis position X-5000, if we want to move it to point X5000, the CNC will issue error 33 when programming the block N10 X5000 since the programmed move will be: 5000 - (-5000) = 10000 mm.

In order to make this move without issuing this error, it must be carried out in two stages as indicated below:

```
N10 X0        ; 5000 mm move
N10 X5000     ; 5000 mm move
```

034 S or F value too large.

035 Not enough information for corner rounding, chamfering or compensation.

036 Repeated subroutine.

037 Function M19 programmed incorrectly.

038 Function G72 or G73 programmed incorrectly.

It must be borne in mind that if G72 is applied only to one axis, this axis must be positioned at part zero (0 value) at the time the scaling factor is applied.

039 This error occurs in the following cases:

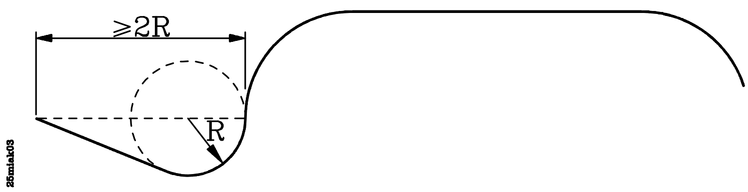
> More than 15 nesting levels when calling subroutines.

> A block has been programmed which contains a jump to itself. Example: N120 G25 N120.

040 The programmed arc does not go through the defined end point (tolerance 0.01mm) or there is no arc that goes through the points defined by G08 or G09.

041 This error is issued when programming a tangential entry as in the following cases:

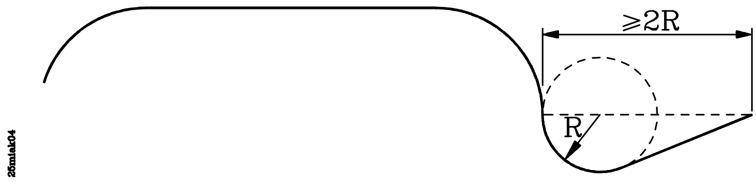
> There is no room to perform the tangential entry. A clearance of twice the rounding radius or greater is required.



> If the tangential entry is to be applied to an arc (G02, G03), The tangential entry must be defined in a linear block.

042 This error is issued when programming a tangential exit as in the following cases:

> There is no room to perform the tangential exit. A clearance of twice the rounding radius or greater is required.



> If the tangential exit is to be applied to an arc (G02, G03), The tangential exit must be defined in a linear block.

043 Polar origin coordinates (G93) defined incorrectly.

044 Canned cycle defined wrong.

045 Function G36, G37, G38 or G39 programmed incorrectly.

046 Polar coordinates defined incorrectly.

047 A zero movement has been programmed during radius compensation or corner rounding.

048 W axis programmed wrong.

049 Chamfer programmed incorrectly.

050 Functions M06, M22, M23, M24, M25 must be programmed alone in a block.

051 * A tool or pallet change cannot be performed without being in the change position.

052 * The requested tool is not in the magazine.

053 * This error occurs when having a machining center and 2 different external Ts have been programmed in a row without programming an M06 in between.

054 There is no disk in the FAGOR Floppy Disk Unit, there is no tape in the cassette reader or the reader head cover is open.

055 Parity error when reading or writing a floppy or cassette.

056 This error comes up in the following cases:

> When the memory is locked and an attempt is made to generate a CNC program by means of function G76.

> When trying to generate program P99999 or a protected program by means of function G76.

> If function G76 is followed by function G22 or G23.

> If there are more than 70 characters after G76.

> If function G76 (block content) has been programmed without having programmed G76 P5 or G76 N5 before.

> If in a G76 P5 or G76 N5 type function does not contain the 5 digits of the program number.

> If while a program is being generated (G76 P5 or G76 N5), its program number is changed without cancelling the previous one.

> If while executing a G76 P5 type block, the program referred to is not the one edited. In other words, that another one has been edited later or that a G76 P5 type block is executed while a program is being edited in background.

057 Write-protected floppy disk or cassette.

058 Irregular floppy drive motion or sluggish tape transport.

059 Communication error between the CNC and the FAGOR Floppy Disk Unit or between the CNC and the cassette reader.

060 Internal CNC hardware error. Consult with the Technical Service Department.

061 Battery error.

The memory contents will be kept for 10 more days (with the CNC off) from the moment this error occurs. The whole battery module located on the back must be replaced. Consult with the Technical Service Department.

Due to danger of explosion or combustion: do not try to recharge the battery, do not expose it to temperatures higher than 100°C (232°F) and do not short the battery leads.

064 * External emergency input (pin 14 of connector I/O1) is activated.

065 * This error comes up in the following cases:

> If while probing (G75) the programmed position is reached without receiving the probe signal.

> If while executing a probing canned cycle, the CNC receives the probe signal without actually carrying out the probing move itself (collision).

066 * X axis travel limit overrun.

It is generated either because the machine is beyond limit or because a block has been programmed which would force the machine to go beyond limits.

067 * Y axis travel limit overrun.

It is generated either because the machine is beyond limit or because a block has been programmed which would force the machine to go beyond limits.

068 * Z axis travel limit overrun.

It is generated either because the machine is beyond limit or because a block has been programmed which would force the machine to go beyond limits.

069 * W axis travel limit overrun.

It is generated either because the machine is beyond limit or because a block has been programmed which would force the machine to go beyond limits.

070 ** X axis following error.

071 ** Y axis following error.

072 ** Z axis following error.

073 ** W axis following error.

074 ** Spindle speed value too large.

075 ** X axis feedback error. Connector A1.

076 ** Y axis feedback error. Connector A2.

077 ** Z axis feedback error. Connector A3.

078 ** W axis feedback error. Connector A4.

079 ** Spindle feedback error. Connector A5.

080 ** Handwheel feedback error. Connector A5.

081 ** V axis feedback error. Connector **A**5.

082 ** Parity error in general parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

083 ** Parity error in V axis parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

084 * V axis travel limit overrun.

085 ** V axis following error.

086 Not being used at this time.

087 ** Internal CNC hardware error. Consult with the Technical Service Department.

088 ** Internal CNC hardware error. Consult with the Technical Service Department.

089 * All the axes have not been homed.

This error comes up when it is mandatory to search home on all axes after power-up. This requirement is set by machine parameter.

090 ** Internal CNC hardware error. Consult with the Technical Service Department.

091 ** Internal CNC hardware error. Consult with the Technical Service Department.

092 ** Internal CNC hardware error. Consult with the Technical Service Department.

093 ** Internal CNC hardware error. Consult with the Technical Service Department.

