FAGOR AUTOMATION

CNC 8025 T, TS

New Features (Ref. 0107 in)

FAGOR

ERRORS FOUND IN THE PROGRAMMINGMANUAL (REF. 9701)

Page 64. Function G51.

When working in diameters, the "I" value in the table is in diameters and the value to be assigned to parameter "I" in the G51 function must always be given in radius.

Section 12.4. (Chapter 12 page 133) Nesting levels.

The figure reads M02 ó M30
It should read: M02 or M30

ERRORS FOUND IN THE OPERATING MANUAL (REF. 9701)

Page 46. Last paragraphs.

It should say:

The CNC asks which is the source program number and which is the new program number, after keying each one of them, press [ENTER].

If the number of the source program does not exist, or there is already a program in memory with the same number as the new one or if there is not enough when copying the new program , the CNC will issue a message indicating the cause.

ERRORS FOUNDS IN THE INSTALLATION MANUAL (REF. 9707)

Section 5.4 (chapter 5 page 7) Machine parameters for spindle control:

Parameter P606(3) is missing:

P606(3) Spindle counting direction

It sets the spindle counting direction. If correct, leave it as it is or change it if otherwise.
Possible values: "0" and "1".

MODIFICATIONS TO THE INSTALLATION MANUAL (REF. 9707)

Section 2.3.4 (chapter 2 page 9). Logic Outputs:

In the table, the following output is missing:

Output "C"
Row 1: (pin 3 I/O 1) M strobe
Row 2: (pin 5 I/O 2) output 3, decoded M function

Section 3.3.3 (chapter 3 page 11). P602(4). Another example:

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

P602(1)=0 Millimeters P501=1 Resolution 0.001 mm. P602(4)=0 x4 Multiplication factor

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 LAN MANUAL (REF. 9701)

Section 2.2 (page 3). P616(7)

The first 2 paragraphs change. They should say:

If "P616(7)=0" the 8025 T CNC uses pin 15 of connector I/O1 as the input for the Feed-Hold, Transfer-Inhibit and M-done signals as described in the Installation manual, chapter 1 section "Inputs of connector I/O 1"

If "P616(7)=1" the CNC behaves as follows:

* The Feed-Hold input will be "taken" ....

Version 7.1 (July 1996)

1. EXPANSION OF THE INTEGRATED PLC RESOURCES

1.1 INPUTS

1.1.1 TYPE OF FEEDRATE (G94/G95)

PLCI input I86 will show at all times the type of feedrate (F) selected a the CNC.

I86 = 0 G94. Feedrate in millimeters (inches) per minute.
I86 = 1 G95. Feedrate in millimeters (inches) per revolution.

1.1.2 TYPE OF CUTTING SPEED (G96/G97)

PLCI input I87 will show at all times the type of cutter speed selected at the CNC.

I87 = 0 G97. Constant tool center speed.
I87 = 1 G96. Constant cutting-edge speed

1.1.3 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 3rd 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 4th axis is being homed (0=No / 1=Yes)
I104	Indicates whether the C axis is being homed (0=No / 1=Yes)

1.1.4 AXIS MOVING DIRECTION

Inputs I42, I43, I44 and I45 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 3rd 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 4th 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 "P621(7)" indicates whether this feature is available or not.

P621(7) = 0 This feature **is not** available.
P621(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 "P621(7)" indicates whether this feature is available or not.

P621(7) = 0 This feature **is not** available.
P621(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 3rd axis limits
O56 / O57	Positive / negative Z axis limits
O58 / O59	Positive / negative 4th 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"P621(7)" indicates whether this feature is available or not.
P621(7) = 0 This feature **is not** available.
P621(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 "P621(7)" indicates whether this feature is available or not.

P621(7) = 0 This feature **is not** available
P621(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
M44	M43	M42	M41				M19	M1	M30			M4	M3	M2	M0

2. 4TH AXIS NOW AVAILABLE ON 8025T MODELS

From this version on, this feature is now available on all these models:

CNC-8025T (not available until now)	CNC-8025TG	CNC-8025TS
CNC-8025TI (not available until now)	CNC-8025TGI	CNC-8025TSI

3. SPINDLE SPEED DISPLAY UNITS

Until now, the spindle speed was always displayed in rpm. From now on, the display units may be selected by means of machine parameter "P621(6)".

P621(6) = 0 In rpm when operating in RPM and in m/min. (ft./min.) when at Constant Surface Speed.
P621(6) = 1 Always in rpm, even when operating at Constant Surface Speed.

4. SINGLE BLOCK TREATMENT

The CNC considers a "Single block" the group of blocks between a G47 and a G48.

After executing function G47, the CNC executes all the following blocks until executing a block containing function G48 **even when in Single Block mode**

If  is pressed while executing a "single block" in Automatic or Single-Block mode, the CNC keeps executing the rest of the blocks until it runs into a G48 and it, then, interrupts program execution.

While function G47 is active, the Manual Feedrate Override switch and the spindle speed override keys will be disabled, thus the program will be executed at 100% of the programmed F and S values.

Functions G47 and G48 are modal and incompatible with each other.

On power-up, after executing an M02/M30, after an EMERGENCY or a RESET, the CNC assumes G48.

5. TWO ELECTRONIC HANDWHEELS ARE NOW POSSIBLE

From this version on, up 2 electronic handwheels may be used one for the X axis and another one for the Z axis. The 4th axis and the Live Tool will no longer be available. The feedback inputs will be used as follows:

A1 - X axis; A2 - Z axis handwheel; A3 - Z axis; A4 - 3rd axis or "C" axis; A5 - Spindle; A6 - X axis handwheel

The handwheels will be operative when selecting the JOG mode. One of the handwheel positions must also be selected at the Manual Feedrate Override switch of the operator panel.

The possible positions are: 1, 10 and 100, which indicate the multiplying factor applied to the pulses coming from the electronic handwheel.

This way and after applying the multiplying factor, one obtains the axis moving units. These units correspond to the units used for the display format

Example: Handwheel Resolution : 250 lines per turn

MFO Switch position	Distance per turn
1	0.250 mm or 0.0250 inch
10	2.500 mm or 0.2500 inch
100	25.000 mm or 2.5000 inches

When attempting to "crank" an axis faster than its maximum feedrate (machine parameters "P110, P310"), the CNC will limit the actual axis feedrate to that parameter value ignoring the rest of the pulses supplied by the handwheel, thus preventing a Following Error message from being issued.

5.1 MACHINE PARAMETERS FOR THE HANDWHEELS:

P622(6) = 0 There **is no** electronic handwheel associated with the Z axis
P622(6) = 1 There **is** electronic handwheel associated with the Z axis

P609(1) = 0 The electronic handwheel being used **is not** a FAGOR 100P model.
P609(1) = 1 The electronic handwheel being used **is** a FAGOR 100P model.

This parameter makes sense when using a single handwheel associated with the X axis. It indicates whether or not it is a FAGOR 100P with axis selector button.

P500 Counting direction of the X axis handwheel (No / Yes)
P622(5) Counting direction of the Z axis handwheel (0 / 1)

P602(1) Feedback units of the X axis handwheel (0 = millimeters /1 = inches)
P622(3) Feedback units of the Z axis handwheel (0 = millimeters /1 = inches)

P501 Square-wave feedback resolution of the X axis handwheel.
P622(1,2) Square-wave feedback resolution of the Z axis handwheel.

P501	P622(2)	P622(1)	Resolution	
1	0	0	0.001 mm	0.0001"
2	0	1	0.002 mm	0.0002"
5	1	0	0.005 mm	0.0005"
10	1	1	0.010 mm	0.0010"

P602(4) Multiplying factor for X axis handwheel feedback pulses (0= x4 / 1= x2)
P622(4) Multiplying factor for Z axis handwheel feedback pulses (0= x4 / 1= x2)

P621(2) = 0 Handwheel disabled for Manual Feedrate Override (MFO) switch positions other than the handwheel positions.
P621(2) = 1 When the MFO is at a position other than those for the handwheel , the CNC takes it into account and applies a "x1" multiplying factor.

Example: Having a Fagor electronic handwheel (25 lines per turn) set as follows:
P602(1)=0 Millimeters; P501=1 Resolution 0.001 mm.; P602(4)=0 x4 Multiplication factor
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

5.2 USING ELECTRONIC HANDWHEELS

The machine uses one electronic handwheel

When using a single electronic handwheel, it must be connected to A6.

If the handwheel is a FAGOR 100P type, machine parameter "P609(1)" must be set to "1".

Once the desired handwheel position has been selected at the MFO switch, press one of the JOG keys of the axis to be jogged. The selected axis appears highlighted.

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

* Press the push-button on the rear of the handwheel. The CNC selects the first axis and it highlights it.

* By pressing the button again, the next axis is selected and so on, rolling over from the last axis to the first one.

* By keeping the button pressed for more than 2 seconds, the CNC de-selects the currently selected axis.

The selected axis will be jogged as the handwheel is turned, reversing directions when reversing the turning direction of the handwheel.

When trying to move an axis faster than the maximum feedrate allowed (machine parameter "P110, P310"), the CNC will limit the actual feedrate to that parameter value ignoring the additional pulses, thus, avoiding following error messages.

The machine uses two electronic handwheels

Each axis will move as its associated handwheel is turned, reversing its direction as the handwheel turning direction is reversed and according to the selected MFO switch position.

When trying to move an axis faster than the maximum feedrate allowed (machine parameter "P110, P310"), the CNC will limit the actual feedrate to that parameter value ignoring the additional pulses, thus, avoiding following error messages.

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 "P619(5)" indicates whether this feature is to be used or not.

P619(5) = 0 This feature is not being used.
P619(5) = 1 This feature is being used.

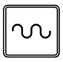
2. JOGGING FEEDRATE

From this version on, machine parameter P812 sets the axis jogging feedrate selected by the CNC when accessing the JOG mode.

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. RAPID TRAVERSE KEY IN JOG MODE

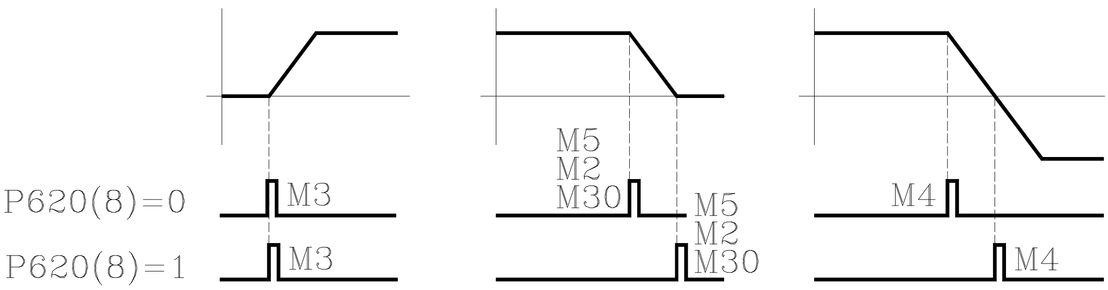
Whenever the conditional input (block skip), pin 18 of connector I/O1, the CNC will ignore the rapid traverse key



Version 7.4 (May 1999)

1. NEW MACHINE PARAMETER ASSOCIATED WITH THE M FUNCTIONS

Machine parameter "P620(8)" 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 P620(5), P620(6), P613(8) and P613(7) are used together with P602(6), P602(5), P612(5) and P614(5) which indicate the multiplying factor to be applied to the feedback signals of the X, Z, 3rd and 4th axes respectively.

X axis	Z axis	3rd axis	4th axis
P602(6)	P602(5)	P612(5)	P614(5)
P620(5)	P620(6)	P613(8)	P613(7)

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

P620(5)=0, P620(6)=0, P613(8)=0 y P613(7)=0 They are not divided
P620(5)=1, P620(6)=1, P613(8)=1 y P613(7)=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 P602(6)=0 & P620(5)=0	x4 multiplying factor	Nr of pulses = 125
With P602(6)=1 & P620(5)=0	x2 multiplying factor	Nr of pulses = 250
With P602(6)=0 & P620(5)=1	x2 multiplying factor	Nr of pulses = 250
With P602(6)=1 & P620(5)=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.

P623(1) = 0 Not affected. It is always at 100%, like in previous versions.
P623(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 3rd axis
P821 Feedback factor for the Z axis — P822 Feedback factor for the 4th 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 TLI is now available.
It offers the same features as the TGI model and it is sold together with the motors and ACS drives..

FAGOR 8025/8030 CNC

Models: T, TG, TS

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 LATHE MODEL FAGOR 8025/8030 CNCs

TECHNICAL DESCRIPTION

	T	TG	TS
INPUTS/OUTPUTS			
Feedback inputs.	**6**	**6**	**6**
Linear axes	**4**	**4**	**4**
Rotary axes	**2**	**2**	**2**
Spindle encoder	**1**	**1**	**1**
Electronic handwheel	**1**	**1**	**1**
Third axis as "C" axis			**x**
Synchronized tool			**x**
Probe input	**x**	**x**	**x**
Square-wave feedback signal multiplying factor, x2/x4	**x**	**x**	**x**
Sine-wave feedback signal multiplying factor, x2/x4/10/x20	**x**	**x**	**x**
Maximum counting resolution 0.001mm/0.001°/0.0001inch	**x**	**x**	**x**
Analog outputs (±10V) for axis servo drives	**4**	**4**	**4**
Spindle analog output (±10V)	**1**	**1**	**1**
Live tool	**1**	**1**	**1**
AXIS CONTROL			
Axes involved in linear interpolations	**3**	**3**	**3**
Axes involved in circular interpolations	**2**	**2**	**2**
Electronic threading	**x**	**x**	**x**
Spindle control	**x**	**x**	**x**
Software travel limits	**x**	**x**	**x**
Spindle orientation	**x**	**x**	**x**
PROGRAMMING			
Part Zero preset by user	**x**	**x**	**x**
Absolute/incremental programming	**x**	**x**	**x**
Programming in cartesian coordinates	**x**	**x**	**x**
Programming in polar coordinates	**x**	**x**	**x**
Programming by angle and cartesian coordinate	**x**	**x**	**x**
COMPENSATION			
Tool radius compensation	**x**	**x**	**x**
Tool length compensation	**x**	**x**	**x**
Leadscrew backlash compensation	**x**	**x**	**x**
Leadscrew error compensation	**x**	**x**	**x**
DISPLAY			
CNC text in Spanish, English, French, German and Italian	**x**	**x**	**x**
Display of execution time	**x**	**x**	**x**
Piece counter	**x**	**x**	**x**
Graphic movement display and part simulation		**x**	**x**
Tool tip position display	**x**	**x**	**x**
Geometric programming aide	**x**	**x**	**x**
COMMUNICATION WITH OTHER DEVICES			
Communication via RS232C	**x**	**x**	**x**
Communication via DNC	**x**	**x**	**x**
Communication via RS485 (FAGOR LAN)	**x**	**x**	**x**
ISO program loading from peripherals	**x**	**x**	**x**
OTHERS			
Parametric programming	**x**	**x**	**x**
Model digitizing			**x**
Possibility of an integrated PLC	**x**	**x**	**x**

PREPARATORY FUNCTIONS

	T	TG	TS
AXES AND COORDINATES SYSTEMS			
Part measuring units. Millimeters or inches (G70,G71)	x	x	x
Absolute/incremental programming (G90,G91)	x	x	x
Independent axis (G65)	x	x	x
REFERENCE SYSTEMS			
Machine reference (home) search (G74)	x	x	x
Coordinate preset (G92)	x	x	x
Zero offsets (G53...G59)	x	x	x
Polar origin offset (G93)	x	x	x
Store current part zero (G31)	x	x	x
Recover stored part zero (G32)	x	x	x
PREPARATORY FUNCTIONS			
Feedrate F	x	x	x
Feedrate in mm/min. or inches/min. (G94)	x	x	x
feedrate in mm/revolution or inches/revolution (G95)	x	x	x
Programmable feed-rate override (G49)	x	x	x
Spindle speed (S)	x	x	x
Spindle speed in rpm (G97)	x	x	x
Constant Surface Speed (G96)	x	x	x
S value limit when working at constant surface speed (G92)	x	x	x
Tool and tool offset selection (T)	x	x	x
Activate "C" axis in degrees (G14)			x
Main plane C-Z (G15)			x
Main plane C-X (G16)			x
AUXILIARY FUNCTIONS			
Program stop (M00)	x	x	x
Conditional program stop (M01)	x	x	x
End of program (M02)	x	x	x
End of program with return to first block (M30)	x	x	x
Clockwise spindle start (M03)	x	x	x
Counter-clockwise spindle start (M04)	x	x	x
Spindle stop (M05)	x	x	x
Spindle orientation (M19)	x	x	x
Spindle speed range change (M41, M42, M43, M44)	x	x	x
Tool change with M06	x	x	x
Live tool (M45 S)	x	x	x
Synchronized tool (M45 K)			x
PATH CONTROL			
Rapid traverse (G00)	x	x	x
Linear interpolation (G01)	x	x	x
Circular interpolation (G02,G03)	x	x	x
Circular interpolation with absolute center coordinates (G06)	x	x	x
Circular path tangent to previous path (G08)	x	x	x
Arc defined by three points (G09)	x	x	x
Tangential entry (G37)	x	x	x
Tangential exit (G38)	x	x	x
Controlled radius blend (G36)	x	x	x
Chamfer (G39)	x	x	x
Electronic threading (G33)	x	x	x
ADDITIONAL PREPARATORY FUNCTIONS			
Dwell (G04 K)	x	x	x
Round and square corner (G05, G07)	x	x	x
Scaling factor (G72)	x	x	x
Single block treatment (G47, G48)	x	x	x
User error display (G30)	x	x	x
Automatic block generation (G76)			x
Communication with FAGOR Local Area Network (G52)	x	x	x

	T	TG	TS
COMPENSATION			
Tool radius compensation (G40,G41,G42)	**x**	**x**	**x**
Loading of tool dimensions into internal tool table (G50, G51)	**x**	**x**	**x**
CANNED CYCLES			
Pattern repeat (G66)	**x**	**x**	**x**
Roughing along X (G68)	**x**	**x**	**x**
Roughing along Z (G69)	**x**	**x**	**x**
Straight section turning (G81)	**x**	**x**	**x**
Straight section facing (G82)	**x**	**x**	**x**
Deep hole drilling (G83)	**x**	**x**	**x**
Circular section turning (G84)	**x**	**x**	**x**
Circular section facing (G85)	**x**	**x**	**x**
Longitudinal threadcutting (G86)	**x**	**x**	**x**
Face threadcutting (G87)	**x**	**x**	**x**
Grooving along X (G88)	**x**	**x**	**x**
Grooving along Z (G89)	**x**	**x**	**x**
PROBING			
Probing (G75)	**x**	**x**	**x**
Tool calibration canned cycle (G75N0)			**x**
Probe calibration canned cycle (G75N1)			**x**
Part measuring canned cycle along X (G75N2)			**x**
Part measuring canned cycle along Z (G75N3)			**x**
Part measuring canned cycle with tool compensation along X (G75N4)			**x**
Part measuring canned cycle with tool compensation along Z (G75N5)			**x**
SUBROUTINES			
Number of standard subroutines	99	99	99
Definition of a standard subroutine (G22)	**x**	**x**	**x**
Call to a standard subroutine (G20)	**x**	**x**	**x**
Number of parametric subroutines	99	99	99
Definition of a parametric subroutine (G23)	**x**	**x**	**x**
Call to a parametric subroutine (G21)	**x**	**x**	**x**
End of standard or parametric subroutine (G24)	**x**	**x**	**x**
JUMP OR CALL FUNCTIONS			
Unconditional jump/call (G25)	**x**	**x**	**x**
Jump or call if zero (G26)	**x**	**x**	**x**
Jump or call if not zero (G27)	**x**	**x**	**x**
Jump or call if smaller (G28)	**x**	**x**	**x**
Jump or call if greater (G29)	**x**	**x**	**x**