094 Parity error in tool table or zero offset table G53-G59. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

095 ** Parity error in W axis parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

096 ** Parity error in Z axis parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

097 ** Parity error in Y axis parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

098 ** Parity error in X axis parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

099 ** Parity error in M table. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

100 ** Internal CNC hardware error. Consult with the Technical Service Department.

101 ** Internal CNC hardware error. Consult with the Technical Service Department.

105 This error comes up in the following cases:
> A comment has more than 43 characters.
> A program has been defined with more than 5 characters.
> A block number has more than 4 characters.
> Strange characters in memory.

106 ** Inside temperature limit exceeded.

107 ** Error in W axis leadscrew error compensation parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

108 ** Error in Z axis leadscrew error compensation parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

109 ** Error in Y axis leadscrew error compensation parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

110 ** Error in X axis leadscrew error compensation parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

111 * FAGOR LAN line error. Hardware installed incorrectly.

112 * FAGOR LAN error. It comes up in the following instances:

> When the configuration of the LAN nodes is incorrect.
> The LAN configuration has been changed. One of the nodes is no longer present (active).

When this error occurs, access the LAN mode, editing or monitoring, before executing a program block.

113 * FAGOR LAN error. A node is not ready to work in the LAN. For example:

> The PLC64 program is not compiled.
>A G52 type block has been sent to an 82CNC while it was in execution.

114 * FAGOR LAN error. An incorrect command has been sent out to a node.

115 * Watch-dog error in the periodic module.

This error occurs when the periodic module takes longer than 5 milliseconds.

116 * Watch-dog error in the main module.

This error occurs when the main module takes longer than half the time indicated in machine parameter "P741".

117 * The internal CNC information requested by activating marks M1901 thru M1949 is not available.

118 * An attempt has been made to modify an unavailable internal CNC variable by means of marks M1950 thru M1964.

119 Error when writing machine parameters, the decoded M function table and the leadscrew error compensation tables into the EEPROM memory.

This error may occur when after locking the machine parameters, the decoded M function table and the leadscrew error compensation tables, one tries to save this information into the EEPROM memory.

120 Checksum error when recovering (restoring) the machine parameters, the decoded M function table and leadscrew error compensation tables from the EEPROM memory.

150 Incoherent data in the 512 Kb memory.

When this error occurs, save as many programs as you can into the Floppy Disk Unit, peripheral or PC.

Then, proceed as follows to format the 512 Kb memory (when doing this, all part-programs stored in this memory will be lost).

Press **[OP MODE] [6]** to select the Editing mode.
Press **[LOCK/UNLOCK]** the screen displays the text: CODE:
Key in: **FM512** and press **[ENTER]**

Once the 512 Kb memory is formatted, recover (restore) the programs you saved into the Floppy Disk Unit, peripheral or PC.

151 Defective 512 Kb memory. Consult with the Technical Service department.

152 Not enough available free space in the 512 Kb memory.

Atention:

The **ERRORS** indicated with "*" behave as follows:

They stop the axis feed and the spindle rotation by cancelling the Enable signals and the analog outputs of the CNC.

They interrupt the execution of the part-program of the CNC if it was being executed.

The **ERRORS** indicated with "**" besides behaving as those with an "*", they activate the INTERNAL EMERGENCY OUTPUT.

FAGOR 8025/8030 CNC

APPLICATIONS MANUAL

Ref. 9701 (in)

ABOUT THE INFORMATION IN THIS MANUAL

This manual describes applications which, while not being specific of milling machines, are also possible with this CNC.

This manual must be read together with the rest of the manuals for this CNC.

Notes: 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 this manual without prior notice.

INDEX

Section | Page

Chapter 1 LASER MACHINES

1.1 Machine parameters 1
1.2 LASER beam proportional to axis feedrate 3
1.3 Sheetmetal tracing 5

Chapter 2 JIG GRINDERS

2.1 Machine parameters 1
2.2 "C" axis perpendicular to XY path 2

Chapter 3 NON-SERVO CONTROLLED OPEN LOOP MOTORS

3.1 Introduction 1
3.2 Machine parameters 4
3.3 Basic operation 5
3.4 Movement execution 6
3.5 Automatic and Single-Block modes 7
3.5.1 Using functions G05 and G07 7
3.5.2 Single Block execution 8
3.5.3 Manual Feedrate Override Switch 8
3.5.4 Cycle Stop and Feedhold signals 8
3.6 JOG mode 9
3.6.1 % Feed area (continuous jog) 9
3.6.2 Incremental JOG 9
3.6.3 Handwheel area 10
3.7 Home search 11

1. LASER MACHINES

1.1 MACHINE PARAMETERS

P619(3) Analog S output proportional to actual axis feedrate.

By setting this parameter, it is possible to control the intensity of the laser BEAM. The CNC outputs an analog S voltage proportional to the actual feedrate of the axes.

0 = This feature is **not** active.
1 = This feature **is** active.

P622(6) Sheetmetal tracing on laser machines

It indicates whether this feature is active or not.

0 = This feature is **not** active.
1 = This feature **is** active.

When using this feature, functions M97 and M98 acquire a special meaning as described in the section on "Laser Machines" in the chapter on "Concepts" of this manual.

P806 Distance between the beam and the sheetmetal

In order to trace the profile of the sheetmetal, a device is used attached to the axis of the laser beam indicating, at all times, the deviations of the sheetmetal surface.

To do this, this device has a cylinder moving in and out of it and whose tip is constantly touching the sheetmetal surface.

This parameter indicates the distance this cylinder must penetrate into the device once its tip touches the sheetmetal. This way, it sets the distance between the beam and the sheetmetal surface.

This distance is given in microns, regardless of the work-units being used.

Possible values: 0 thru 32000 microns.

If set to "0", this feature will not be activated.

P807 Maximum sheetmetal deflection

This parameter is used when the sheetmetal tracing feature is active. It prevents abrupt laser movements when detecting holes, objects, etc. while machining the sheetmetal.

It indicates the maximum sheetmetal deflection value. It is always expressed in microns regardless of the work-units being used.

Possible values: 0 thru 32000 microns.

If set to "0", the "sheetmetal tracing" feature will not be activated.

If the value assigned to this parameter is exceeded, the CNC will deactivate the "sheetmetal tracing" feature and depending, on the operating mode, it will act as follows:

* In the JOG mode, the CNC will display the following error of the Z axis.

* In the rest of the operating modes, the CNC will generate an external CYCLE STOP executing the emergency subroutine if it has been programmed.

P808 Analog corresponding to maximum Z axis feedrate

It sets the analog voltage corresponding to the maximum Z axis feedrate when executing special functions: **M97 or M98 (sheetmetal tracing)**.

It is always defined in microns regardless of the work units being used.

Possible values: 0 to 32000 microns.

If set to "0", the maximum analog voltage value for the Z axis will be:

$$\text{Analog (mV.)} = \mathbf{P806} \times \text{K1} \times \frac{2.5\text{mV.}}{64}$$

If set to a value other than "0", the maximum analog voltage for the Z axis will be:

$$\text{Analog (mV.)} = \mathbf{P808} \times \text{K1} \times \frac{2.5\text{mV.}}{64}$$

The proportional gain **K1** of the Z axis is determined by machine parameter **P314**.

Example: Machine parameters: “P314”= 32 and “P806”= 2000.

The Z axis is controlled by a 10V-system providing 3000 mm/min and the Z axis feedrate is to be limited to 300 mm/min.

Therefore, the Z axis analog voltage must be limited to 1V (10% of 10V). This will give us the following value:

$$1000 \text{ mV.} = \mathbf{P808} \times 32 \times \frac{2.5\text{mV.}}{64} \quad ===> \quad \mathbf{P808 = 800}$$

1.2 LASER BEAM PROPORTIONAL TO AXIS FEEDRATE

In order to work with this feature, machine parameter “P619(3)” must be set to "1".

It is also necessary to adjust the axis servo drives so their maximum desired feedrates (G00) are obtained with ±9.5 V.

When this feature is selected, the CNC will provide an analog "S" voltage (pins 36 and 37 connector I/O1) proportional to the actual (real) feedrate of the axes.

The CNC uses the "S analog" output to control the intensity of the laser beam; therefore, the spindle parameters must be set accordingly.

Spindle machine parameters “P601(3)” and “P601(2)” must be set to "0" for the analog output range to be ±10 V.

To get a unipolar analog voltage, set spindle machine parameter “P610(4)” to "1". The sign of this unipolar analog output will be determined by spindle machine parameter “P601(4)”.

The machine parameters related to spindle speed range (P7, P8, P9, P10) are set in rpm. Therefore, whenever an "S" value is programmed, it will be indicated in rpm and the CNC will provide the corresponding analog voltage depending on the speed range it corresponds to the programmed "S" value.

The programming format used by the CNC for using this feature is as follows:

N4 G1 X±4.3 Y±4.3 F5.5 S4.4 M3/M4

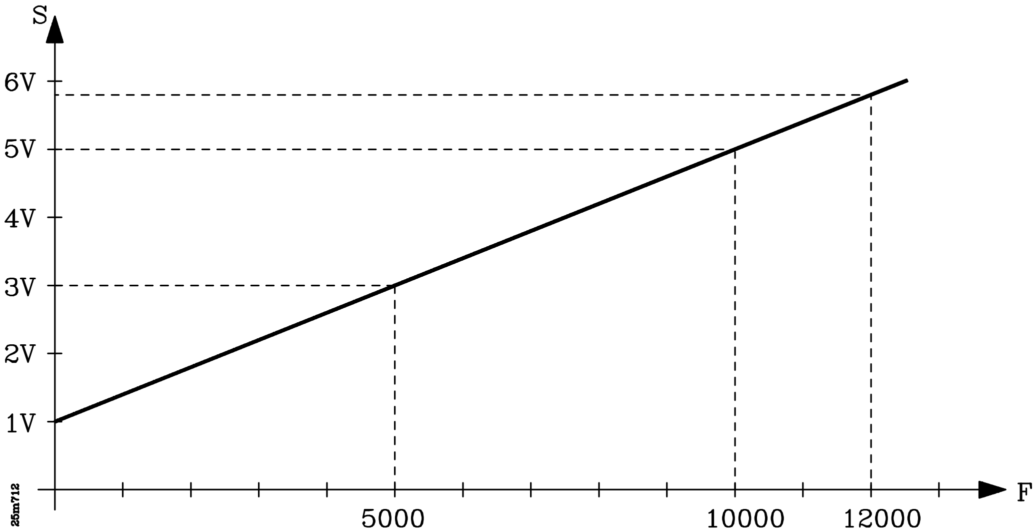
The figure located between the "S" and the "." indicates the minimum analog value (rpm) provided by the CNC and the figure after the "." indicates the analog (rpm) corresponding to the programmed feedrate F5.5.

If there is no axis movement, the CNC will provide the programmed minimum S. Otherwise, the CNC will supply the analog voltage corresponding to the actual (real) axis feedrate.

In rapid positioning moves (G00), the CNC will output the programmed minimum S.

Example:

If we have one single spindle range with 1000 rpm for 10V of analog voltage and we program F10000 S100.500



If there is no movement or it is moving in G00 (rapid positioning), the CNC will output an analog voltage of 1V.

For an actual feedrate of the axes of 12000 mm/min., the CNC will provide 5.8V.

When the actual axis feedrate is 5000 mm/min., the CNC will output 3V.

To temporarily cancel the laser beam, program M05. To recover it, program M03 or M04. To select a new "laser beam"-"feedrate" ratio, program a new block as follows

N4 G1 X±4.3 Y±4.3 F5.5 S4.4 M3/M4

When the axis feedrate for arcs depends on the arc radius, (machine parameter "P729") the corresponding laser analog (S) will also depend on it.

1.3 SHEETMETAL TRACING

When using this feature, it is necessary to set machine parameter P622(6) to "1".

With this feature it is possible to keep the focusing distance of the laser beam constant, thus achieving an optimum machining quality even on very wavy sheetmetal surfaces.

To do this, a sensor must be used attached to the axis of the laser beam. This sensor will provide, at all times, the feedback signals indicating to the CNC the deviations of the actual sheetmetal surface with respect to its theoretical value.

The axis supporting the focus of the laser beam must be set as the "Z" axis and the feedback signals of the sensor must be connected through the "V" axis feedback input. Also, the "V" axis must be set as a DRO axis by means of machine parameter “P617(3)=1”.

The CNC offers miscellaneous (auxiliary) functions M97, M98 and M99 which have the following meaning when working with this feature (sheetmetal tracing):

M97 to activate the "sheetmetal tracing" feature. This function is used when the counting directions of the Z and V axes are the same.

M98 to activate the "sheetmetal tracing" feature. This function is used when the counting directions of the Z and V axes are **not** the same.

M99 to end or cancel the "sheetmetal tracing" feature.

The machine parameters to be set when using this feature are:

P619(3)	Analog S output proportional to actual axis feedrate.
P622(6)	Sheetmetal tracing on laser machines.
P806	Beam focusing distance (between beam and sheetmetal).
P807	Maximum sheetmetal deflection.
P808	Analog voltage corresponding to maximum Z axis feedrate.

Whenever the "sheetmetal tracing" feature is used, the CNC behaves as follows:

1.- When M97 or M98 is executed, the CNC will activate this feature.