NEW FEATURES AND MODIFICATIONS

Date: **March 1991** ***Software Version:*** **2.1 and newer**

FEATURE	MODIFIED MANUAL & SECTION	
The home searching direction is set by machine parameter P618(5,6,7,8)	Installation Manual	Section 4.7
The 2nd home searching feedrate is set by machine parameter P807...P810	Installation Manual	Section 4.7
New resolution values 1, 2, 5 and 10 for sine-wave feedback signals P619(1,2,3,4)	Installation Manual	Section 4.1
Access to PLCI registers from the CNC	Programming Manual	G52

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

FEATURE	MODIFIED MANUAL AND SECTION	
New function: F36. It takes the value of the selected tool number	Programming Manual	Chapter 13
G68 and G69 canned cycles modified. if P9=0 it runs another final roughing pass	Programming Manual	Chapter 13

Date: **September 1991** ***Software Version:*** **3.2 and newer**

FEATURE	MODIFIED MANUAL AND SECTION	
Subroutine associated with the T function	Installation Manual	Section 3.3.5
G68 and G69 canned cycles modified. P9 can now have a negative value	Programming Manual	Chapter 13

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

FEATURE	MODIFIED MANUAL AND SECTION	
Bell-shaped ACC./DEC.	Installation Manual	Section 4.8
It is now possible to enter the sign of the leadscrew backlash for each axis P620(1,2,3,4)	Installation Manual	Section 4.4
Independent axis movement execution	Programming Manual	G65
It is now possible to work at Constant Surface Speed in JOG mode P619(8)	Installation Manual	Section 3.3.9

Date: **July 1992** ***Software Version:*** **4.2 and newer**

FEATURE	MODIFIED MANUAL AND SECTION	
Synchronisation with independent axis P621(4)	Installation Manual	Section 3.3.10

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

FEATURE	MODIFIED MANUAL AND SECTION	
Linear & Bell-shaped acc./dec. ramp combination	Installation Manual	Section 4.8
Spindle acc/dec control. P811	Installation Manual	Section 5.
The subroutine associated with the tool is executed before the T function. P617(2)	Installation Manual	Section 3.3.5
G68 and G69 cycles modified. If P10 <> 0, it runs a final roughing pass before the finishing pass	Programming Manual	Chapter 13
When having only one spindle range, if G96 is executed without any range being selected, the CNC will automatically select it.	Programming Manual	Chapter 6
8030 CNC with VGA Monitor	Installation Manual	Chapter 1

Date: **March1995** ***Software Version:*** **5.3 and newer**

FEATURE	MODIFIED MANUAL AND SECTION	
Management of semi-absolute feedback devices (with coded Io)	Installation Manual	Sections 4.7 & 6.5.
Spindle inhibit by PLC	Installation Manual	Section 3.3.10
Handwheel managed by PLC	Installation Manual	Section 3.3.3
Simulation of the "rapid JOG" key from PLC	PLCI Manual	
Initialization of machine parameters in case of memory loss.		

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

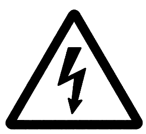
WARNING. symbol

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

Symbols that may be carried on the product

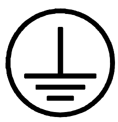
WARNING. symbol

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

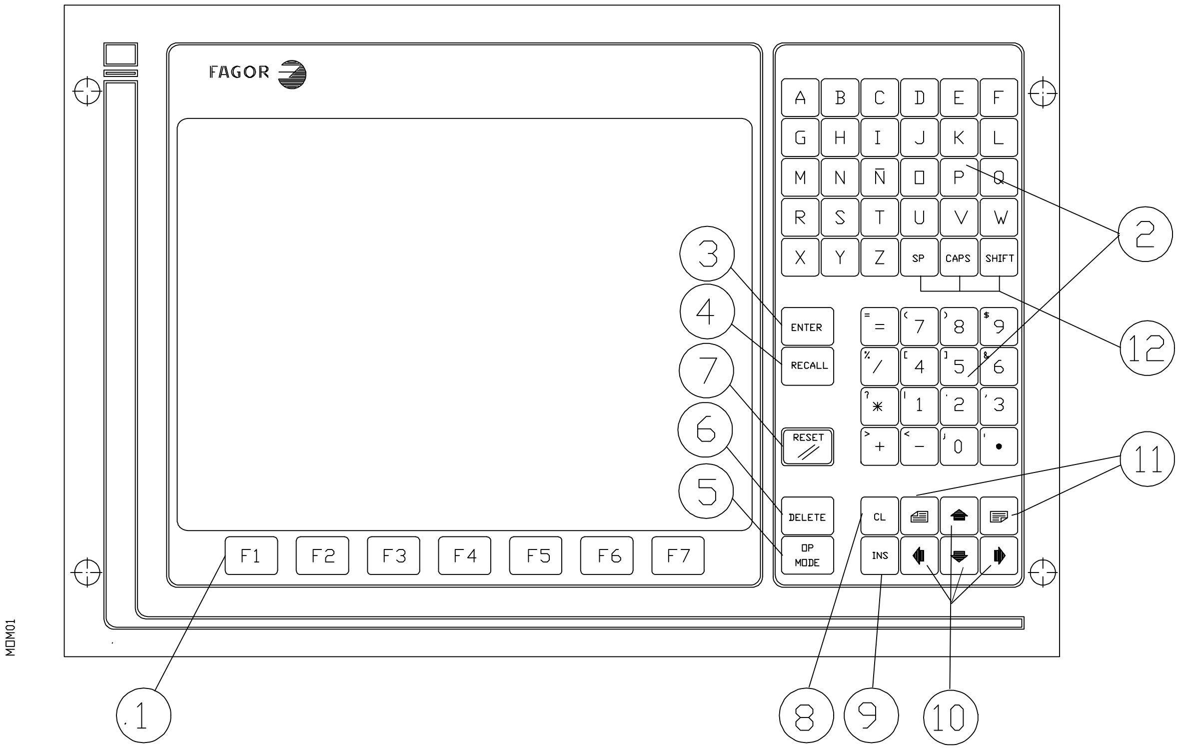
This manual contains the information required for the proper operation of the CNC.

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.

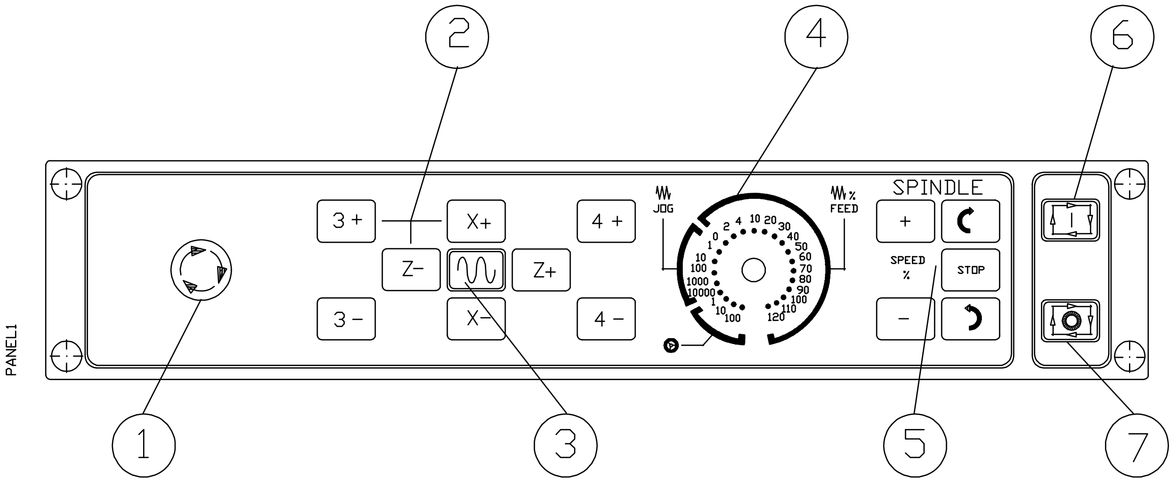
9. **INS**. Key which allows characters to be inserted during the edition of a program block.

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.

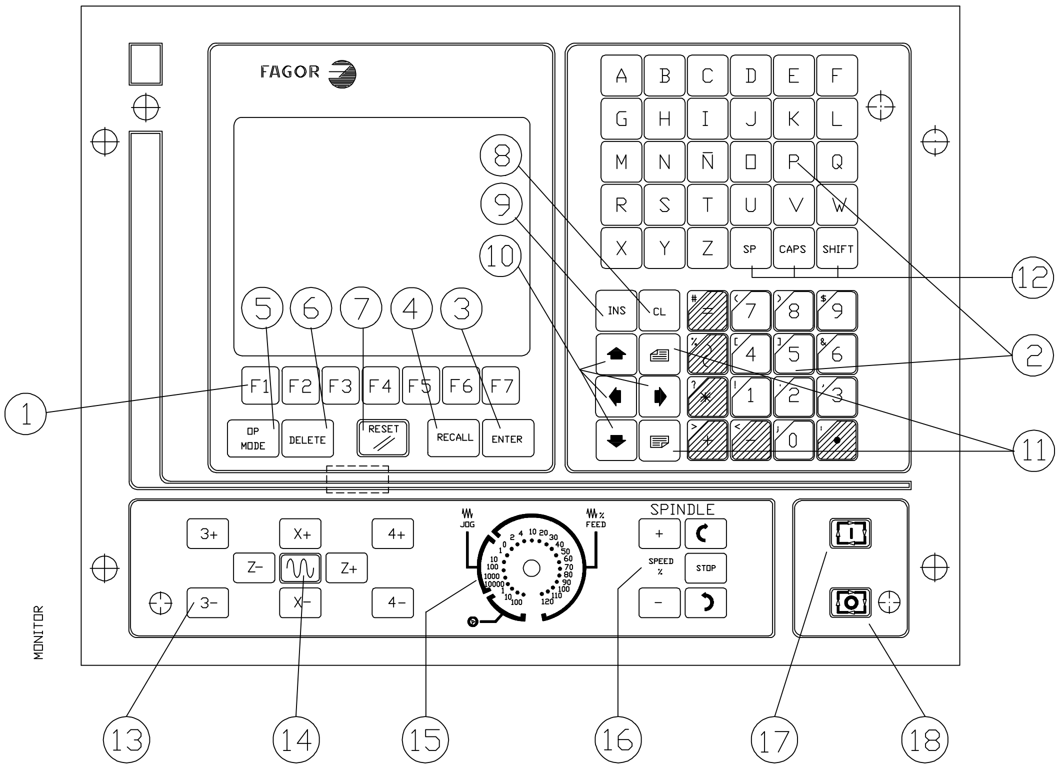
4. Switch (M.F.O.), which allows a % variation of the programmed feedrate and to choose the different ways of working in the JOG MODE (continuous, incremental, electronic handwheel).

5. Spindle operating keys. Allow the spindle to be put into OPERATION and to STOP it, in the JOG 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 mode. The $\boxed{+}$ and $\boxed{-}$ 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 P611 bits (8) and (7).

P611 (8)	P611 (7)	Monitor
0	0	Monochrome
0	1	Combination 1
1	0	Combination 2

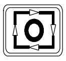
Each of the combinations, 1 and 2, are a group of 3 different colors to distinguish the characters displayed.

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.
- 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 STAR**T 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 progra**m**

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 MOD**E : 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 **RECAL**L

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.

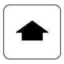
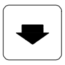
If you wish to begin with a different block, the following procedure should be followed:

- Press the N key
- Key in the number of the block
- Press **RECAL**L

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 block**s**

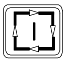
To display the contents of the blocks prior or subsequent to those appearing on the screen:

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

<u>Atention:</u>

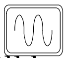
The program always starts with the block whose number appears to the right of the program number, regardless of which ones are displayed on the screen.

3.1.1.5. Cycle Start

- Press 

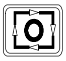
. Once the program and block number have been selected, just press this key to execute the program in **AUTOMATIC** or the block in **SINGLE BLOCK**.

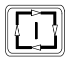
. If the program contains any conditional block it will be executed when the relevant input is activated (see INSTALLATION AND START-UP MANUAL). If it is not activated, the CNC will disregard such block.

. 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 P600(3) has a value equal to zero.

. 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 pres 

The cycle is also stopped by means of:

- Codes M00,M02,M30.
- Code M01 when the relevant input is activated.
- The external signal **STOP**.
- The external signal **FEED-HOLD** (the cycle continues when the signal disappears)
- The external signal **EMERGENCY STOP** (in this case the program must be restarted, since the CNC is reset to initial state).
- The external **EMERGENCY** Subroutine Jump signal

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 BLOC**K 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.

Underneath and at the same level as **COMMAND**, the Programmed S value, multiplied by the %, on the same level as **ACTUAL**, the real S value and at the same level as **TO GO** (RPM) or (M/MIN).

. 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

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.

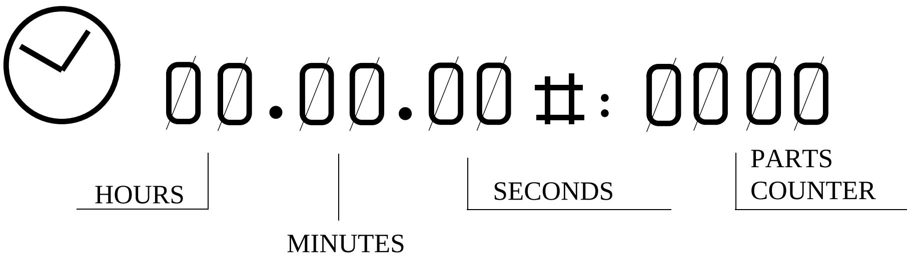
When the running of a program is interrupted or finished, the counting of the clock is also interrupted.

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 th**e 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 BLOC**K 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.

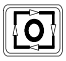
The value keyed in appears on the right of the **I**.

- 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:

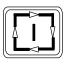
RETURN
AXES NOT POSITIONED

(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.

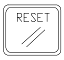
When the axes are in position, on the screen there will appear:

RETURN
AXES NOT POSITIONED
NONE

e) Press 

The cycle will continue normally**.**

3.1.7. CNC reset

When the  key is pressed on the top right-hand side of the screen, the blinking message RESET? will appear on the screen.

If the  key is pressed once more, the CNC goes back to its initial conditions.

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.

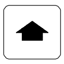
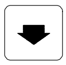
The CNC always displays the message with most priority, if there is more than one active message, the **+** sign will be highlighted (displayed in reverse video).

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(6).

3.2.3. Deletion of a complete program

Same as section 3.6.3. in EDITING mode (6).

3.2.4. Change of program number

Same as section 3.6.4. in EDITING mode (6).

3.2.5. Display and search of memorized subroutines

Same as section 3.6.5. in EDITING mode (6).

3.2.6. Selection of a program

Same as section 3.6.6. in EDITING mode (6).

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.

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

- 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 (6).

3.3.3. Deletion of a complete program

Same as section 3.6.3. in EDITING mode (6).

3.3.4. Change of program number

Same as section 3.6.4. in EDITING mode (6).

3.3.5. Display and search of memorized subroutines

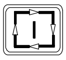
Same as section 3.6.5. in EDITING mode (6).

3.3.6. Selection of a program

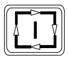
Same as section 3.6.6. in EDITING mode (6).

3.3.7. Creation of a program

Same as section 3.6.7. in EDITING mode (6) 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.

Atention:

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.

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 - RAPID MOVE
3 - 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 - 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 parameters P712, P713, P714, P724 are active too, acceleration/deceleration will be applied to **F0**, avoiding the generation of following errors.

3 - 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 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.

3.4.3. CNC reset

Same as paragraph 3.1.4.

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, T** and **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, the **S** value and the number of the active tool will appear on the screen in large characters.

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,Z** 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).

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.

<u>**Atention:**</u>

The indications which are made here for the X,Z axes, will have the treatment for the 3rd and 4th axes in machines which have them.

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.

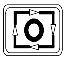
To cancel the preset, before pressing **ENTER**, operate **CL** as many times as characters to be deleted.

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:

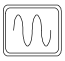
- Press  to stop the movement.

  or.

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

Atention:

On selecting the **JOG** operating mode the feedrate **F0** remains selected. If another feed is not entered later, the axes move at the % of **F** indicated by the switch on the front.

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

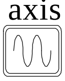
3.5.4.2. Incremental movement

- Front panel M.F.O. switch in the **JOG** zone.
- Press any of the **JOG** 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** remains selected. If another feed is not entered later, the axes move at the % of **F** indicated by the switch on the front.

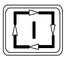
   Rapid feed of an axis in **JOG** mode can be obtained while pressing the **RAPID FEED**  key.

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.

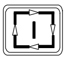
3.5.5. Entering F,S, M and T

The required values of **F,S, M** and **T** may be entered in this operating mode. The last three will depend on the value of the P603 parameter, bits 5,6,7.

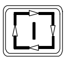
3.5.5.1. Entering an F value

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

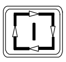
3.5.5.2. Entering an S value

- 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 

3.5.5.4. Entering an T value

- Press the **T** key
- Key in the required value (T2.2)
- Press 

3.5.6. Measurement and loading of the tool dimensions in the offset table.

Once the **JOG** mode has been chosen, the tool dimensions can be measured and introduced into the table, by using a part with known dimensions. For this, machine parameter P806 will be assigned a value of **0**. The sequence to be executed is as follows:

. Press the function key [TOOL MEASUREMENT]

. Press **X**.

. Key in the dimension of the part according to the **X** axis. This value will be in radii or diameters, depending on how the machine is working.

. Press **ENTER**.

. Press **Z**.

. Key in the part dimension according to the **Z** axis.

. Press **ENTER**.

. Key in the desired tool number (T2.2).

. Press **START**.

. Move the **X** axis with the manual controls, until the part is touched.

. Press **X**.

. Press [LOAD]. At that moment the new **X** dimension of the tool calculated by the control becomes active, due to which the dimension displayed on the **X** axis must be the same as the one introduced as the radius or diameter of the part.

. Move the **Z** axis with the manual controls, until the part is touched.

. Press **Z**.

. Press [LOAD]. At that moment the new **Z** dimension of the tool calculated by the control becomes active, due to which the dimension displayed on the **Z** axis must be the same as the one introduced as the radius or diameter of the part.

. If you want to do the same with another tool, it is necessary to begin once more by keying in the new tool (T2.2); the rest of the operation is the same as the first tool.