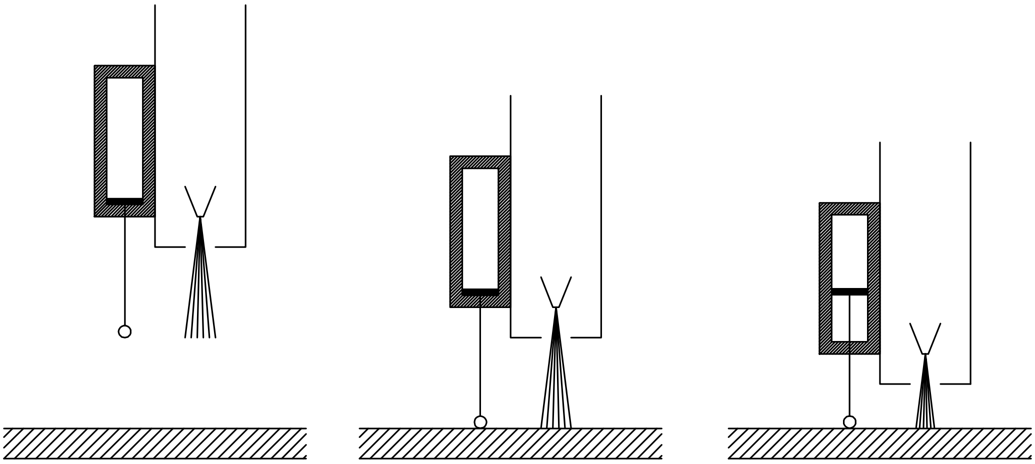
2.- The laser beam (Z axis) will move towards the sheetmetal until the sensor attached to it makes contact with the sheetmetal surface.

The maximum feedrate to be used in this approach move is set by machine parameter "P808".

As a safety measure, the Z axis must be moved before executing M97 or M98. Otherwise, the CNC will issue error 102.

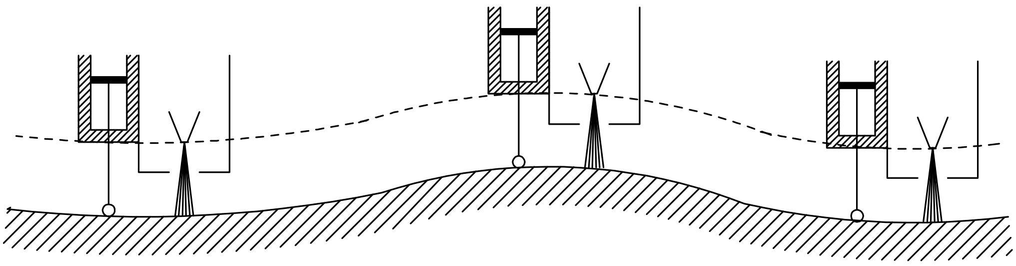
3.- The laser beam will keep approaching the sheetmetal until the sensor indicates that it has penetrated a "P806" distance into the sheetmetal.

This distance will be maintained between the laser beam and the sheetmetal surface during the whole machining process.



4.- From then on, the CNC will start making the programmed cut.

Only the XY movements must be programmed. The Z axis is controlled by the CNC and it will move the distance indicated by the sensor maintaining the same distance between the beam and the sheetmetal surface during the whole machining process.



The Z axis position display will not correspond to its real position since this axis is affected by the variations of the sensor.

In order to avoid abrupt movements of the laser while cutting the sheetmetal (as when detecting holes, objects, etc.), machine parameter "P807" indicates the maximum sheetmetal deflection allowed.

To control the laser beam so the analog S output is proportional to the feedrate of the axes, set machine parameter “P619(3)” to “1”.

2. *JIG GRINDERS*

2.1 MACHINE PARAMETERS

P622(8) JIG GRINDER

It indicates whether the machine is or not a JIG GRINDER.

0 = It is **not** a JIG GRINDER.
1 = It **is** a JIG GRINDER.

When using this feature, the CNC controls the C axis so it always stays perpendicular to the XY path. This is further described in the section on "Jig Grinder" in the chapter on "concepts" of this manual.

2.2 "C" AXIS PERPENDICULAR TO XY PATH

When it is desired to have the C axis perpendicular to the XY path (JIG GRINDER type machines), it is necessary to set machine parameter "P622(8)" to "1".

The axes controlled by the CNC will be defined as:

X,Y Main machine axes. It is possible to interpolate them.

C Rotary axis. When this feature is active, the CNC will maintain this axis perpendicular to the XY path.

U Auxiliary axis that can be interpolated with the X and Y axes.

Also, the following indications must be considered when reading the CNC manuals:

* Any reference to the Z axis must be interpreted as references to the U axis.
* Any reference to the W axis must be interpreted as references to the C axis.
* The following functions are no longer available:

Helical interpolation (with G2/G3)	
G17 G18 G19	Plane selection
G33	Electronic threading
G77 G78	Slaving of the 4th to its associated axis
G81 ... G89	Machining canned cycles
G98 G99	Withdraw in canned cycles

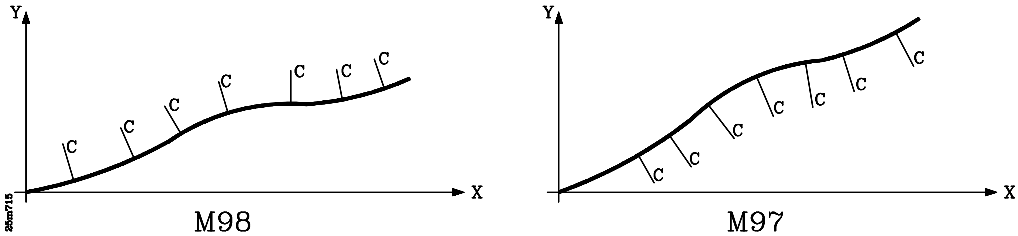
Machine parameters “P600(1)” and “P606(1)” must be set to "1" since the "C" axis must be rotary and "ROLLOVER".

When using this feature, the CNC offers functions M97, M98 and M99 which acquire the following meaning:

M97 Activates this feature keeping the C axis to the right of the programmed XY path.

M98 Activates this feature keeping the C axis to the left of the programmed XY path.

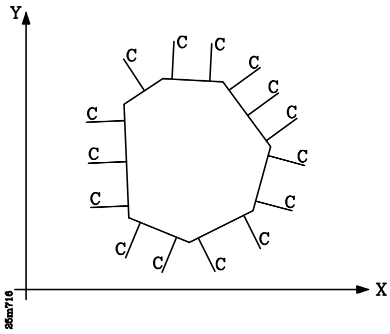
M99 Ends or cancels this feature.



When working with this feature, the CNC acts as follows:

1.- When executing M97 or M98, the CNC will activate this feature.

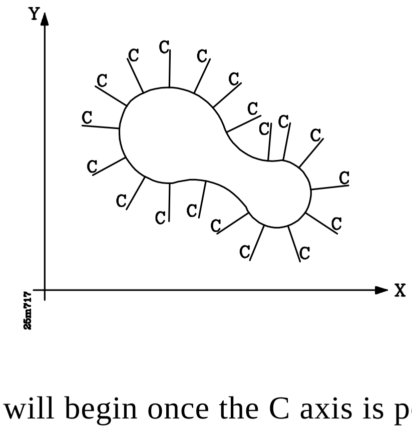
2.- If a linear interpolation has been programmed for the XY axes, the CNC positions the C axis perpendicular to the programmed path and on the desired side.



The XY interpolation will begin once the C axis is positioned perpendicular to the programmed path and the CNC will keep it perpendicular to this path during the whole XY movement.

3.- If a circular interpolation has been programmed for the XY axes, the CNC positions the C axis in a radial position (and on the desired side) with respect to the first point of the arc.

.



The XY interpolation will begin once the C axis is positioned radial to the first point of the arc.

The CNC will keep the C axis radial to the programmed path during the whole interpolation.

4.- Execute function "M99" to cancel this feature.

3. *NON-SERVO CONTROLLED OPEN LOOP MOTORS*

3.1 INTRODUCTION

When the motor does not have a servo drive, it is called: non-servo controlled.

Therefore, Non-servo controlled Open Loop means that the CNC only controls the position of the axis during the programmed movement. Once the axis reaches position, the CNC no longer controls it.

This feature may only be used at the GP model. Up to a maximum of 4 axes may be controlled (X, Y, Z, W).

Axes in Closed Loop may not be combined with other axes in Open Loop. Therefore, all the axes must operate in non-servo controlled open loop.

There are 5 signals to control the motors and they are:

- Fast.
- Slow.
- Moving direction (+/-)
- Brake.
- In-Position.

The CNC outputs these signals via connectors IO1/IO2, as described next.

CONNECTOR I/O 1

Pin	SIGNAL AND FUNCTION	
1	0V.	Input from external power supply
2	T Strobe	Output. The BCD outputs refer to a tool code.
3	S Strobe	Output. The BCD outputs refer to an "S" code (spindle).
4	M Strobe	Output. The BCD outputs refer to an "M" code.
5	Emergency	Output
6	**W axis Brake**	**Output**
7	**Z axis Brake**	**Output**
8	**Y axis Brake**	**Output**
9	**X axis Brake**	**Output**
10	Home switch (X)	Input from X axis home switch.
11	Home switch (Y)	Input from Y axis home switch.
12	Home switch (Z)	Input from Z axis home switch.
13	Home switch (W)	Input from W axis home switch.
14	Emergency Stop	Input
15	Feed Hold	Input
	Transfer inhibit	
	M done	
16	Cycle Stop	Input
	Emergency Subroutine	
17	Cycle Start	Input
	Rapid feed	
	Enter in Play-back mode	
18	Block Skip	Conditional input
19	DRO	Input. The CNC acts as a DRO.
20	MST80	BCD coded output, weight: 80
21	MST40	BCD coded output, weight: 40
22	MST20	BCD coded output, weight: 20
23	MST10	BCD coded output, weight: 10
24	MST08	BCD coded output, weight: 8
25	MST04	BCD coded output, weight: 4
26	MST02	BCD coded output, weight: 2
27	MST01	BCD coded output, weight: 1
28	CHASSIS	Connect all cable shields to this pin.
29	24V.	Input from external power supply.
30		Not being used at this time.
31		Not being used at this time.
32		Not being used at this time.
33		Not being used at this time.
34		Not being used at this time.
35		Not being used at this time.
36		Not being used at this time.
37		Not being used at this time.

CONNECTOR I/O 2

PIN	SIGNAL AND FUNCTION	
1	0V.	Input from external power supply
2	0V.	Input from external power supply
3	Output M1 **Fast for X**	Value of bit 1 of "M" function table.
4	Output M2 **Fast for Y**	Value of bit 2 of "M" function table.
5	Output M3 **Fast for Z**	Value of bit 3 of "M" function table.
6	Output M4 **Fast for W**	Value of bit 4 of "M" function table.
7	Output M5 **Slow for X**	Value of bit 5 of "M" function table.
8	Output M6 **Slow for Y**	Value of bit 6 of "M" function table.
9	Output M7 **Slow for Z**	Value of bit 7 of "M" function table.
10	Output M8 **Slow for W**	Value of bit 8 of "M" function table.
11	Output M9 **Direction for X**	Value of bit 9 of "M" function table.
12	Output M10 **Direction for Y**	Value of bit 10 of "M" function table.
13	Output M11 **Direction for Z**	Value of bit 11 of "M" function table
14		Not being used at this time
15		Not being used at this time
16	CHASSIS	Connect all cable shields to this pin.
17		Not being used at this time
18		Not being used at this time
19	24V.	Input from external power supply.
20	24V.	Input from external power supply.
21	JOG **W In-Position**	Output. JOG mode selected.
22	Output M15 **Z In-Position**	Value of bit 15 of "M" function table.
23	Output M14 **Y In-Position**	Value of bit 14 of "M" function table.
24	Output M13 **X In-Position**	Value of bit 13 of "M" function table.
25	Output M12 **Direction for W**	Value of bit 12 of "M" function table.

Atention:

It is recommended NOT to use the decoded "M" function table because the CNC utilizes the same outputs to activate the table bits and the "Fast", "Slow", "Direction" and "In-Position" signals for each axis.

When the machine has a "W" axis, the CNC uses pin 21 for the "In-Position" signal of the "W" axis. It **does not** output the "JOG" signal.

3.2 *MACHINE PARAMETERS*

P626(8) The machine uses non-servo controlled open loop motors.

It indicates whether or not the machine uses non-servo controlled open loop motors.

0 = No
1 = Yes

When using this feature "P626(8)=1" the axes must not be continuously controlled: "P105=N", "P205=N", "P305=N", "P405=N".

P807, P811, P816, P820 Delay between "Brake" and "Fast" signals for X,Y,Z,W

It indicates the delay, in milliseconds, to be applied from the moment the brake is deactivated until the axes start moving ("Fast" signal).

Possible values: 0 through 65535 milliseconds.

P808, P812, P817, P821 Delay between "Slow" and "Brake" signals for X,Y,Z,W

It indicates the delay, in milliseconds, to be applied from the moment the "Slow" signal is deactivated until the "Brake" is activated for the corresponding axis.

Possible values: 0 through 65535 milliseconds.