. To go on to work in the normal way in the **MANUAL** mode, the [TOOL MEASUREMENT] key must be pressed.

3.5.7. 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.

3.5.8. Change of measuring units

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

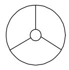
3.5.9. CNC Reset

Once the **JOG** mode is selected, when 

is pressed, the CNC returns to the initial conditions.

3.5.10. 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 highlighted 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.

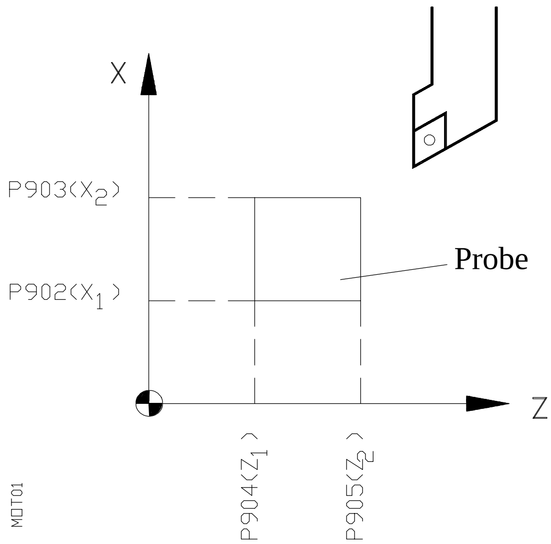
- Turn the M.F.O. switch to any other position or press the **STOP** key or keep the axis selector button pressed until the CRT stops displaying (blinking) the selected axis, if a FAGOR Handwheel (mod 100 P) is used.

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

As long as the machine parameter **P806** is assigned a value different from zero the CNC permits that, 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:

P902 minimum (X1) value according to X axis (in radii)
P903 maximum (X2) value according to X axis (in radii)
P904 minimum (Z1) value according to Z axis
P905 maximum (Z2) value according to Z axis



The sequence to be followed is:

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

2- Select the tool to be measured by keying in: **Txx.xx START**.

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

4- Press the key of the axis to be measured (X or Z).

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 P806.

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

7- Repeat from step 3 to measure and load the length of the tool in the other axis.

8 - Once the tool has been removed, repeat from point 2 for the measurement and loading of the tools.

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

The radius values of the **R** plate and the **F** shape factor of the tool are introduced manually by means of mode o operation 8 or by means of programming the **G50** function.

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

3.5.12. 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: **MKAI1** to lock the memory.
  MKAI0 to unlock the memory.

- Press **ENTER**.

Atention:

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. Deletion of a complete program

- Press [**PROG**ram **DIR**ectory]. A list of up to 14 programs in memory appears on the screen as well as the number of characters used and those remaining available.

- 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 P801 will be deleted.

> **Atention:**
>
> If there are more than 14 programs stored in memory, it may happen that the one to be deleted does not appear on the screen. In this case, by operating the keys ↑ / ↓ the various programs may be moved back and forth until the desired program is displayed.

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.

<u>Atention:</u>

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 memorized subroutines in the CNC memory

- By pressing [STANDARD SUBROUTINES DIRECTORY] or [PARAMETER SUBROUTINES 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.

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. Unassisted programming

Format of a block

(dimensions in millimeters)

N4 G2 X+/-4.3 Z+/-4.3 F5.4 S4 T2.2 M2 (in this order)

(dimensions in inches)

N4 G2 X+/-3.4 Z+/-3.4 F5.5 S4 T2.2 M2 (in this order)

Programming:

The CNC automatically numbers the blocks in multiples of 10. 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.
- Press the **M** key and key in the number of the auxiliary function wanted. Up to a maximum or 7 may be programmed.
- Finally it is possible to write a **COMMENT** which must be within brackets (COMMENT).
- 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.

3.6.7.2. 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.

3.6.7.3. 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**.

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.

3.6.7.4. 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, first the [**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 P605(6) 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 received 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? (N/Y)

  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 control’s memory.

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

- 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, the leadscrew error compensation table and the PLC user program, should this option be available.

- 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? (N/Y)

     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 in 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.

Atention:

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

```
T01   X —— . — Z —— . — F -
       R —— . — I — . —    K — . —
T02   X —— . — Z —— . — F -
       R —— . — I — . —    K — . —
T03   X —— . — Z —— . — F -
       R —— . — 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 **X**.

- Key in the value of the length of the tool.
  Max. value: +/- 8388.607 mm or +/-330.2599 inch.

- Press **Z**.

- Key in the value of the length of the tool.
  Max. value: +/- 8388.607 mm or +/-330.2599 inch.

- Press **F**.

- Key in the shape code (0-9) of the tool used.

- Press **R**.

- Key in the tool radius value
  Maximum value 1000.00 mm or +/-39.3700 inches.

- Press **I**.

- Key in the length correction value of the tool according to the **X** axis. This value must be given in diameters.
  Maximum value +/- 32.766 mm or +/-1.2900 inches.

- Press **K**.

- Key in the length correction value of the tool according to the **Z** axis. Maximum value +/-32.766 mm or +/-1.2900 inches.

- Press **ENTER**.

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 (X,Z,F,R,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.

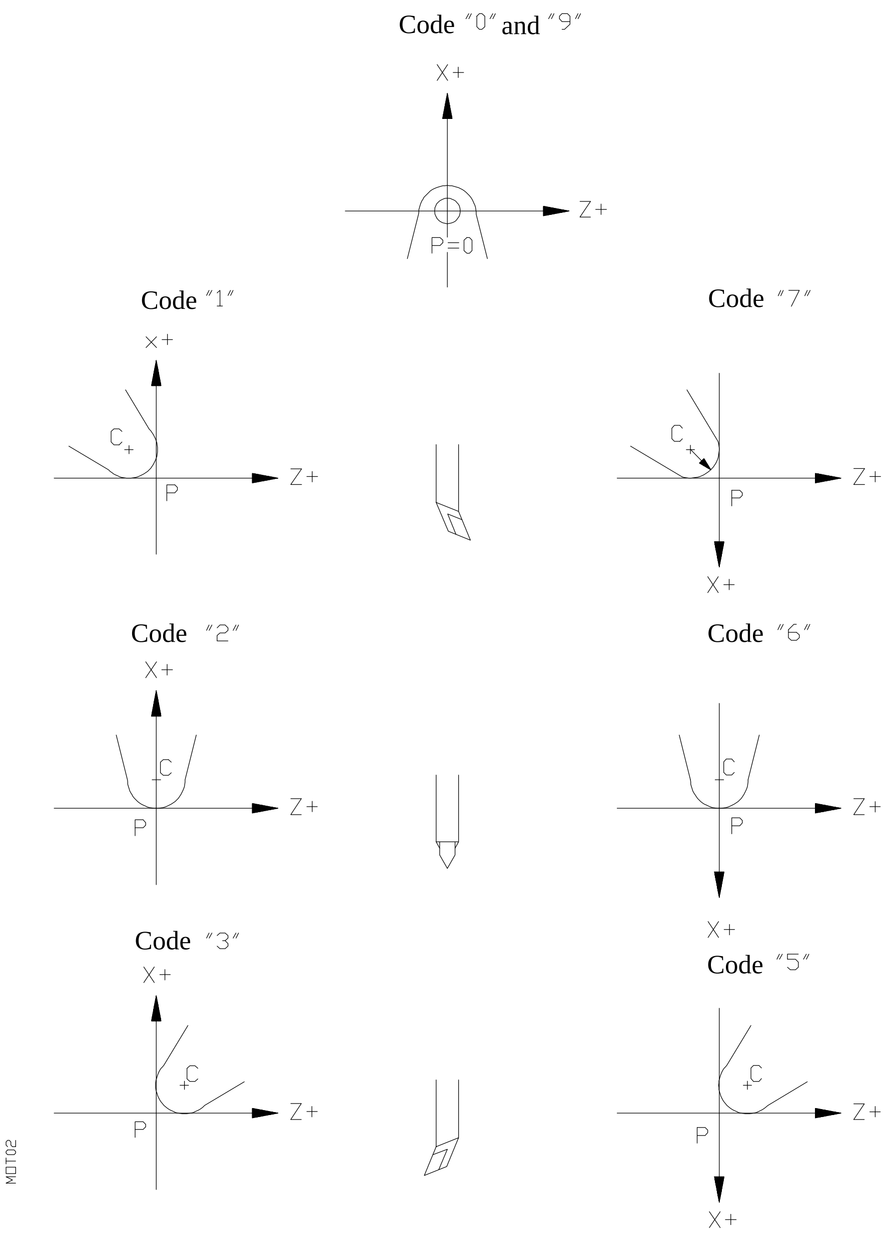
II) Dimensions already entered in memory

- 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.

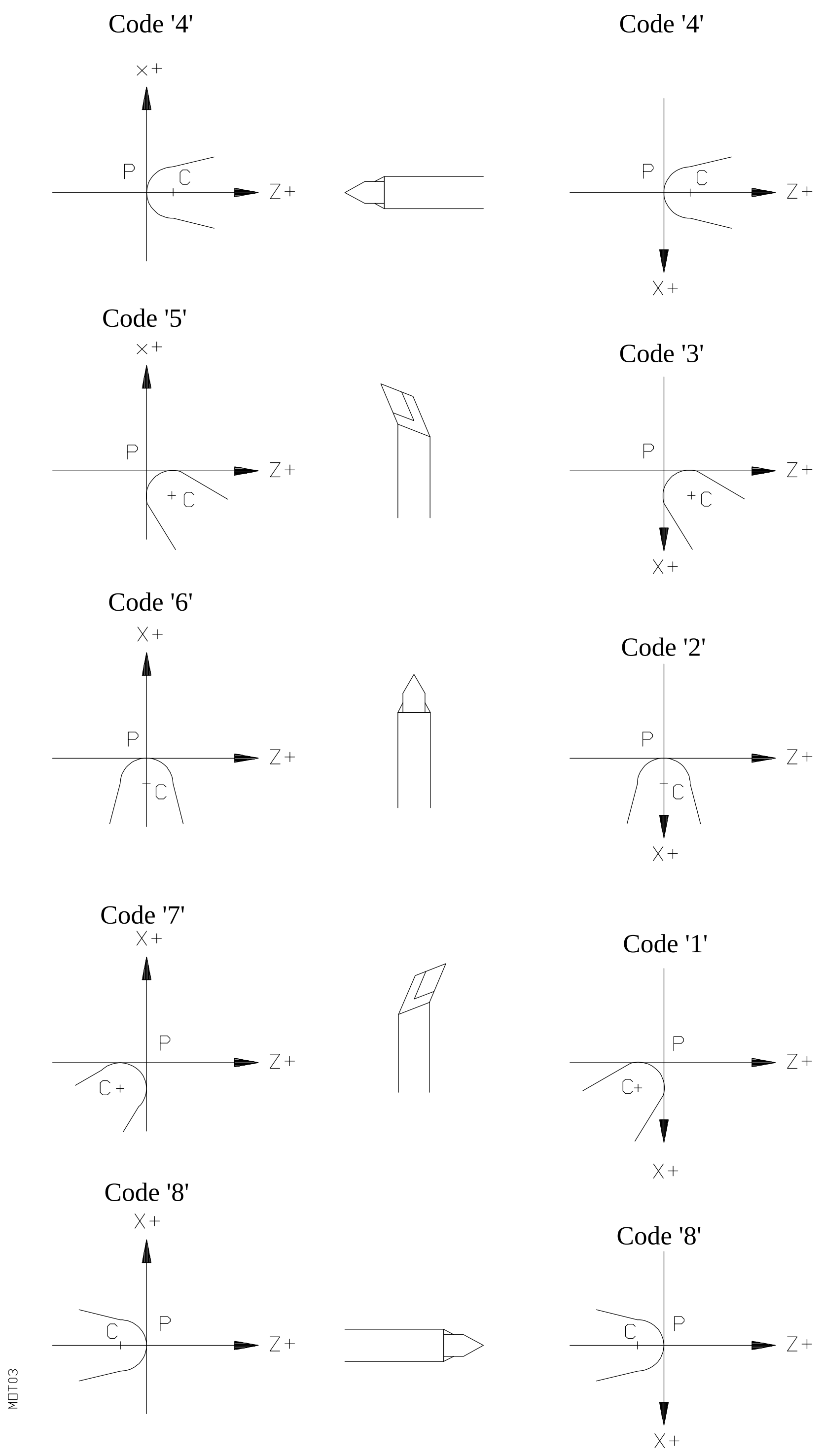
3.8.5. Change of measurement units

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

SHAPE CODES



P: Tool tip
C: Tool centre



Code '4'
Code '4'
Code '5'
Code '3'
Code '6'
Code '2'
Code '7'
Code '1'
Code '8'
Code '8'
X+
Z+
P
C
MDT03

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 X —— . — Z —— . —-
G54 X —— . — Z —— . —-
G55 X —— . — Z —— . —-
G56 X —— . — Z —— . —-
G57 X —— . — Z —— . —-
G58 X —— . — Z —— . —-
G59 X —— . — Z —— . —-

3.8.6.1. Read-out of zero offset table

- Key in the number of the zero offsets (G53/G59)
- Write the desired **X,Z** values
- Press **ENTER**

> **Attention:**
>
> The X,Z values refer to the machine zero point.

3.8.6.2. Entering zero offsets values

Operate as in the case of 3.8.4.

3.8.6.3. Modification of zero offset values

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,A,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 **TS** or **TG** 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-V] key.

Next, the values of the coordinates **X, Z** must be keyed in from the point at which it is required that it appear in the center of the screen and the value of the width we want it to represent. After keying in each value, the **ENTER** key must be pressed.

The definition of the display area must be made every time the CNC is switched on, if it is required to use the graphic representation feature.

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 TS)

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 * The range or the Constant Surface Speed has not 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 Canned cycle profile 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 G14, G15, G16, 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.

> Synchronization factor K of the synchronized tool too large.

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 More than one spindle range have been defined in the same block.

021 This error will be issued in the following cases:

> There is no block at the address defined by the parameter assigned to F18, F19, F20, F21, F22.
> The corresponding axis has not been defined in the addressed block

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

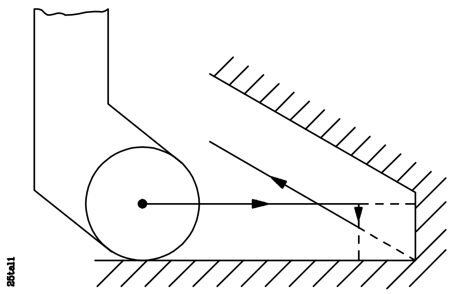
028 An attempt has been made to select a tool offset at the tool table or a non-existent external tool (the number of tools is set by machine parameter).

029 Too large a value assigned to a function.

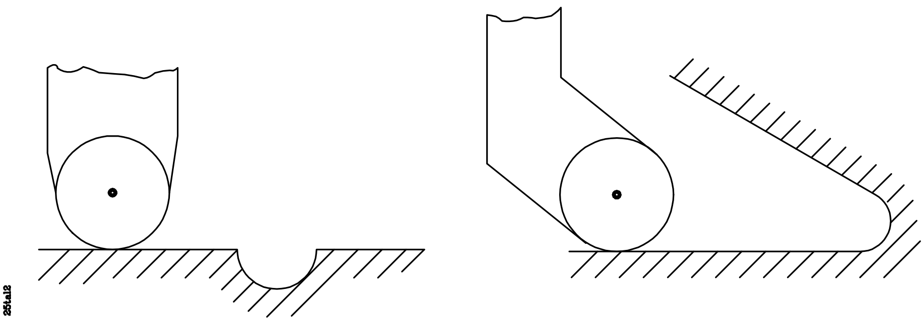
This error is often issued when programming an F value in mm/min (inch/min) and, then, switching to work in mm/rev (inch/rev) without changing the F value.

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 Z axis position Z-5000, if we want to move it to point Z5000, the CNC will issue error 33 when programming the block N10 Z5000 since the programmed move will be: Z5000 - Z-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 Z0          ; 5000 mm move
N10 Z5000       ; 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 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:

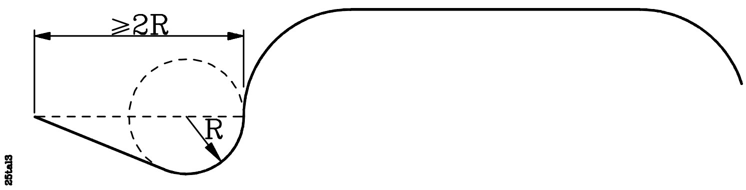
> More than 15 nesting levels when calling subroutines.

> 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:

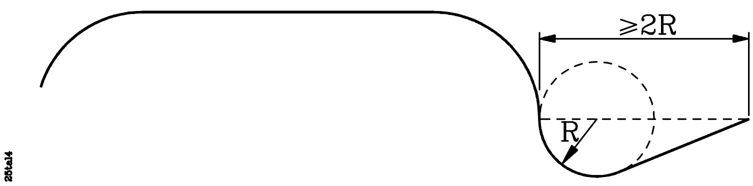
> There is no room to perform the tangential entry. A clearance of twice the rounding radius or greater is required.



> 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 Function M45 S programmed wrong (speed of the live tool).

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 Start or cancel tool radius compensation while in G02 or G03.

049 Chamfer programmed incorrectly.

050 G96 has been programmed while the S output is in BCD as set by machine parameter. (AC spindle).

051 * "C" axis programmed incorrectly

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

055 Parity error when reading or recording a cassette or a floppy disk.

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 tape.

058 Problems in floppy disk movement or sluggish tape movement.

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

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.

070 ** X axis following error.

071 ** Synchronized tool following error

072 ** Z axis following error.

073 ** 4th axis following error.

074 ** This error is issued in the following cases:

> 3rd axis following error

>"C" axis following error

075 ** Feedback error at connector A1.

076 ** Feedback error at connector A2.

077 ** Feedback error at connector A3.

078 ** Feedback error at connector A4.

079 ** Feedback error at connector A5.

081 ** 3rd axis travel limit overrun.

082 ** Parity error in 4th axis parameters. The CNC initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=1.

083 ** 4th axis travel limit overrun.

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 initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=1.

095 ** Parity error in general parameters. The CNC initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=1.

096 ** Parity error in Z axis parameters. The CNC initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=1.

097 ** Parity error in 3rd or "C" axis parameters. The CNC initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=1.

098 ** Parity error in X axis parameters. The CNC initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=1.

099 ** Parity error in M table. The CNC initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=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.

108 ** Error in Z axis leadscrew error compensation parameters. The CNC initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=1.

110 ** Error in X axis leadscrew error compensation parameters. The CNC initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=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 "P729".

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.

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: T, TG, TS

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

ERROR CODES

COMPARISON TABLE FOR LATHE MODEL FAGOR 8025/8030 CNCs

TECHNICAL DESCRIPTION

	T	TG	TS
INPUTS/OUTPUTS			
Feedback inputs.	**6**	**6**	**6**
Linear axes	**4**	**4**	**4**
Rotary axes	**2**	**2**	**2**
Spindle encoder	**1**	**1**	**1**
Electronic handwheel	**1**	**1**	**1**
Third axis as "C" axis			**x**
Synchronized tool			**x**
Probe input	**x**	**x**	**x**
Square-wave feedback signal multiplying factor, x2/x4	**x**	**x**	**x**
Sine-wave feedback signal multiplying factor, x2/x4/10/x20	**x**	**x**	**x**
Maximum counting resolution 0.001mm/0.001°/0.0001inch	**x**	**x**	**x**
Analog outputs (±10V) for axis servo drives	**4**	**4**	**4**
Spindle analog output (±10V)	**1**	**1**	**1**
Live tool	**1**	**1**	**1**
AXIS CONTROL			
Axes involved in linear interpolations	**3**	**3**	**3**
Axes involved in circular interpolations	**2**	**2**	**2**
Electronic threading	**x**	**x**	**x**
Spindle control	**x**	**x**	**x**
Software travel limits	**x**	**x**	**x**
Spindle orientation	**x**	**x**	**x**
PROGRAMMING			
Part Zero preset by user	**x**	**x**	**x**
Absolute/incremental programming	**x**	**x**	**x**
Programming in cartesian coordinates	**x**	**x**	**x**
Programming in polar coordinates	**x**	**x**	**x**
Programming by angle and cartesian coordinate	**x**	**x**	**x**
COMPENSATION			
Tool radius compensation	**x**	**x**	**x**
Tool length compensation	**x**	**x**	**x**
Leadscrew backlash compensation	**x**	**x**	**x**
Leadscrew error compensation	**x**	**x**	**x**
DISPLAY			
CNC text in Spanish, English, French, German and Italian	**x**	**x**	**x**
Display of execution time	**x**	**x**	**x**
Piece counter	**x**	**x**	**x**
Graphic movement display and part simulation		**x**	**x**
Tool tip position display	**x**	**x**	**x**
Geometric programming aide	**x**	**x**	**x**
COMMUNICATION WITH OTHER DEVICES			
Communication via RS232C	**x**	**x**	**x**
Communication via DNC	**x**	**x**	**x**
Communication via RS485 (FAGOR LAN)	**x**	**x**	**x**
ISO program loading from peripherals	**x**	**x**	**x**
OTHERS			
Parametric programming	**x**	**x**	**x**
Model digitizing			**x**
Possibility of an integrated PLC	**x**	**x**	**x**

PREPARATORY FUNCTIONS

	T	TG	TS
AXES AND COORDINATES SYSTEMS			
Part measuring units. Millimeters or inches (G70,G71)	x	x	x
Absolute/incremental programming (G90,G91)	x	x	x
Independent axis (G65)	x	x	x
REFERENCE SYSTEMS			
Machine reference (home) search (G74)	x	x	x
Coordinate preset (G92)	x	x	x
Zero offsets (G53...G59)	x	x	x
Polar origin offset (G93)	x	x	x
Store current part zero (G31)	x	x	x
Recover stored part zero (G32)	x	x	x
PREPARATORY FUNCTIONS			
Feedrate F	x	x	x
Feedrate in mm/min. or inches/min. (G94)	x	x	x
feedrate in mm/revolution or inches/revolution (G95)	x	x	x
Programmable feed-rate override (G49)	x	x	x
Spindle speed (S)	x	x	x
Spindle speed in rpm (G97)	x	x	x
Constant Surface Speed (G96)	x	x	x
S value limit when working at constant surface speed (G92)	x	x	x
Tool and tool offset selection (T)	x	x	x
Activate "C" axis in degrees (G14)			x
Main plane C-Z (G15)			x
Main plane C-X (G16)			x
AUXILIARY FUNCTIONS			
Program stop (M00)	x	x	x
Conditional program stop (M01)	x	x	x
End of program (M02)	x	x	x
End of program with return to first block (M30)	x	x	x
Clockwise spindle start (M03)	x	x	x
Counter-clockwise spindle start (M04)	x	x	x
Spindle stop (M05)	x	x	x
Spindle orientation (M19)	x	x	x
Spindle speed range change (M41, M42, M43, M44)	x	x	x
Tool change with M06	x	x	x
Live tool (M45 S)	x	x	x
Synchronized tool (M45 K)			x
PATH CONTROL			
Rapid traverse (G00)	x	x	x
Linear interpolation (G01)	x	x	x
Circular interpolation (G02,G03)	x	x	x
Circular interpolation with absolute center coordinates (G06)	x	x	x
Circular path tangent to previous path (G08)	x	x	x
Arc defined by three points (G09)	x	x	x
Tangential entry (G37)	x	x	x
Tangential exit (G38)	x	x	x
Controlled radius blend (G36)	x	x	x
Chamfer (G39)	x	x	x
Electronic threading (G33)	x	x	x
ADDITIONAL PREPARATORY FUNCTIONS			
Dwell (G04 K)	x	x	x
Round and square corner (G05, G07)	x	x	x
Scaling factor (G72)	x	x	x
Single block treatment (G47, G48)	x	x	x
User error display (G30)	x	x	x
Automatic block generation (G76)			x
Communication with FAGOR Local Area Network (G52)	x	x	x

	T	TG	TS
COMPENSATION			
Tool radius compensation (G40,G41,G42)	**x**	**x**	**x**
Loading of tool dimensions into internal tool table (G50, G51)	**x**	**x**	**x**
CANNED CYCLES			
Pattern repeat (G66)	**x**	**x**	**x**
Roughing along X (G68)	**x**	**x**	**x**
Roughing along Z (G69)	**x**	**x**	**x**
Straight section turning (G81)	**x**	**x**	**x**
Straight section facing (G82)	**x**	**x**	**x**
Deep hole drilling (G83)	**x**	**x**	**x**
Circular section turning (G84)	**x**	**x**	**x**