P809, P813, P818, P822 Delay between "Brake" and "In-Position" signals: X,Y,Z,W

It indicates the delay, in milliseconds, to be applied from the moment the "Brake" signal is activated until the "In-Position" signal is activated for that axis.

Possible values: 0 through 65535 milliseconds.

P810, P814, P819, P823 Duration of the In-Position signal for X, Y, Z, W

It indicates the number of milliseconds that the "In-Position" signal remains active for the corresponding axis.

Possible values: 0 through 65535 milliseconds.

P900, P901, P902, P903 Braking distance for X, Y, Z, W

It indicates how far ahead of the target point the "Slow" signal is to be activated.

Possible values: ± 8388.607 millimeters.
± 330.2599 inches.

It must be assigned a value greater than the stopping distance: "P904, P905, P906, P907"

P904, P905, P906, P907 Stopping distance for X, Y, Z, W

It indicates how far ahead of the target point the "Slow" signal is to be deactivated.

Possible values: ± 8388.607 millimeters.
± 330.2599 inches.

3.3 BASIC OPERATION

Whenever an axis is to be moved, the CNC behaves as follows:

1.- Sets the "Brake" output high (24V) for the electrical cabinet to deactivate the axis brake.

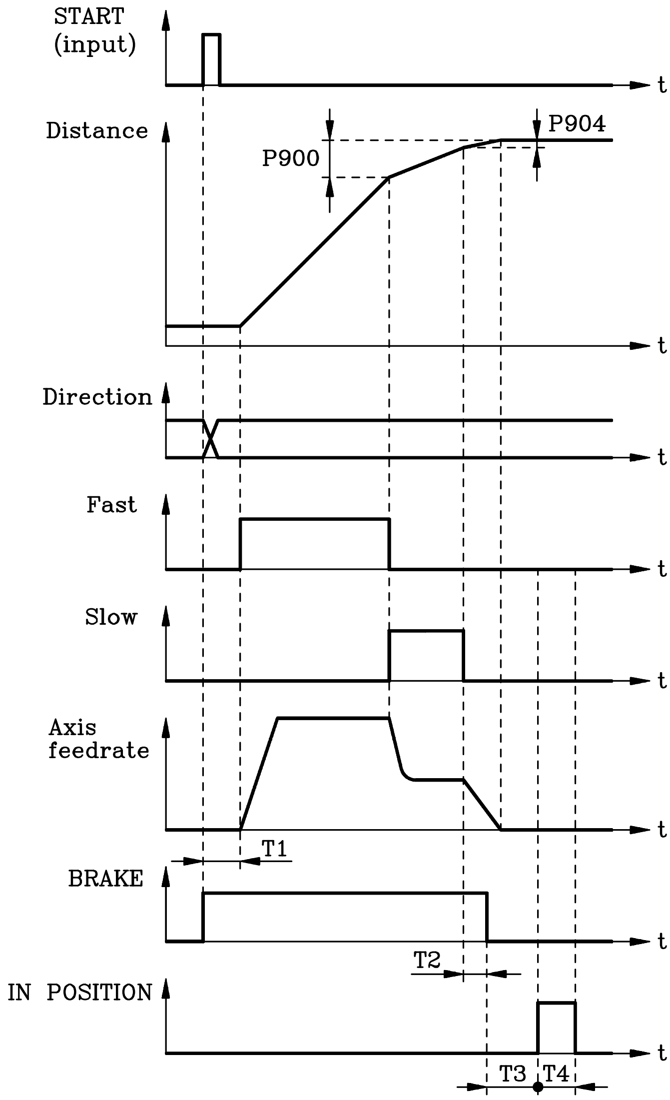
2.- Since the brake is not cancelled instantaneously, it is possible to set a delay "T1" by means of machine parameters P807, P811, P816, P820, before activating the "Fast" output.

3.- Once "T1" has elapsed, the CNC activates the "Fast" output for the axis to start moving.

4.- This "Fast" output is maintained active until the axis reaches the braking distance set by P900, P901, P902, P903 from the target point. At this point on, the CNC activates the "Slow" output.

5.- When the axis enters the stopping zone (at a P904, P905, P906, P907 distance from the target point), the CNC deactivates the "Slow" output.

6.- In order to allow the axis enough time to position before activating its brake, it is possible to set a delay "T2" by means of machine parameters P808, P812, P817, P821



This parameter indicates the delay applied from the moment the "Slow" output is deactivated until the "Brake" output is set low.

6.- After setting the "Brake" output low, the CNC waits a "T3" time period, set by P809, P813, P818, P822, before activating the "In-Position" signal for the axis.

This "In-Position" output is kept high during "T4" which as established by machine parameters P810, P814, P819, P823

3.4 MOVEMENT EXECUTION

The movements of the axes may be programmed by means of either G00 or G01 functions. If G02 or G03 is programmed, the CNC will issue error 14.

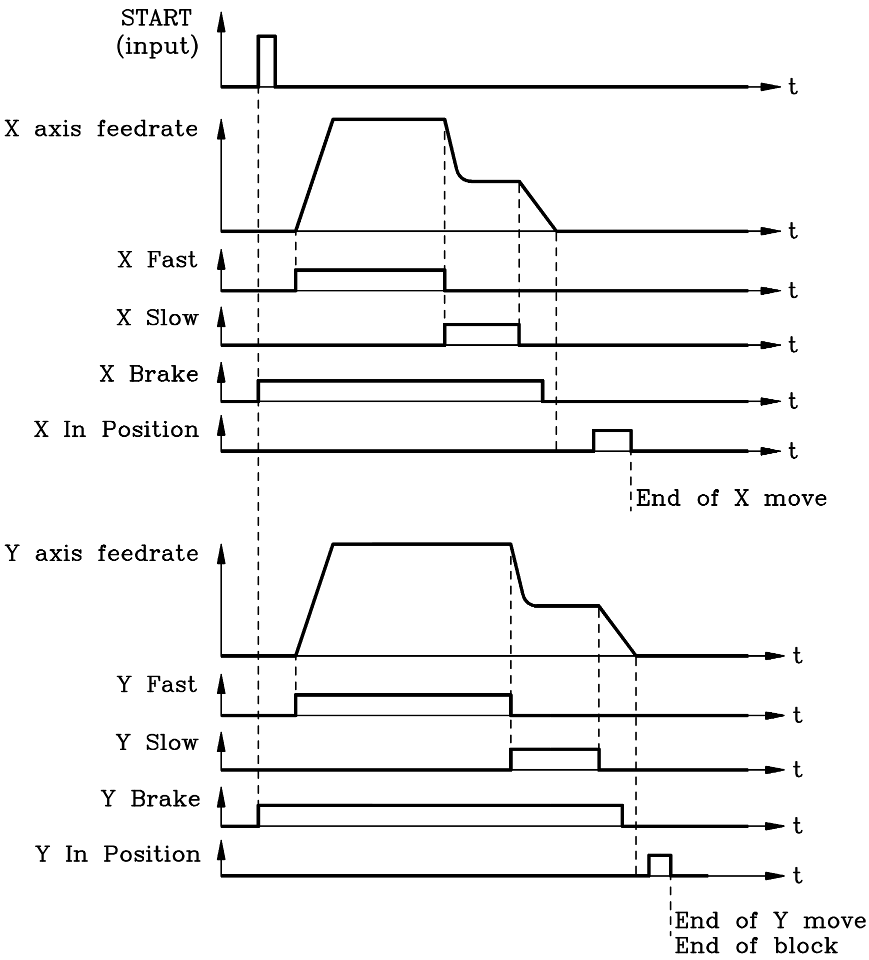
All the movements are carried out as described earlier. Therefore, it is the same to program G00 or G01.