Circular section facing (G85)	**x**	**x**	**x**
Longitudinal threadcutting (G86)	**x**	**x**	**x**
Face threadcutting (G87)	**x**	**x**	**x**
Grooving along X (G88)	**x**	**x**	**x**
Grooving along Z (G89)	**x**	**x**	**x**
PROBING			
Probing (G75)	**x**	**x**	**x**
Tool calibration canned cycle (G75N0)			**x**
Probe calibration canned cycle (G75N1)			**x**
Part measuring canned cycle along X (G75N2)			**x**
Part measuring canned cycle along Z (G75N3)			**x**
Part measuring canned cycle with tool compensation along X (G75N4)			**x**
Part measuring canned cycle with tool compensation along Z (G75N5)			**x**
SUBROUTINES			
Number of standard subroutines	99	99	99
Definition of a standard subroutine (G22)	**x**	**x**	**x**
Call to a standard subroutine (G20)	**x**	**x**	**x**
Number of parametric subroutines	99	99	99
Definition of a parametric subroutine (G23)	**x**	**x**	**x**
Call to a parametric subroutine (G21)	**x**	**x**	**x**
End of standard or parametric subroutine (G24)	**x**	**x**	**x**
JUMP OR CALL FUNCTIONS			
Unconditional jump/call (G25)	**x**	**x**	**x**
Jump or call if zero (G26)	**x**	**x**	**x**
Jump or call if not zero (G27)	**x**	**x**	**x**
Jump or call if smaller (G28)	**x**	**x**	**x**
Jump or call if greater (G29)	**x**	**x**	**x**

NEW FEATURES AND MODIFICATIONS

Date: **March 1991** ***Software Version:*** **2.1 and newer**

FEATURE	MODIFIED MANUAL & SECTION	
The home searching direction is set by machine parameter P618(5,6,7,8)	Installation Manual	Section 4.7
The 2nd home searching feedrate is set by machine parameter P807...P810	Installation Manual	Section 4.7
New resolution values 1, 2, 5 and 10 for sine-wave feedback signals P619(1,2,3,4)	Installation Manual	Section 4.1
Access to PLCI registers from the CNC	Programming Manual	G52

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

FEATURE	MODIFIED MANUAL AND SECTION	
New function: F36. It takes the value of the selected tool number	Programming Manual	Chapter 13
G68 and G69 canned cycles modified. if P9=0 it runs another final roughing pass	Programming Manual	Chapter 13

Date: **September 1991** ***Software Version:*** **3.2 and newer**

FEATURE	MODIFIED MANUAL AND SECTION	
Subroutine associated with the T function	Installation Manual	Section 3.3.5
G68 and G69 canned cycles modified. P9 can now have a negative value	Programming Manual	Chapter 13

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

FEATURE	**MODIFIED MANUAL AND SECTION**	
Bell-shaped ACC./DEC.	Installation Manual	Section 4.8
It is now possible to enter the sign of the leadscrew backlash for each axis P620(1,2,3,4)	Installation Manual	Section 4.4
Independent axis movement execution	Programming Manual	G65
It is now possible to work at Constant Surface Speed in JOG mode P619(8)	Installation Manual	Section 3.3.9

Date: **July 1992** ***Software Version:*** **4.2 and newer**

FEATURE	**MODIFIED MANUAL AND SECTION**	
Synchronisation with independent axis P621(4)	Installation Manual	Section 3.3.10

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

FEATURE	**MODIFIED MANUAL AND SECTION**	
Linear & Bell-shaped acc./dec. ramp combination	Installation Manual	Section 4.8
Spindle acc/dec control. P811	Installation Manual	Section 5.
The subroutine associated with the tool is executed before the T function. P617(2)	Installation Manual	Section 3.3.5
G68 and G69 cycles modified. If P10 <> 0, it runs a final roughing pass before the finishing pass	Programming Manual	Chapter 13
When having only one spindle range, if G96 is executed without any range being selected, the CNC will automatically select it.	Programming Manual	Chapter 6
8030 CNC with VGA Monitor	Installation Manual	Chapter 1

Date: **March1995** ***Software Version:*** **5.3 and newer**

FEATURE	**MODIFIED MANUAL AND SECTION**	
Management of semi-absolute feedback devices (with coded Io)	Installation Manual	Sections 4.7 & 6.5.
Spindle inhibit by PLC	Installation Manual	Section 3.3.10
Handwheel managed by PLC	Installation Manual	Section 3.3.3
Simulation of the "rapid JOG" key from PLC	PLCI Manual	
Initialization of machine parameters in case of memory loss.		

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.

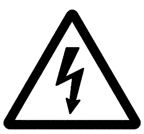
Safety symbols

Symbols which may appear on the manual

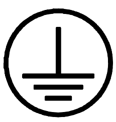
WARNING. symbol
It has an associated text indicating those actions or operations may hurt people or damage products.

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 T CNC

8025 T 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.
8025 T CNC USER Manual	Is directed to the end user or CNC operator. It contains 2 manuals: Operating Manual — describing how to operate the CNC. Programming Manual — describing how to program the CNC. 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:

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:

OPERATING MODE 2 - PLAY BACK
OPERATING MODE 3 - TEACH IN
OPERATING MODE 6 - EDITING
OPERATING MODE 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). **RETURN** 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 **ESCAPE (ESC)** or **END OF TAPE (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 ( ).

The maximum number of characters which can be written in a comment is 43, 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 a computer .

. Execution of an 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 3.50" flexible diskette is an application for the connection of FAGOR numerical controls to a PC or 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
4th, 3rd,X,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.

If the machine has a **3rd** and/or **4th** axis (the denomination of which is defined by the machine parameter) it is possible to program them both in rapid positioning **G00** and in Linear Interpolation **G01**, with a maximum of 3 axes in the same block and in the following order: **4th, 3rd X Z.**

For example: N4 G1 W50 X12 Z35 F550

However, if the **3rd** axis is of the **C** axis type, it is also possible to program circular interpolations **G02/G03** with it, as long as the **G15** or **G16** functions are activated.

Atention:

In the different programming formats throughout the manual, the 4th and 3rd axes will be indicated as such, although depending on the type of machine, their display and programming will be as follows:

. The 4th axis can be W or Y
. The 3rd axis can be W, Y or C

3. PROGRAM FORMAT

The CNC can be programmed in millimeters or in inches.

Metric format (in mm):

P(%)5 N4 G2 X+/-4.3 Z+/-4.3 F5.5 S4 T2.2 M2

Format in inches:

P(%)5 N4 G2 X+/-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 A positive value of up to two digits to the left and two to the right of the decimal point may be programmed.

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 ZP13 FP10 S1500 TP4.P4 MP2

The CNC has 255 arithmetic parameters (P00/P254). (See chapter 13 of this manual regarding 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.

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 any program execution, the CNC reads **4** blocks ahead of 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) N4 SPECIAL CONDITIONAL BLOCK

If next to the block number **N4** (0-999), 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 (enabling input for conditional blocks), 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 IN 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)	G14	: Activation of **C** axis in degrees
(Modal)	G15	: Machining the cylindrical surface of a part
(Modal)	G16	: Machining the surface of a part face
	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 (controlled corner rounding)
	G37	: Tangential approach
	G38	: Tangential exit
	G39	: Chamfering
(Modal)	G40*	: Cancellation of radius compensation
(Modal)	G41	: Left hand radius compensation
(Modal)	G42	: Right hand radius compensation
(Modal)	G47	: Single block treatment
(Modal)	G48*	: Cancellation of single block treatment
	G50	: Loading of the values in the tool offset table
	G51	: Correction of the dimensions of the tool in use
	G52	: Communication with FAGOR LOCAL AREA NETWORK
(Modal)	G53-G59	: Zero offsets
	G64	: Multiple arc pattern machining cycle
	G65	: Independent axis execution
	G66	: Pattern repeat (roughing canned cycle following part shape)
	G68	: Roughing canned cycle (X)
	G69	: Roughing canned cycle (Z)
(Modal)	G70	: Programming in inches
(Modal)	G71	: Programming in millimeters
(Modal)	G72	: Scaling factor
	G74	: Automatic search for machine reference
	G75	: Probing
	G75 N2	: Probing canned cycles
	G76	: Automatic block generation
	G81	: Canned turning cycle with straight sections
	G82	: Canned facing cycle with straight sections
	G83	: Deep hole drilling
	G84	: Turning with arcs
	G85	: Facing with arcs
	G86	: Longitudinal threadcutting cycle
	G87	: Face threadcutting cycle
	G88	: Grooving cycle (X)
	G89	: Grooving cycle (Z)
(Modal)	G90*	: Programming of absolute coordinates
(Modal)	G91	: Programming of incremental coordinates
	G92	: Preselection of coordinates and setting of max. **S** value
	G93	: Preselection of polar origin
(Modal)	G94	: Feedrate **F** in mm/min (inch/min.)
(Modal)	G95*	: Feedrate **F** in mm/rev (inch/rev.)
(Modal)	G96	: Speed **S** in m/min (feet/min.) (Constant surface speed)
(Modal)	G97*	: Speed **S** in rev/min.

Functions **G14, G15, G16, G75 N2** and **G76** are only available on the model TS CNC 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 E**MERGENCY** 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 **G14, G15, G16, 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 P607(3) machine-parameter.

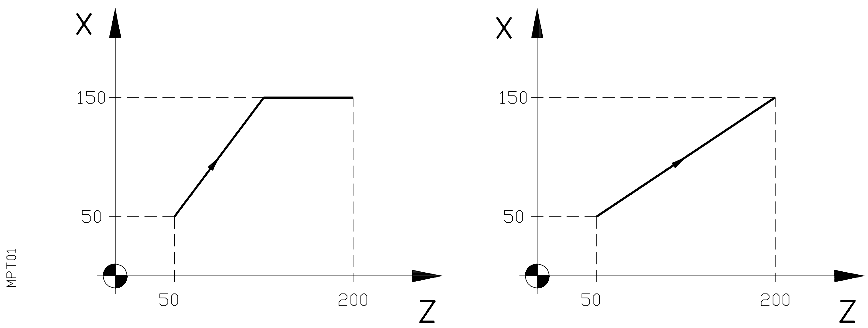
a) G00 path not controlled. P607(3)=0

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. P607(3)=1

When several axes move simultaneously, the resulting path is a straight line between the initial and the final point. Feed will be set by the slowest axis.

a) P607(3)=0 b) P607(3)=1



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.

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%.

The **G00** code freezes the tool radius offset (G41, G42). I.e., when it is working with **G41** or **G42** and **G00** is programmed, the radius offset remains ineffective until **G01, G02** or **G03** is programmed again.

The code **G00** is modal and incompatible with G01,G02,G03 and G33. G00 function can be programmed with **G** or **G0**.

6.2.2. G01. Linear interpolation

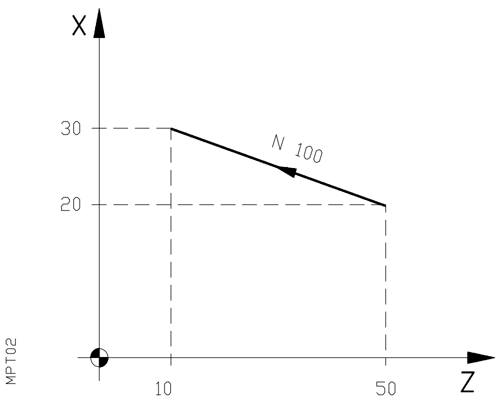
The movements programmed after **G01** are performed in a straight line at the feedrate **F** programmed.

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**.

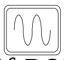
Example:

Programming the **X** axis in diameters. Initial point X40 Z50.



N100 G90 G01 X60 Z10 F300

The knob on the front panel of the CNC can be used to vary the programmed feedrate **F** between 0% and 120% or between 0% and 100%, according to parameter P600(3).

If, during a **G01** movement, the  key is pressed, the movement will be performed at twice the programmed feedrate if P600(3) is zero.

Function **G01** is modal and incompatible with **G00,G02,G03** and **G33**.

Function **G01** can be programmed as **G1**.

When the CNC is turned on, after executing **M02/M30**, after an **EMERGENCY** or after a **RESET**, the CNC assumes the **G01** code.

6.2.3. G02/G03. Circular interpolation

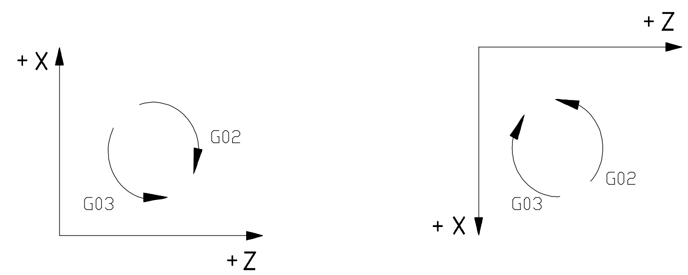
G02: Clockwise circular interpolation.
G03: Counter-clockwise circular interpolation.

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).

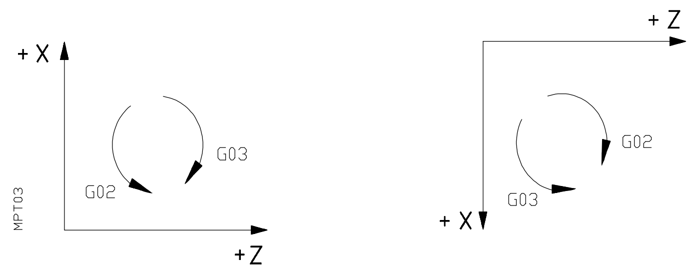
a) Parameter P600(1)=0

Machine axis direction



b) Parameter P600(1)=1

Machine axis direction



Functions G02/G03 are modal and incompatible both with one another and with G00,G01 and G33.

Functions G74,G75, or any canned cycle cancel G02/G03 functions.

Functions G02/G03 can be programmed as G2/G3.

The block format to program a circular interpolation with cartesian coordinates is:

N4 G02 (G03) X+/-4.3 Z+/-4.3 I+/-4.3 K+/-4.3

N4 : Block number
G02 (G03) : It defines the interpolation
X+/-4.3 : Coordinate value of the arc's final point along the **X** axis.
Z+/-4.3 : Coordinate value of the arc's final point along the **Z** axis.
I+/-4.3 : Distance from the arc's starting point to the center along **X** axis.
K+/-4.3 : Distance from the arc's starting point to the center along **Z** axis.

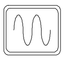
I,K Must be programmed with a sign. They must be programmed even when their value is 0.

The block format to program a circular interpolation with polar coordinates is:

N4 G02 (G03) A+/-3.3 I+/-4.3 K+/-4.3

N4 : Block number
G02 (G03) : It defines the interpolation
A+/-3.3 : Angle from the polar arc center to the arc's final point.
I+/-4.3 : Distance from the starting point to the arc's center along **X** axis.
K+/-4.3 : Distance from the starting point to the arc's center along **Z** axis.

When a circular interpolation is programmed in **G02** or **G03**, the arc's center is taken as the new polar origin. Even when the **X** axis is programmed in diameters, **I** is always programmed in radius.

If, during a **G02/G03** movement, the  key is pressed, the movement will be performed at twice the programmed feedrate if P600(3) is zero.

6.2.3.1. Circular interpolation in cartesian coordinates by programming the radius of the arc

The format is as follows:

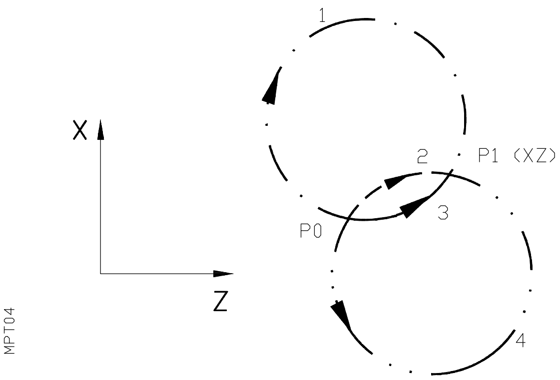
In mm : G02(G03) X+/-4.3 Z+/-4.3 R+/-4.3
In inches : G02(G03) X+/-3.4 Z+/-3.4 R+/-3.4

G02(G03) being the function which defines the circular interpolation direction.

X coordinate value of the arc's final point along X axis.
Z coordinate value of the arc's final point along Z axis.
R= Arc's radius.

That means that the circular interpolation can be programmed with its final point plus the radius instead of the center (I,K) coordinates.

If the arc is smaller than 180º, the radius will be programmed with positive sign and if it is bigger than 180º, the sign will be negative.



If **P0** is the starting point and **P1** is the final point of the arc, for a same value of **R**, there are four different arcs which pass through both points.

By combining the direction (G02/G03) and the sign of **R**(+/-) the required arc is identified. In this way, the format of the programming of the drawing's arcs is as follows:

Arc 1 G02 X Z R-
Arc 2 G02 X Z R+
Arc 3 G03 X Z R+
Arc 4 G03 X Z R-

XZ being the final point in cartesian coordinates.

Atention:

If an entire circle is programmed by the radius programming, the CNC will display error 47, as there are infinite solutions.

6.2.3.2. G06. Circular interpolation with absolute center coordinates

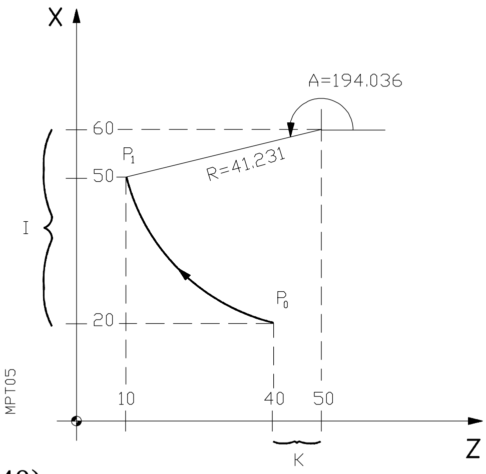
By adding **G06** in a block containing a circular interpolation, the coordinates for the center of the arc (I,K) can be given in absolute values. That is, referred to the part's datum point instead of being referred to the arc's starting point.

G06 is not **MODAL**, therefore, it must be programmed every time absolute center coordinates are to be used.

When programming in this way, the value of **I** must be either in diameters or radius depending on the setting of machine parameter **P11**.

Examples: Let us suppose that programming is in absolute coordinate values (G90) and the X axis one is in diameters. The arcs may be programmed in the following 4 ways:

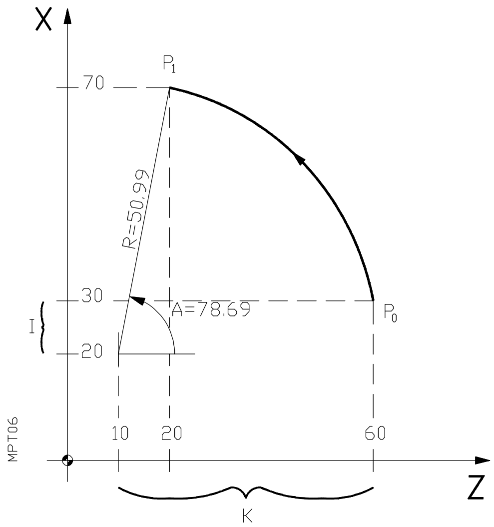
1.



Starting point P0 (X40 Z40)

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

2.



Starting point P0 (X60 Z60)

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

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**.

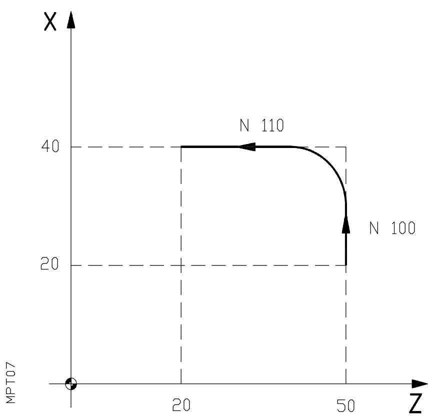
6.4. TRANSITION BETWEEN BLOCKS

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.

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: X in diameters. The starting point is X40 Z50



N100 G90 G01 G05 X80
N110 Z20

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 the theoretical and actual profiles.

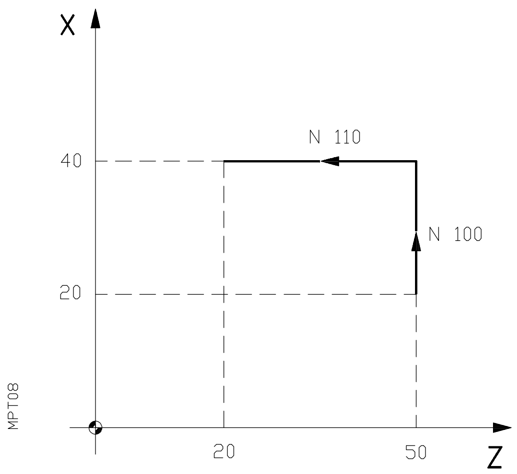
Function **G05** is modal and incompatible with **G07**.

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 programmed in the previous block has been reached.

Example: X in diameters. The starting point is X40 Z50



N100 G90 G01 G07 X80
N110 Z20

The theoretical and actual profiles coincide. Function **G07** is modal and incompatible with **G05**. Function **G07** can be programmed as **G7**. The CNC assumes function **G07** or **G05** on power **ON** and after **M02,M30,EMERGENCY** or **RESET** depending on the value given to machine parameter P607(8).

. P607(8) = 0, it assumes G07.
. P607(8) = 1, it assumes G05.

6.5. G08. CIRCULAR PATH TANGENT TO PREVIOUS PATH

A circular path tangent to the previous path can be programmed by means of **G08**. Center coordinates (I,K) are not required.

Format with cartesian coordinates:

N4 G08 X+/-4.3 Z+/-4.3 in mm.
N4 G08 X+/-3.4 Z+/-3.4 in inches.

N4 : Block number

G08 : Code defining circular interpolation tangent to previous path.

X+/-4.3 : Coordinate values of the arc's final point.
X+/-3.4
Z+/-4.3 : Coordinate values of the arc's final point.
Z+/-3.4

Format with polar coordinates:

N4 G08 R+/-4.3 A+/-4.3 in mm.
N4 G08 R+/-3.4 A+/-4.3 in inches

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.
R+/-3.4

A+/-4.3 : Angle (referred to polar origin) of the arc's final point.

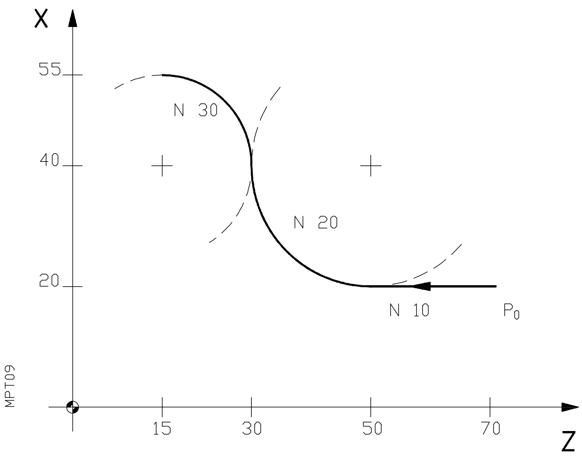
Example: **X** in diameters

The starting point being X40 Z70, the programming of the following path is described.

- Straight line
- Arc tangent to the straight line
- Arc tangent to the previous arc

Its programming may be:

N110 G90 G01 Z50
N120 G08 X80 Z30
N130 G08 X110 Z15



The arcs being tangent, there is no need of programming the center coordinates (I,K).

When G08 is not used, the programming will be:

N110 G90 G01   Z50
N120 G02 X80   Z30 I20 K0
N130 G03 X110 Z15 I0   K-15

The function **G08** is not modal. It can be used every time an arc tangent to the previous path is to be executed.

The previous path may be either a straight line or an arc.

The function **G08** replaces **G02** and **G03** only in the block in which it is written.

Atention:

A complete circle cannot be executed by means of **G08**, as there are infinite solutions. The CNC will display the error code **47**.

6.6. G09. ARC PROGRAMMED BY THREE POINTS

By means of the **G09** function a circular path (arc) may be defined, by programming the end point and an intermediate point (the initial point of the arc is the starting point of the movement).

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 (XZ plane)

N4 G09 X+/-4.3 Z+/-4.3 I+/-4.3 K+/-4.3

N4 : Block number.

G09 : Code identifying 3 point arc definition.

X+/-4.3 : X value of the arc's final point.

Z+/-4.3 : Z value of the arc's final point.

I+/-4.3 : X value of the intermediate point.

K+/-4.3 : Z value of the arc's intermediate point

Polar coordinates (XZ plane)

N4 G09 R+/-4.3 A+/-4.3 I+/-4.3 K+/-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.

K+/-4.3 : Z value of the arc's intermediate point

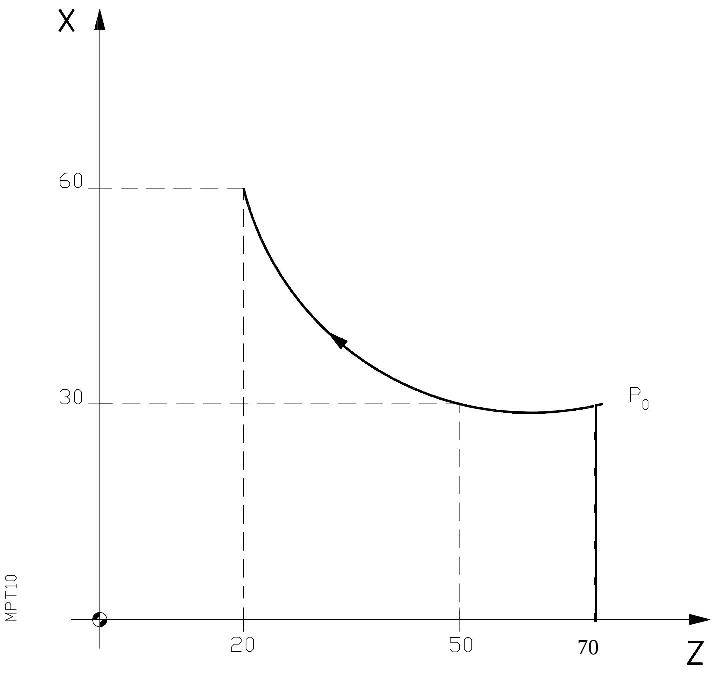
The intermediate point must always be programmed in cartesian coordinates.

Example:

Programming of the X axis is in diameters.

Let us suppose that the starting point is P0(X60 Z70) and the end point of the arc is X120 Z20). The program block to define this arc will be:

N4 G09 X120 Z20 I60 K50.



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. G14,G15,G16. C AXIS PROGRAMMING

These functions are only available on the TS model.

. G14. Activate **C** axis in degrees.

. G15. Machining of the cylindrical surface of the part (main plane C,Z)

. G16. Machining of the face of the part (Main plane C,X)

Once the typical turning operations are completed, other operations, like the milling of the cylindrical surface and/or the face of the part, are necessary.

To avoid down-time which mean changing and clamping the part from one machine to another the CNC can control the machine's main leadscrew (C axis).

In this way and by means of using a live tool, for example, a milling tool, it is possible to machine a cylindrical or front surface of the part by carrying out linear (G1) and rapid positioning moves (G00).

G14. ACTIVATE THE C AXIS IN DEGREES

General considerations:

. Programming **G14**, the positioning of the **C** axis can be controlled if machine-parameter P613(5)=1.

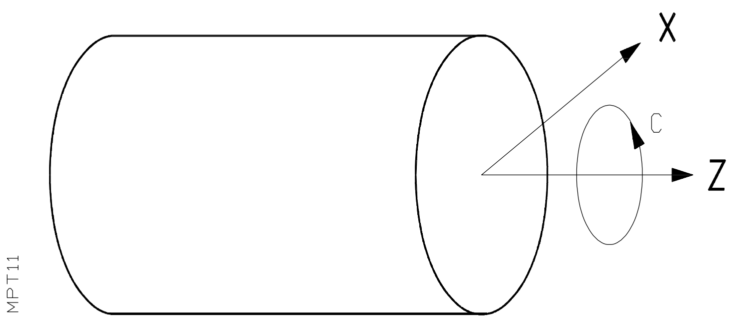
. **G14** must be programmed alone in a block.

. When the **C** axis is activated by means of **G14**, the CNC executes automatically a machine-reference-point search for that axis

. When **G14** is active, **G00** and **G01** may be programmed between the **C,X,Z** axes.

. When programming **G14, G95** and **G96** are cancelled.

. When **G14** is activated, **M3** or **M4** must be programmed to return to regular turning operation.



The **C** axis movement must be programmed in degrees and the feedrate **F4** in degrees/ minute. The programming format is the following:

In millimeters : N4 C+/-4.3 X+/-4.3 Z+/-4.3
In inches : N4 C+/-4.3 X+/-3.4 Z+/-3.4

When being **G14** activated, the following block is executed:

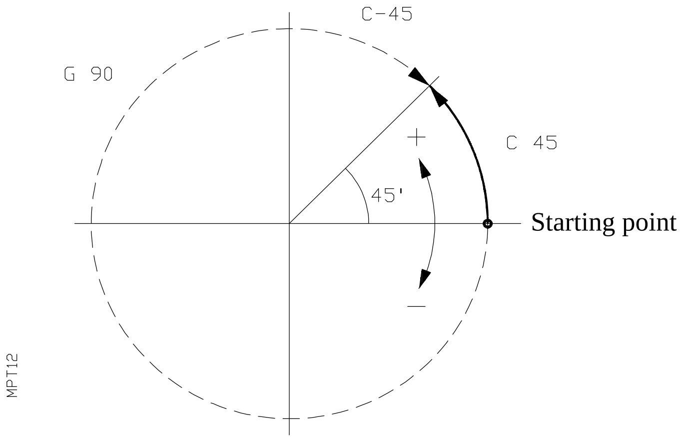
N4 G91 G01 C720 F500

The **C** axis will rotate two full revolutions resetting the count at every revolution and at a feedrate of 500 degrees/minute.

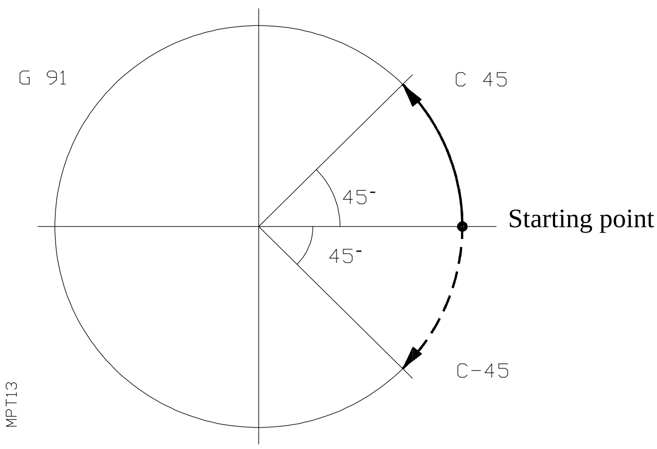
Also possible:

N4 G91 G01 C720° X100 F500
or
N4 G91 G01 C720° X100 Z100 F500
or
N4 G91 G01 X100 Z100 F500
or
N4 G91 G01 C002° X100 Z100 F500

When using **G90** with this axis, the sign of the programmed value indicates the rotating direction of the axis; so, if the same value is programmed with two different signs, the final point reached will be the same, but the rotation will be in opposite directions.



But when working with G91, the values will be incremental from the previous point and the programming will be similar to the one for a linear axis, except in degrees.



G15. MACHINING OF THE CYLINDRICAL SURFACE OF THE PART (plane C Z)

G16. MACHINING OF THE FACE OF THE PART (main plane C X)

General considerations for the programming of both functions:

. Either **G15** or **G16** must be programmed alone in the block.

. By programming either **G15** or **G16**, the tool radius compensation **G41/G42** are cancelled.

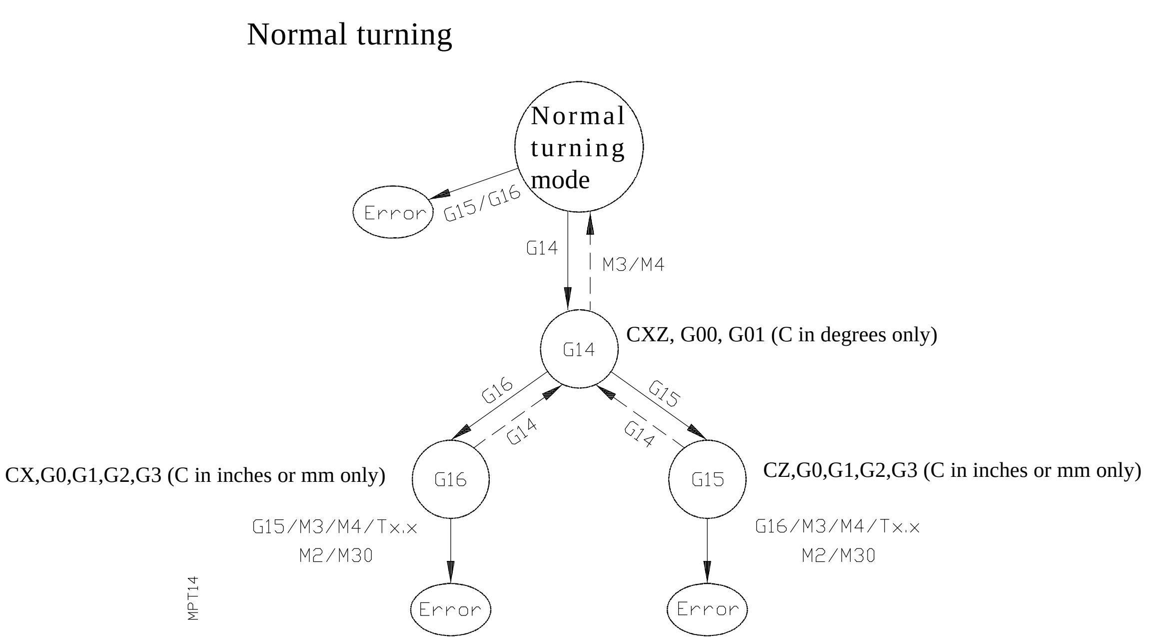
. **G15** and **G16** cancel functions **G95** and **G96**.

. **G14** must be active when programming either **G15** or **G16**; otherwise, the CNC will display error 51.

. When either **G15** or **G16** are activated, no tool (Txx.xx) programming is possible.

. Linear (G01) interpolations and rapid positioning moves (G00) can be carried out both in cartesian and polar coordinates.

. To cancel **G15** or **G16**, program **G14**.



The treatment of the **C** axis in making a program for machining cylindrical and face surfaces is similar to a linear axis. Therefore, with functions **G15** or **G16** active, C axis movements are programmed in millimeters or inches and the feed velocity (F4) in millimeters/minute or 0.1 inches/minute in accordance with the measurement system used. Programming is made as if it were a milling machine, the **C** axis coordinates being programmed in millimeters or inches on the surface of the part, are calculated and converted into degrees by the CNC to be able to do the machining.

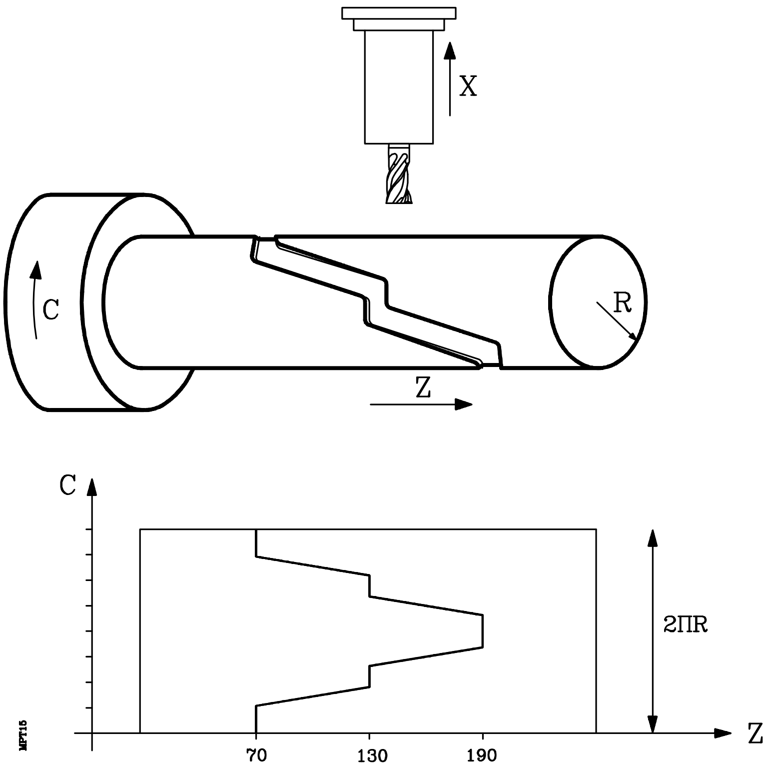
G15. MACHINING ON THE CYLINDRICAL SURFACE OF THE PART (main plane CZ)

When **G15** is programmed, in order to convert the programmed values from inches or mm into degrees, the CNC will assume as radius the distance from the tip of the tool to the rotation center line (X0).

The origin point of the defined plane is the one corresponding to the C axis machine-reference point.

C Z and X can be programmed simultaneously.

Example:



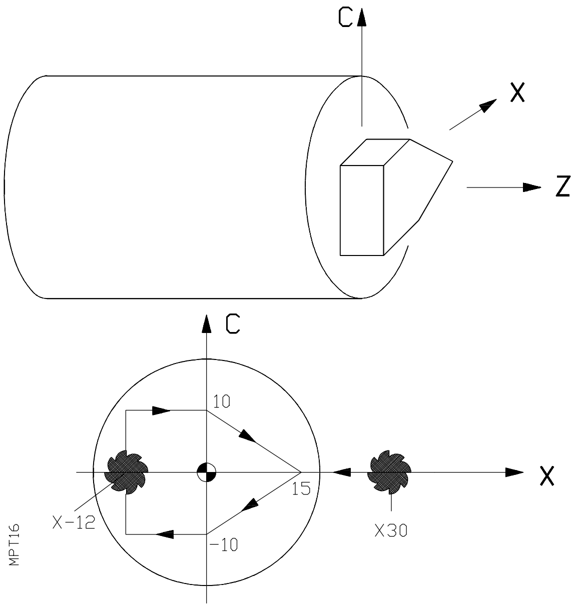