A program block may contain the movements of up to 3 axes simultaneously.

The CNC considers the execution of a block concluded when all the axes involved have reached position. In other words, when all their "In-Position" outputs are high.

Usually, the movements of all the axes involved do not end at the same time (different distances, different T1, T2, T3, T4 delays, etc.).

Execution example of a block involving X and Y movements.



3.5 AUTOMATIC AND SINGLE BLOCK MODES

3.5.1 USING FUNCTION G05 AND G07

When operating in automatic mode, the CNC waits for a block to be finished before starting the execution of the next block.

When operating in G07, the CNC considers the block execution concluded when all the axes involved have reached position. That is, when all their "In-Position" outputs are high.

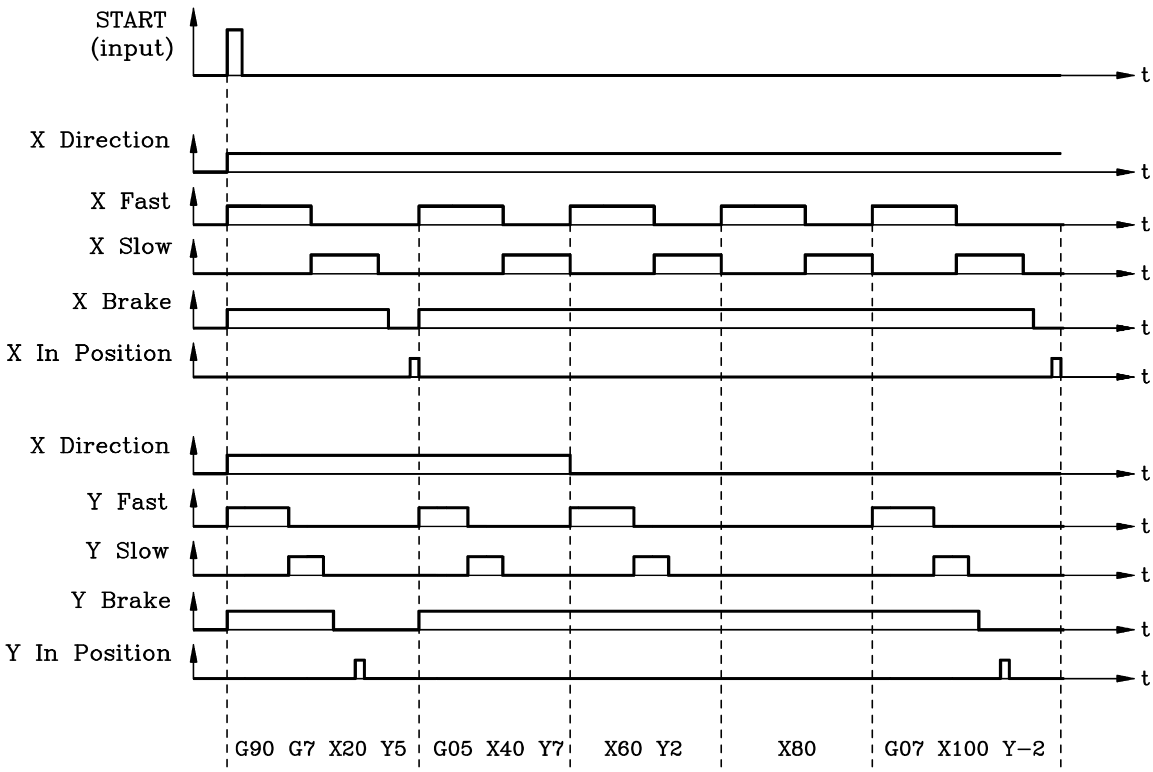
When operating in G05, the CNC acts as follows:

When the axis enters the stopping zone (at a P904, P905, P906, P907 distance from the target point), the CNC deactivates the "Slow" output and it does not issue the "Brake" signal or the "In-Position" signal.

The CNC considers the block execution concluded when all the axes involved have entered the stopping zone. In other words, when all their "Slow" outputs are set low (deactivated).

Example:

N00 G90 G07 X20 Y5
N10 G05 X40 Y7
N20 X60 Y2
N30 X80
N40 G07 X100 Y-2
N50 M30



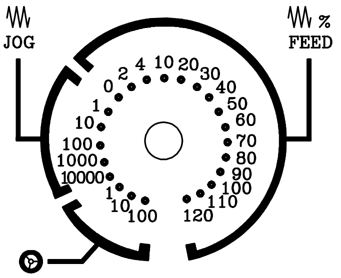
3.5.2 SINGLE BLOCK EXECUTION

When operating in Single-Block mode, the CNC ignores function G05 and executes all the movements as if they were programmed in G07.

The CNC considers the execution of the block concluded when all the involved axes have reached position. In other words, when all their "In-Position" outputs are high.

If while executing a G05 in Automatic mode, you switch to Single Block mode, the CNC will execute this block in G07 and it will generate the "Brake" and "In-Position" signals at the end of the block.

3.5.3 MANUAL FEEDRATE OVERRIDE SWITCH



When selecting any position between 4% and 120% of the %Feed area, the Fast/Slow sequence will be the one described earlier.

When selecting the 2% position of the %Feed area, the CNC always applies the "Slow " feed.

In other words, if while being the "Fast" output active, the 2% position is selected, the CNC cancels the "Fast" output and it activates the "Slow" output, instead.

When selecting the 0% position of the %Feed area, any JOG position or any Handwheel position ( ), the CNC will stop the axes cancelling the "Fast" and "Slow" outputs. It does not change the status of the "Brake" and "In-Position" outputs.

3.5.4 CYCLE STOP AND FEEDHOLD SIGNALS

Whenever the key is pressed, the "Cycle Stop" input is set low (pin 16 of connector I/O1 to 0V) or the Feedhold input is set low (pin 15 of connector I/O1 to 0V), the CNC behaves as follows:

* It stops the axes cancelling the "Fast" and "Slow" outputs.
* It does not change the status of the "Brake" and "In-Position" outputs.

If the Feedhold input was set low, when setting it back high, the "Fast" and "Slow" signals will recover their previous status.

3.6 JOG MODE

When operating in JOG mode, the CNC maintains the "In-Position" signals low. It does not generate these signals at the end of the movement.

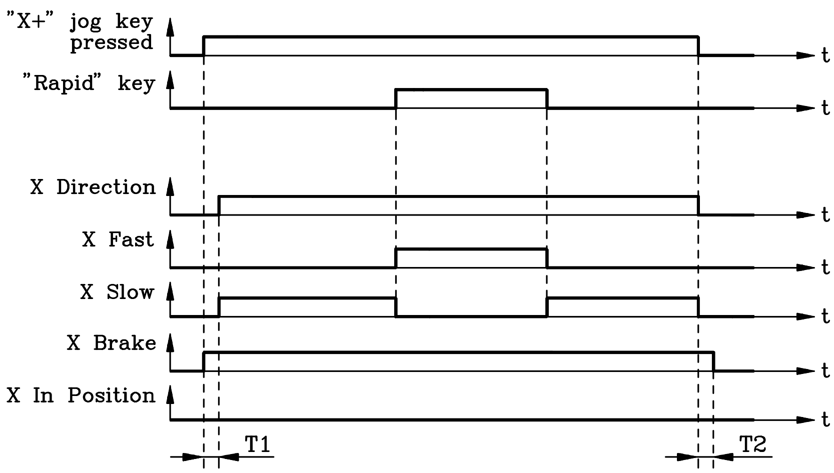
Next, a description of the CNC's behavior in each of the MFO switch position:

3.6.1 % FEED AREA (CONTINUOUS JOG)

When selecting any of the positions between 2% and 120%, the movements are carried out at "Slow" feed. When selecting the 0% position, the CNC inhibits the axes.

If while jogging the axes, the key is pressed, the CNC deactivates the "Slow" output and activates the "Fast" output. When releasing the key, the "Fast" output is set low and the "Slow" output is set back high.

Example:



3.6.2 INCREMENTAL JOG

Every time a JOG key is pressed, the CNC moves the axis the distance selected at the MFO switch (1, 10, 100, 1000 or 10000).

Depending on the selected distance and feedrate, the movement will be carried out at "Fast and Slow" or just at "Slow" feed.

The CNC also takes into account the "T1" and "T2" delays for treating the "Brake" signal.

3.6.3 HANDWHEEL AREA

If while any handwheel position is selected ( ), an axis key is pressed or the axis selector button on the back of the Fagor 100P model handwheel is pressed, the CNC sets the "Brake" output high.

From this moment on, the CNC will move the axis depending on the feedback pulses provided by the handwheel and applying the x1, x10 or x100 multiplying factor currently selected at the MFO switch.

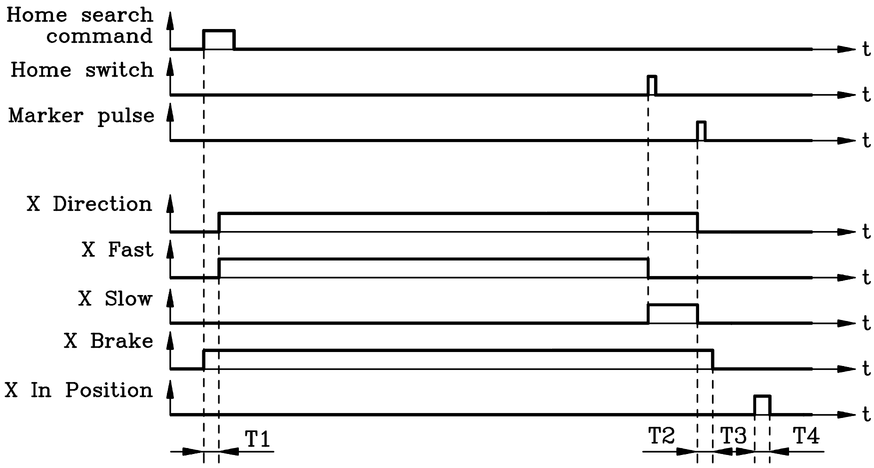
Depending on the pulses received, on the selected MFO position and feedrate, the movement will be carried out at "Fast and Slow" feed or just at "Slow" feed.

The CNC also takes into account the "T1" and "T2" delays for treating the "Brake" signal.

3.7 HOME SEARCH

Although it is possible to program the home search on several axes in the same block, the CNC homes the axes one at a time as described below:

The axis has a home switch:



The homing direction is set by machine parameters "P623(8), P623(7), P623(6), P623(5)".

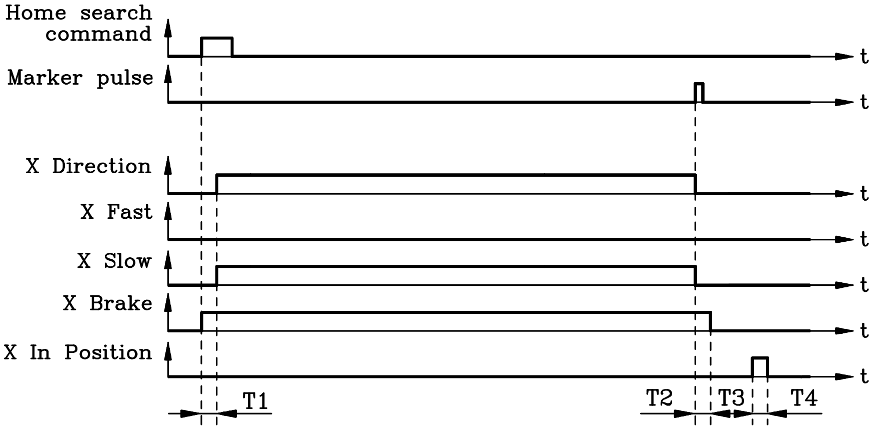
The axis moves at "Fast" feed until the home switch is pressed. Once the home switch is pressed, the axis changes to "Slow" feed until the home pulse (marker pulse Io) of the feedback system is detected.

Atention:

If, when initiating the homing process, the home switch is being pressed, the axis will reverse until the switch is released before resuming the home search.

When homing in JOG mode, the CNC **does not** activate the "In-Position" signal.

The axis does not have a home switch:



The moving direction is set by machine parameters "P623(8), P623(7), P623(6), P623(5)".

The axis moves at "Slow" feed until the marker pulse (Io) from the feedback device is detected.

Atention:

When homing in JOG mode, the CNC **does not** activate the "In-Position" signal.