```
G14                 ;Activate the "C" axis in degrees
G15                 ;Machine the cylindrical surface
G90 G01 C0 Z70      ;Position at starting point
G91 X-4             ;Penetrate
G90 C15.708
C31.416 Z130
C39.270
C54.978 Z190
C70.686
C86.394 Z130
C94.248
C109.956 Z70
C125.664
G91 X4              ;Withdraw
M30
```

G16. MACHINING OF THE FACE OF THE PART (main plane C X)

It should be borne in mind that when the **G16** function is active, the coordinates of the axes must be programmed like a milling machine, i.e., the machine parameter **P11** will not be borne in mind where it is indicated, if the **X** axis is programmed in Radii or Diameters.

Observe in the example that the **X** coordinate of the N400 block; (X60) and that of the N490 (X30) block corresponding to the same point.
C Z and X can be programmed simultaneously.

Example: Let's suppose that the programming of the **X** axis is in diameters.

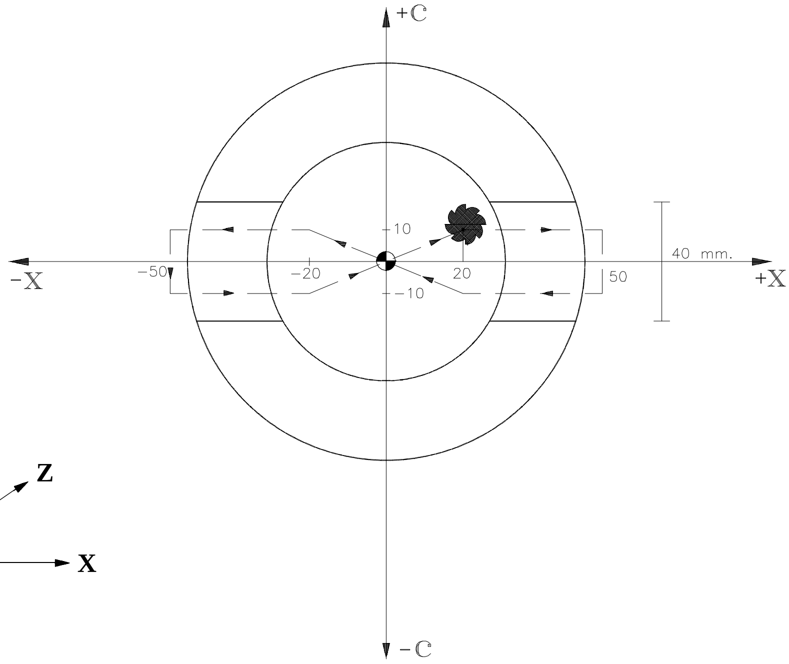


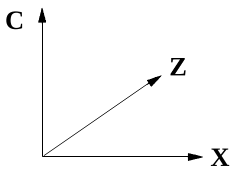
N ——
N400 G0 X60 Z-2
N410 G14
N420 G16
N430 G1 G90 X15 F1000
N440 C-10 X0
N450 X-12
N460 C10
N470 X0
N480 C0 X15
N490 X30
N500 G14
N510 G0 X100 Z200
N ——

PROGRAMMING THE PREVIOUS PATH

Another example of **C** axis programming in G16 (main plane CX)

N10 G14
N20 G0 G90 X0 Z2
N30 Z -9.5
N40 G16
N50 G1 G91 C10 X20 F500
N60 X30
N70 C-20
N80 X-30
N90 G90 C10 X-20 F0
N100 G91 X-30 F500
N110 C-20
N120 X30
N130 G90 G0 X0 F0
N140 G14
N150 G90 G0 Z100
N160 M30





6.8. G25. UNCONDITIONAL JUMP/CALL

The function **G25** can be used to jump to another block of the current program.It is not possible to program more information in the same block as the **G25** function is programmed. 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.

Format 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. Then, it executes the section of the program between the mentioned block and the one identified between the two decimal points as many times as set by the last digit. This digit may have a value between 0 and 99, unless it is programmed by 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 after 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 the block **30**.

Functions **G26,G27,G28,G29** and **G30** (conditional jumps/calls) will be described in the corresponding Chapter of this manual: PARAMETRIC PROGRAMMING. OPERATIONS WITH PARAMETERS.

6.9. G31-G32. STORAGE AND RETRIEVAL OF PART PROGRAM'S DATUM POINT

G31: Store current program's datum point.
G32: Retrieve datum point stored by G31.

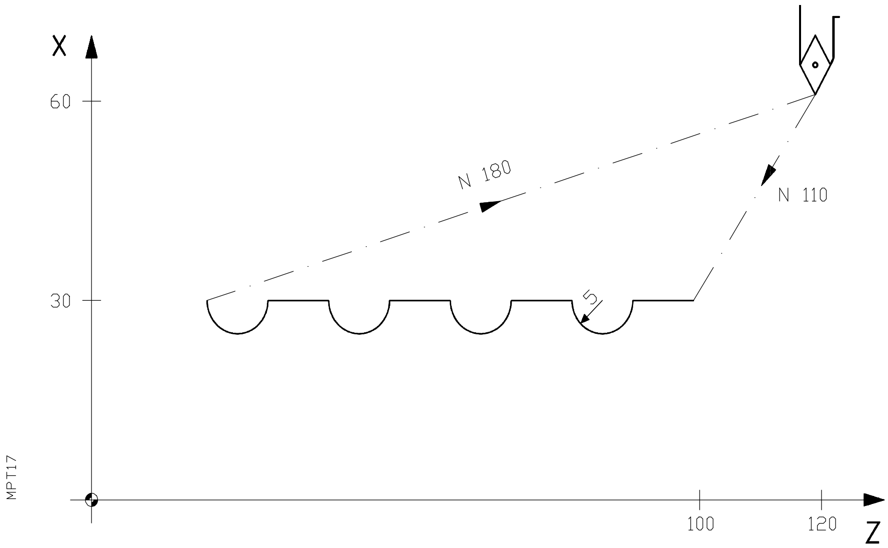
By means of the **G3**1 function, it is possible at any time to store the zero point which we are working with and recover it later by means of the **G32** function.

This feature is intended to simplify the operation with multi- datum part programs. A datum point can be stored any time and later retrieved by **G32**. Meantime, different datum points can be used by means of **G92** or **G53-G59**. 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 : Keep the current coordinate origin
G32 : Recover the coordinate origin kept by **G31**

Example:



Programming of axis X in diameters. Starting point X120 Z120.

N110 X60 Z100....................... (Approach to the part)
N120 G31................................. (Keep the origin coordinates)
N130 G92 X0 Z0 ....(Zero offset)
N140 G01 X0 Z-10 .... (Machining)
N150 G02 X0 Z-20 R5 ....(Machining)
N160 G25 N130.150.3 .... (Machining)
N170 G32 .... (Recover the initial origin)
N180 G00 X120 Z120 .... (Return to the starting point)

6.10. G33. THREADCUTTING

Longitudinal frontal and tapered threads can be cut using **G33** function. To apply this function it is necessary for the machine to have a rotary encoder in the spindle. **G33** is modal, i.e., once programmed it is maintained. It is cancelled by **G00, G01, G02, G03, M02, M30, EMERGENCY** or **RESET**.

Longitudinal thread

It can be programmed by means of:

N4 G33 Z+/-4.3 K3.4 (mm)
Z+/-3.4 K2.4 (inches)

where:

N4	: Block number
G33	: Threadcutting code
Z+/-4.3 (Z+/-3.4)	: Final coordinate of the thread along Z axis
K3.4 (K2.4)	: Thread pitch along Z axis

The Z value will be absolute or incremental depending on whether **G90** or **G91** has been programmed. If the function **G33** is active, the **F** feedrate speed cannot be altered by turning the **FEEDRATE** knob, whose value will be frozen at 100%.

Frontal thread (scroll)

It can programmed as follows:

N4 G33 X+/-4.3 I3.4 (mm)
X+/-3.4 I2.4 (inches)

N4	: Block number
G33	: Threadcutting code
X+/-4.3 (X+/-3.4)	: Final coordinate of the thread along **X** axis
I3.4 (I2.4)	: Thread pitch along **X** axis

The X value will be absolute or incremental depending on whether G90 or G91 has been programmed.

Tapered thread

It can be programmed as follows:

N4 G33 X+/-4.3 Z+/-4.3 I3.4 K3.4 (mm)
X+/-3.4   Z+/-3.4 I2.4 K2.4 (inches)

N4	: Block number
G33	: Threadcutting code
X+/-4.3 (X+/-3.4)	: Final coordinate of the thread along X axis
Z+/-4.3 (Z+/-3.4)	: Final coordinate of the thread along Z axis
I3.4 (I2.4)	: Thread pitch along X axis
K3.4 (K2.4)	: Thread pitch along Z axis

The **X** and **Z** values will be absolute or incremental depending on whether **G90** or **G91** has been programmed.

Only one pitch value (**I,K**) need be programmed. The CNC will calculate the other one. Thus:

N4 G33 X+/-4.3 Z+/-4.3 I3.4 (mm)
X+/-3.4   Z+/-3.4 I2.4 (inches)

or,

N4 G33 X+/-4.3 Z+/-4.3 K3.4 (mm)
X+/-3.4 Z+/-3.4 K2.4 (inches)

Can be programmed.

Nevertheless, both pitch values (I and K) can also be entered to force the CNC to cut the tapered thread with a pitch different from the one the CNC would have calculated.

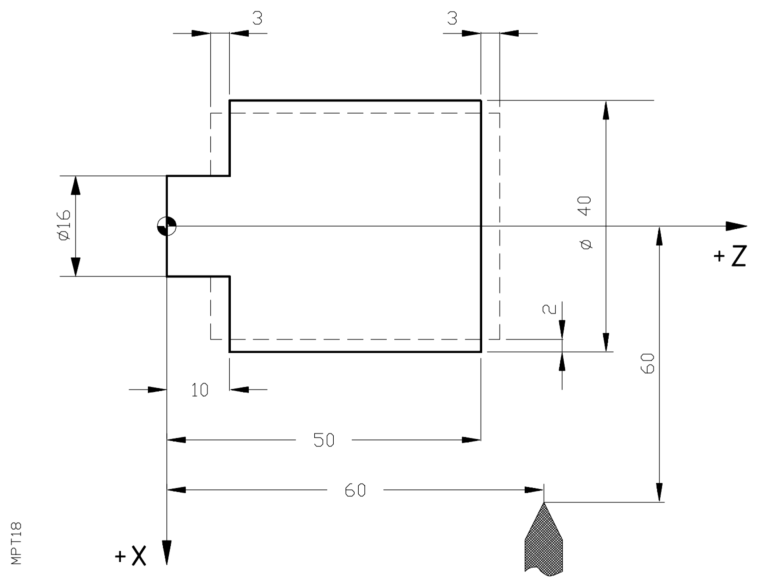
Atention:

The following error will normally produce incorrect pitches at the starting and ending points of the threadcut. The threading length should therefore be longer than required to avoid defective parts.

EXAMPLES:

a) Longitudinal thread

Cutting of a longitudinal thread of 5 mm pitch and 2 mm depth.



The tool is positioned at X60 Z60 (X in radius).

Absolute coordinates

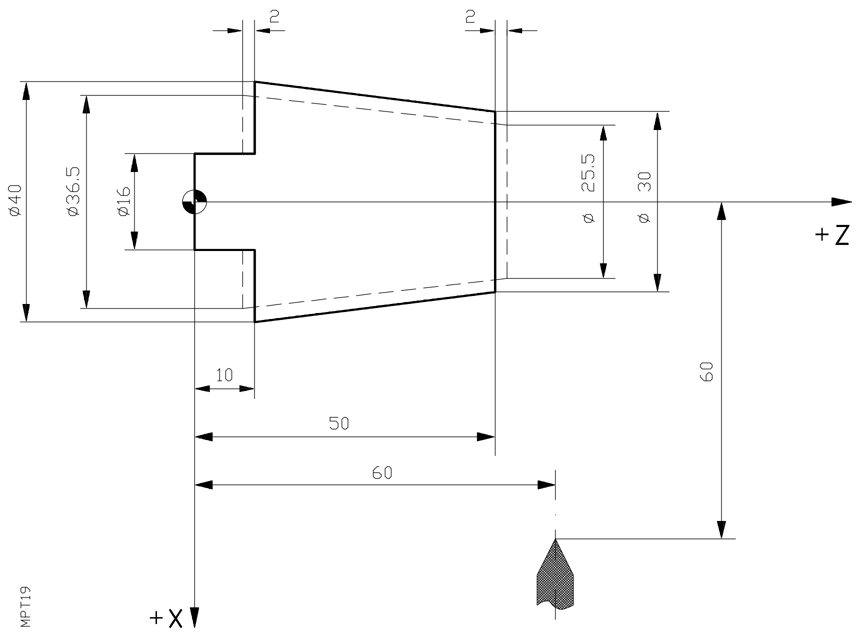
N0   G00   G90 X18 Z53
N5   G33   Z7 K5
N10 G00 X60
N15 Z60

Incremental coordinates

N0   G00 G91 X-42 Z-7
N5   G33 Z-46 K5
N10 G00 X42
N15 Z53

b) Tapered thread

Cutting of a tapered thread of 5 mm pitch along Z axis and 2 mm depth.



Let us assume that the tool is positioned at X60 Z60 (X in radius).

Absolute coordinates

N0   G00 G90 X12,75 Z52
N5   G33 X18,25 Z8 K5
N10 G00 X60
N15 Z60

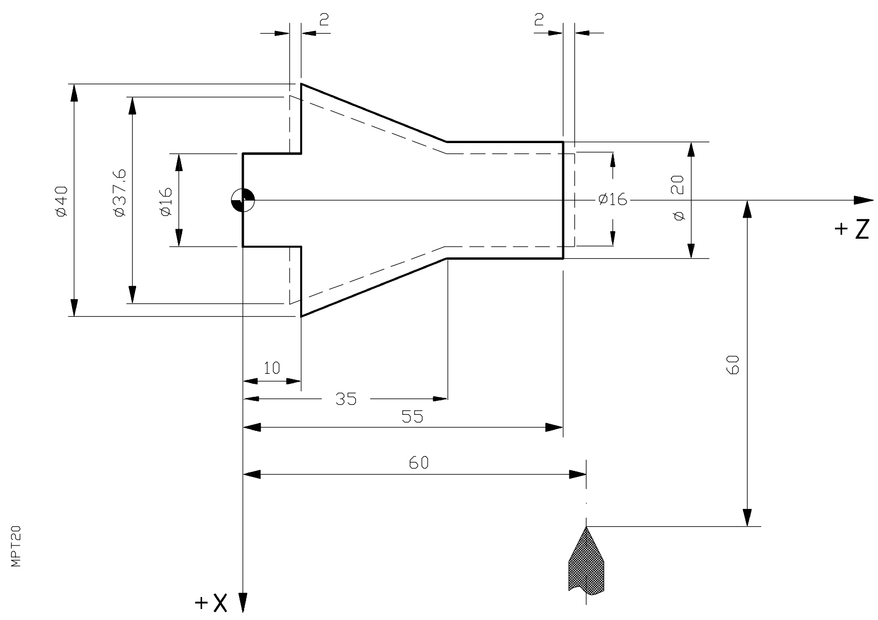
Incremental coordinates

N0   G00   G91 X-47,25 Z-8
N5   G33   X6   Z-44 K5
N10 G00   X41,75
N15 Z52

c) Thread coupling

Using G05, different threads can be coupled in a continuous way on the same part.

A longitudinal and a tapered thread of 5 mm pitch and 2 mm depth must be coupled.



Let us assume the tool is positioned at X60 Z60 (X in radius).

Absolute coordinates

```
N0   G00    G90  X8    Z57
N5   G33    G05  Z35  K5
N10 X18
```

6.11. G36. AUTOMATIC RADIUS BLEND

This function 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.

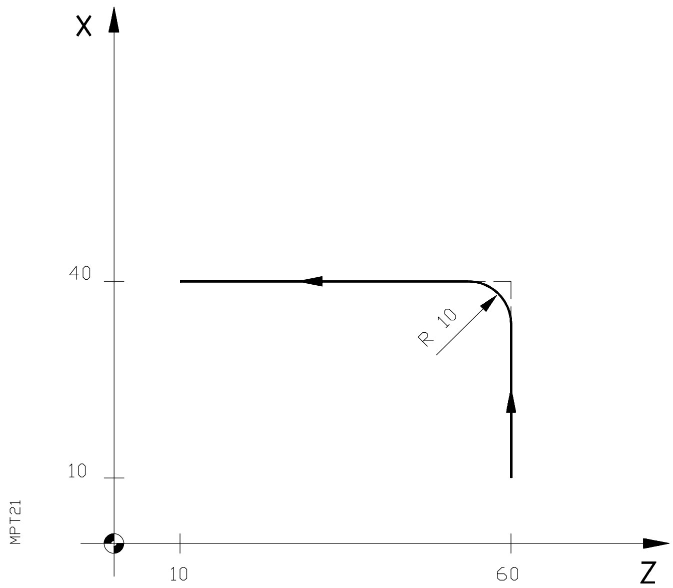
G36 is not modal; i.e. it must be programmed every time a corner rounding is needed.

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 or R3.4).

Examples: X in diameters

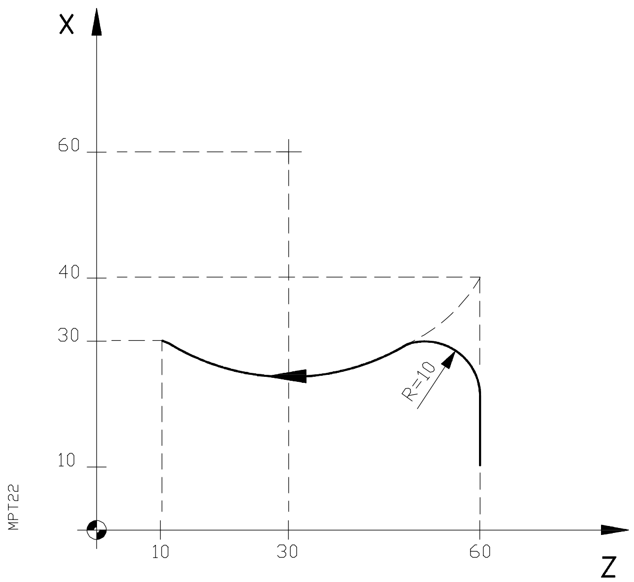
1. Straight-straight rounding



Starting point X20 Z60

N100 G90 G01 G36 R10 X80
N110 Z10

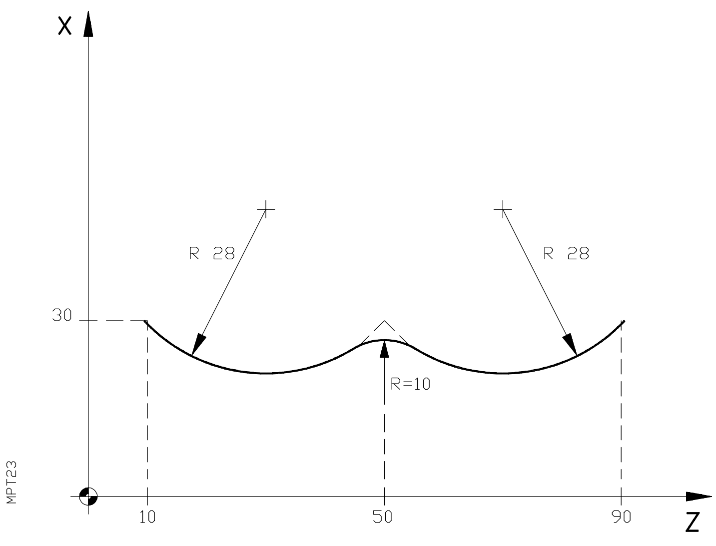
2. Straight-arc rounding



Starting point X20 Z60

N100 G90 G01 G36 R10 X80
N110 G02 X60 Z10 I20   K-30

3. Arc-arc rounding



Starting point X60 Z90

N100 G90 G02 G36 R10 X60 Z50 R28
N110 X60 Z10 R28

6.12. 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.

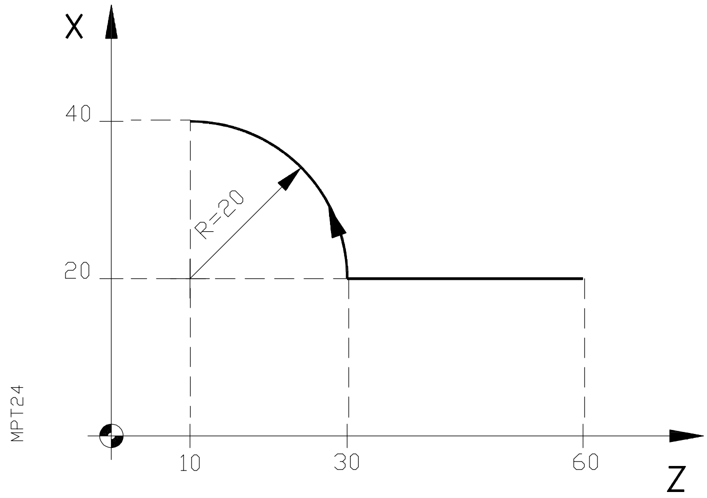
Function **G37** is not modal, i.e., it has to be programmed every time two paths are to be linked tangentially. There paths may be straight-straight or straight-arc. The radius, **R4.3** in mm or **R3.4** in inches, of the entry arc must be programmed following **G37**.

The value of the radius must be positive.

That programming has to be carried out in the block which incorporates the movement whose path is to be altered. The movement must be rectilinear (G00 or G01).

When **G37 R4.3** is programmed in a block in which a circular movement (G02 or G03) is incorporated, the CNC will display the error **41**.

Example: X in radius

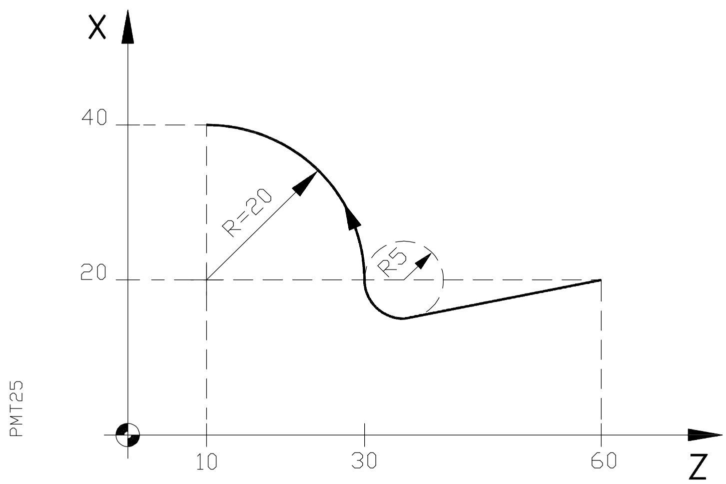


Let us assume that the starting point is X20, Z60, program:

N100 G90 G01 X20 Z30
N110 G03 X40 Z10 R20

In the same example, if we want to program a tangential entry, describing an arc of 5 mm radius, program:

N100 G90 G01 G37 R5 X20 Z30
N110 G03 X40 Z10 R20



6.13. G38. TANGENTIAL EXIT ON COMPLETION OF MACHINING

The preparatory function **G38** can be used to link two paths tangentially without having to calculate the intersection points.

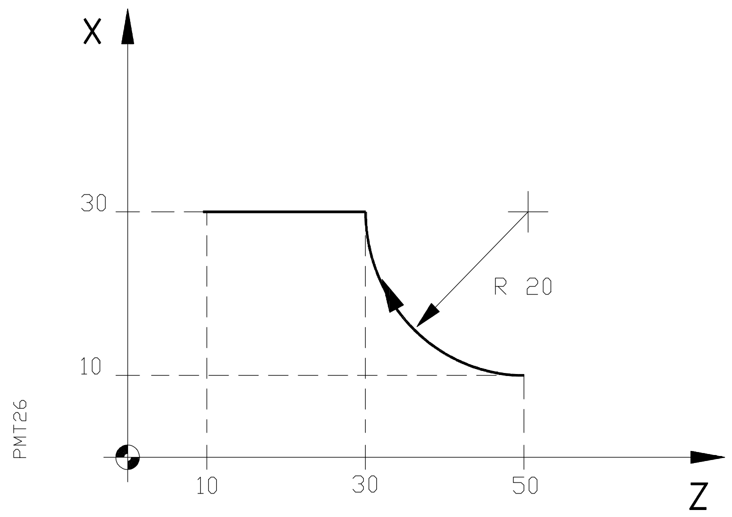
Function **G38** is not modal; i.e., it has to be programmed every time two paths are to be linked tangentially. These paths may be straight-straight or straight-arc.

The radius **R4.3** in mm or **R3.4** in inches, of the exit arc must be programmed following **G38**. The value must be positive.

The path of the subsequent block must be rectilinear (G00 or G01), to enable the programming in a **G38** block.

If the subsequent path is circular (G02 or G03), the CNC will display error **42.**

Example: Programming the X axis in radius

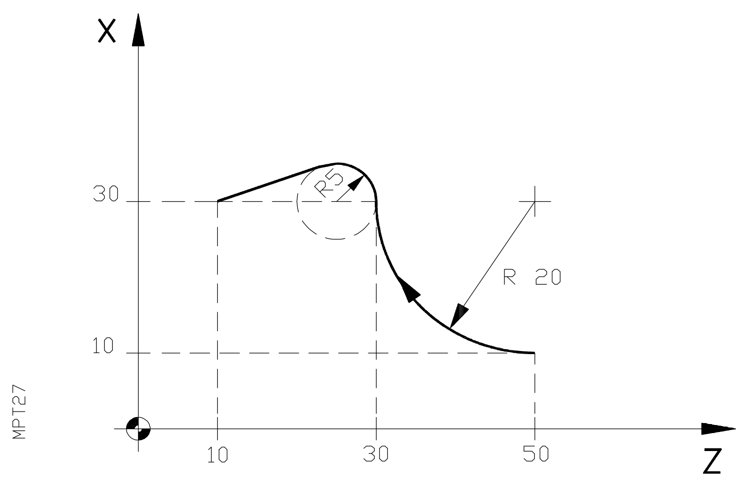


Let us assume the starting point is X10, Z50. Program:

N100 G90 G02 X30 Z30 R20
N110 G01 X30 Z10

In the same example, if we want to program a tangential exit by describing the arc of **5** mm radius, program:

N100 G90 G38 R5 G02 X30 Z30 R20
N110 G01 X30 Z10



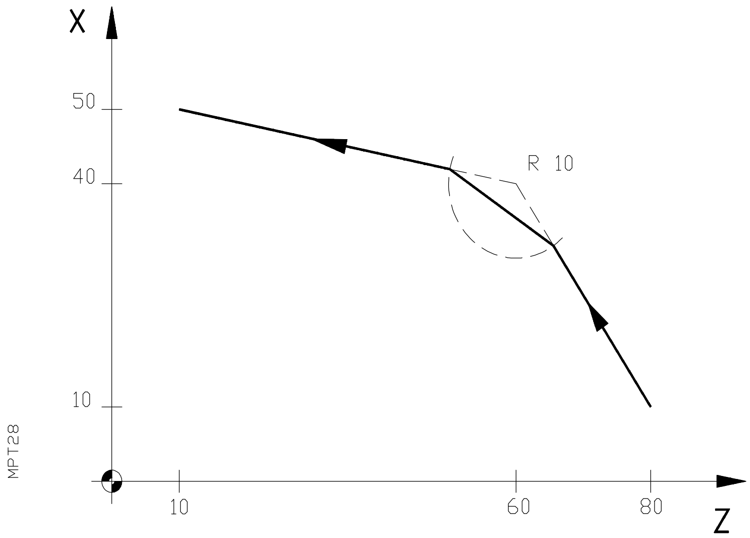
6.14. G39. CHAMFERING

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 need. It must be programmed in the same block as the movement whose end must be chamfered.

Use the code **R4.3** (R3.4) always positive to program the distance between the final point programmed and the point in which the chamfer is to start.

Example: X in diameters



Starting point X20 Z80

N100 G90 G01 G39 R10 X80 Z60
N110 X100 Z10

6.15. TOOL RADIUS COMPENSATION

In normal turning work the path of the tool has to be calculated and defined taking its dimensions into account so as to obtain the required dimensions of the part produced.

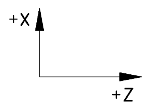
Tool compensation enables the contour of the part to be programmed directly without taking the dimensions of the tool into account. The CNC automatically calculates the path to be followed by the tool, based on the contour of the part and the tool dimensions stored in the tool table.

Every time a tool (T2.2) is selected the CNC automatically applies the tool length compensation (X,Z,I,K) stored in the table, without having to program any **G** code. If **P604(5)** is **1** the tool length compensation is effective when **M06** is executed.

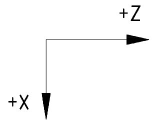
There are three preparatory functions for tool radius compensation:

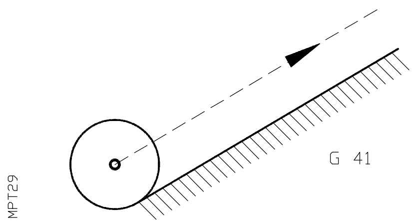
G40 : Cancellation of tool radius compensation
G41 : Left hand tool radius compensation
G42 : Right hand tool radius compensation

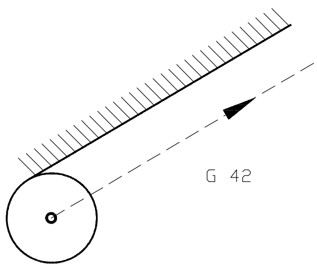
Parameter P600(1) = 0 and axes



Parameter P600(1) = 1 and axes





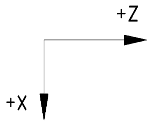


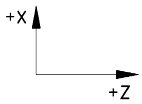
G41:

The tool is on the part's left side as seen following the direction of the movement.

G42:

The tool is on the part's right side as seen following the direction of the movement.

If parameter P600(1) = 0 and axes 

or parameter P600(1) = 1 and axes 

G41 operates as G42 and vice versa.

The CNC has a table of up to **32** tool offsets comprising for each tool length **(X,Z,I,K)** and radius **(R)** values plus location codes **(F)**. The compensation values must be stored in the tool offsets mode (8). The values of **I,K** can also be checked and modified, without stopping the execution of a program (see Operation Manual). The tool table can also be loaded by using G50 in the program.

The max. values are:

X,Z (tool length) +/- 8388.607 mm (+/-330.2599 inches)

I,K (tool length offsets) +/-32.766 mm (+/-1.2900 inches)

R (Radius) 1000.000 mm (39.3700 inches)

The location code of the tool **(F)** is also necessary to perform radius compensation.

Possible codes are **F0-F9** (see figure).

The compensation is made effective by means of **G41** or **G42** and acquires the table value selected by code Txx.xx (Txx.01-Txx.32). If Txx.xx has not been programmed, the CNC assumes the value T00.00 which corresponds to a tool whose dimensions are zero.

Functions **G41** and **G42** are modal (persistent) and are cancelled by **G40,M02,M30** as well as by an **EMERGENCY** or by a general **RESET**.

The CNC applies tool length compensation **(X,Z,I,K)** as soon as a tool (Txx.01) is programmed, unless **P604(5)** is **1**. In this case the compensation is effective after **M06**.

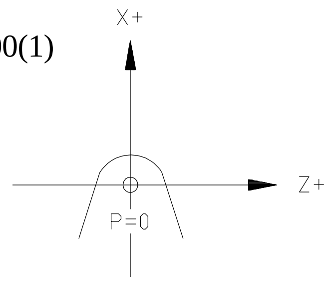
Atention:

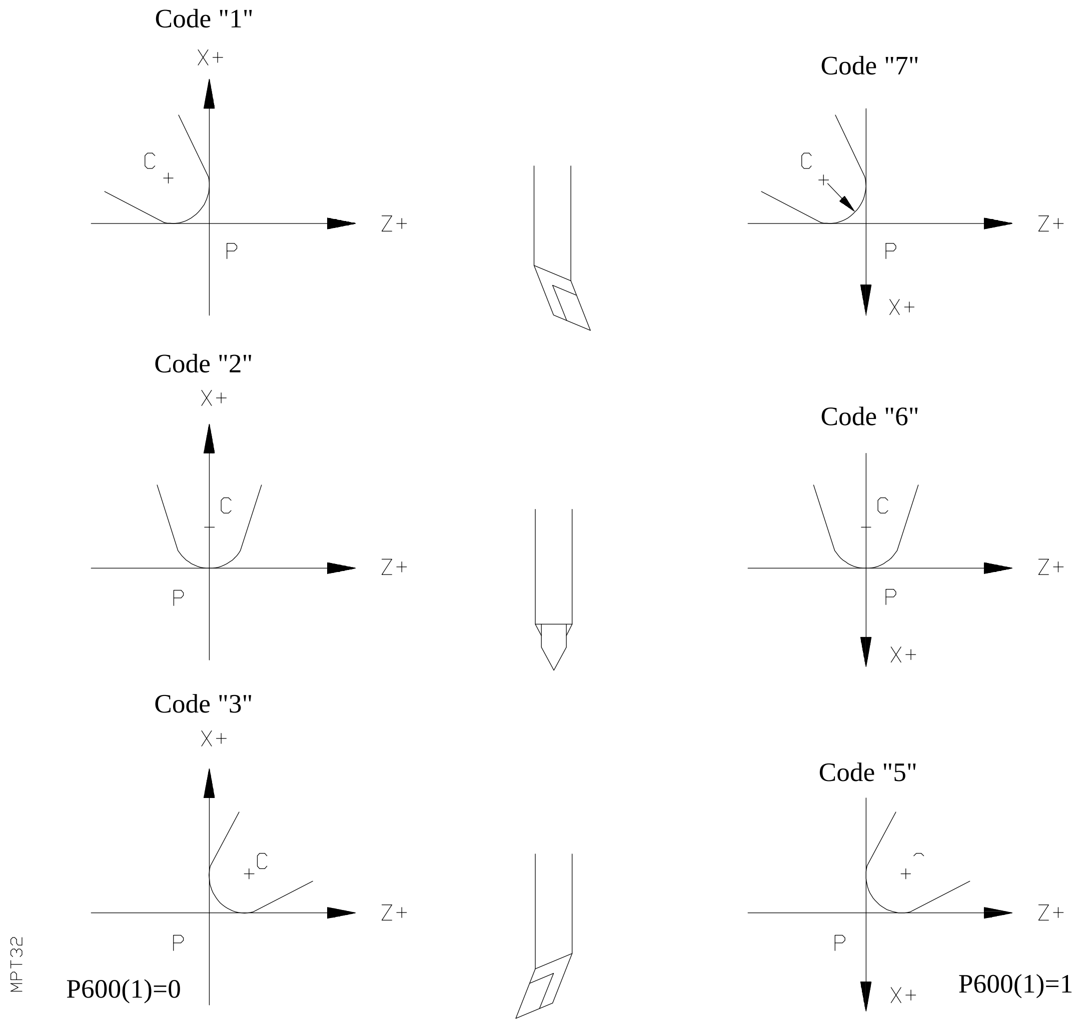
The values of **I**, used to offset tool wear, must be entered in diameters.

LOCATION CODES

Each axis orientation is selected by P600(1) as described on page 48

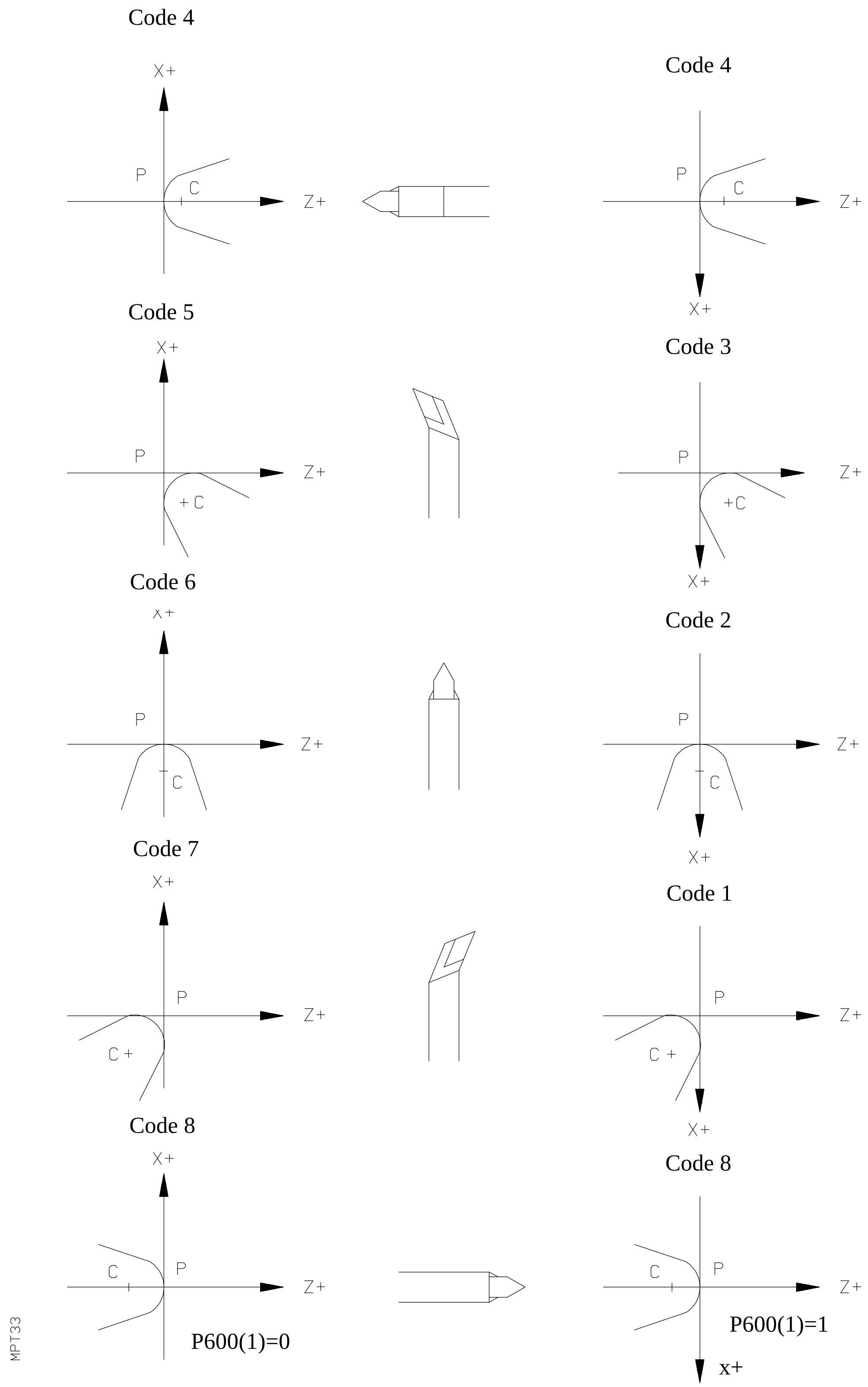
Code "0" and "9"





P : Tool tip
C : Tool nose radius center

Note: Graphics always displays the theoretical tool tip (or the part surface when in dry run theoretical path) also mode (0, 1, 2).



6.15.1. Selection and initiation of tool radius compensation

The code **G41** or **G42** must be used to initiate compensation.

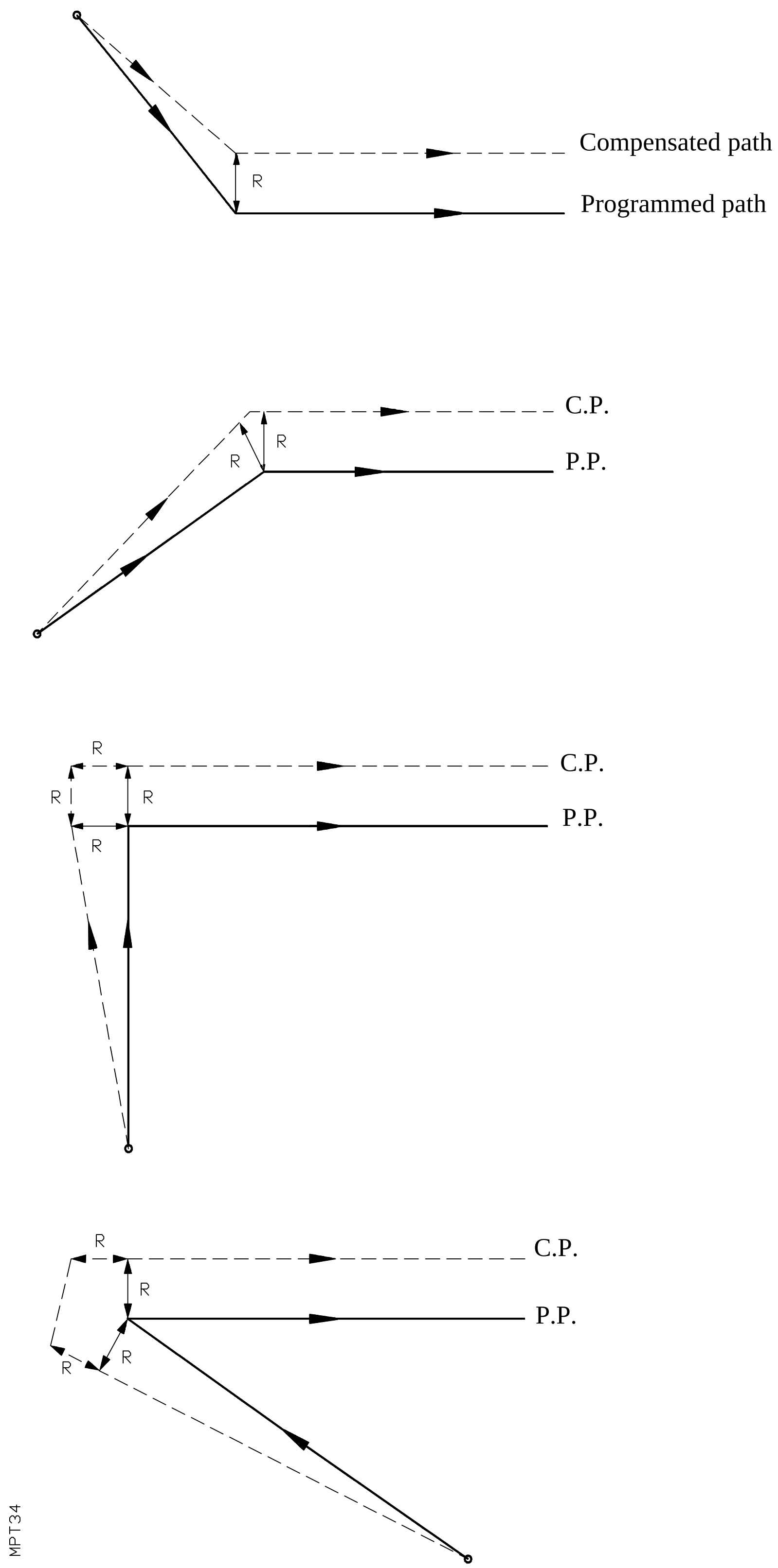
Either the block in which **G41/G42** is programmed or a previous block must include programming of function Txx.xx (Txx.00- Txx.32) to select from the tool table the correction value to be applied. If no tool is selected, the CNC assumes the value T00.00.

Atention:

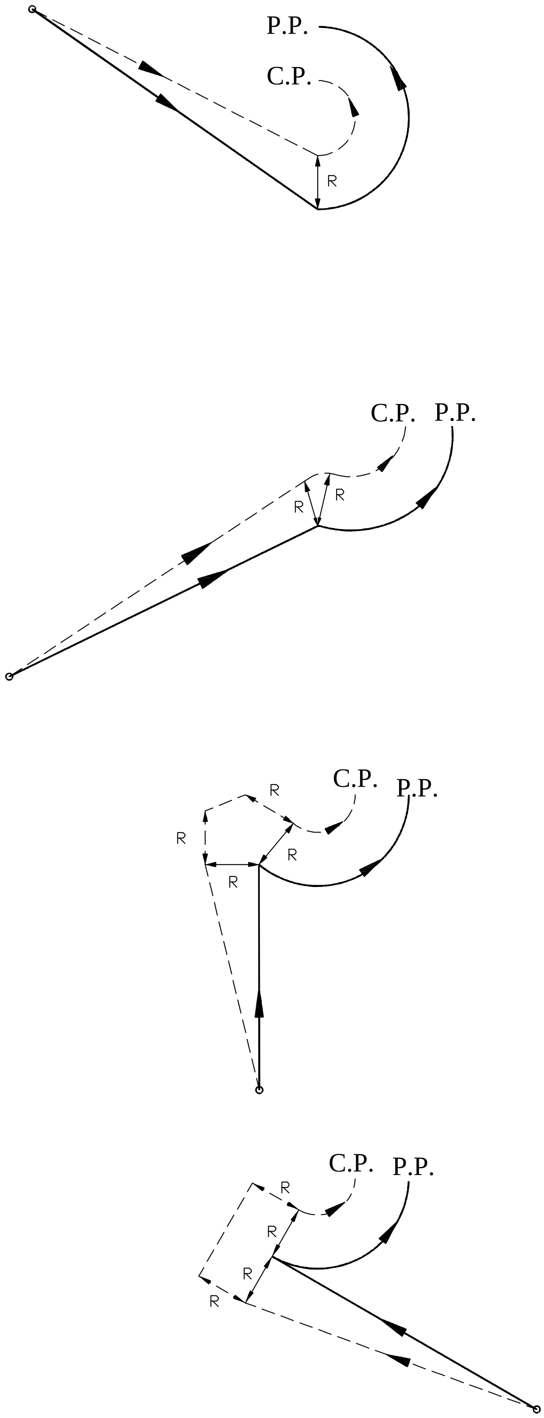
Tool radius compensation selection (**G41/G42**) can only be carried out when **G00** or **G01** (rectilinear movements) is active.

If the first call for compensation is made when **G02** or **G03** are active, the CNC will display error code **41**.

The next page illustrates various cases of initiation of tool radius compensation.



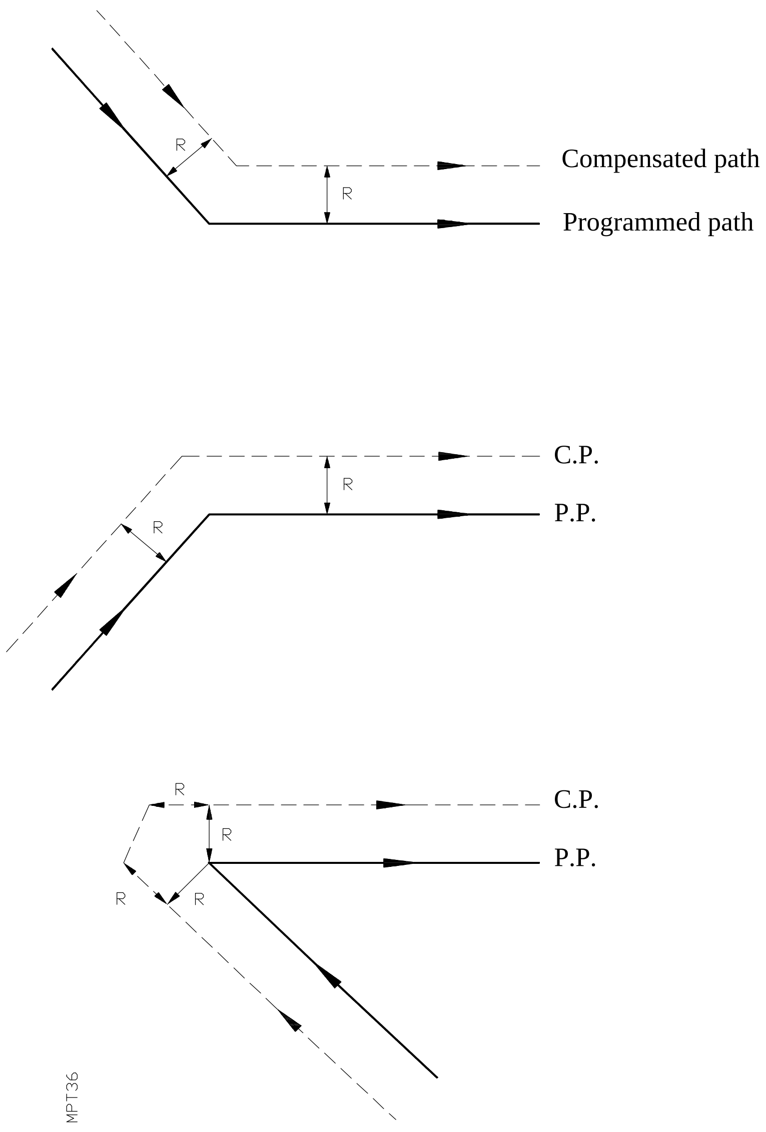
Compensated path
Programmed path
R
C.P.
P.P.
R
R
C.P.
P.P.
R
R
R
R
C.P.
P.P.
R
R
R
R
MPT34

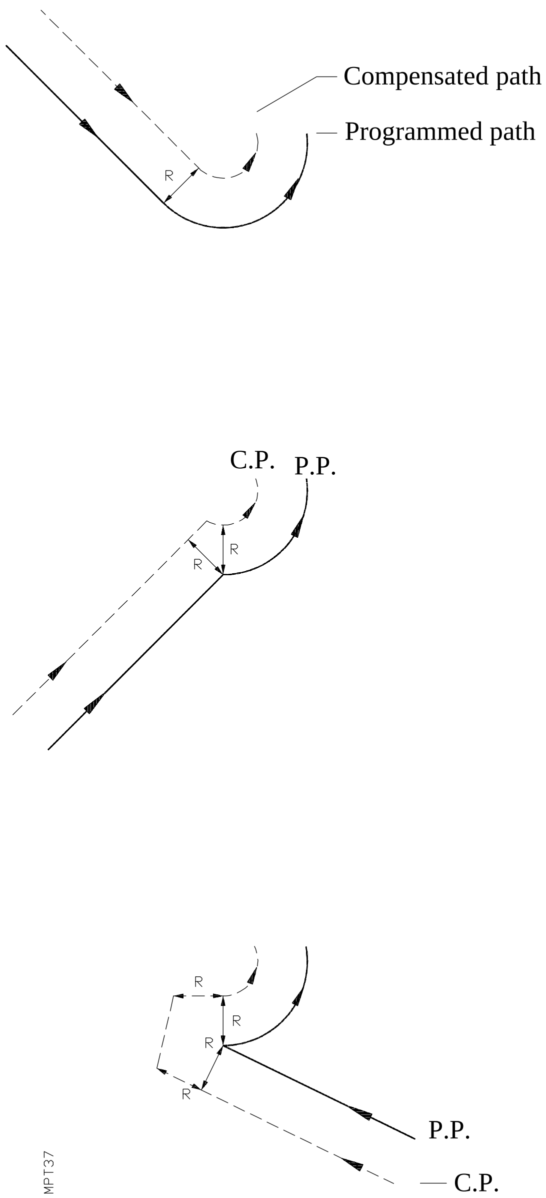


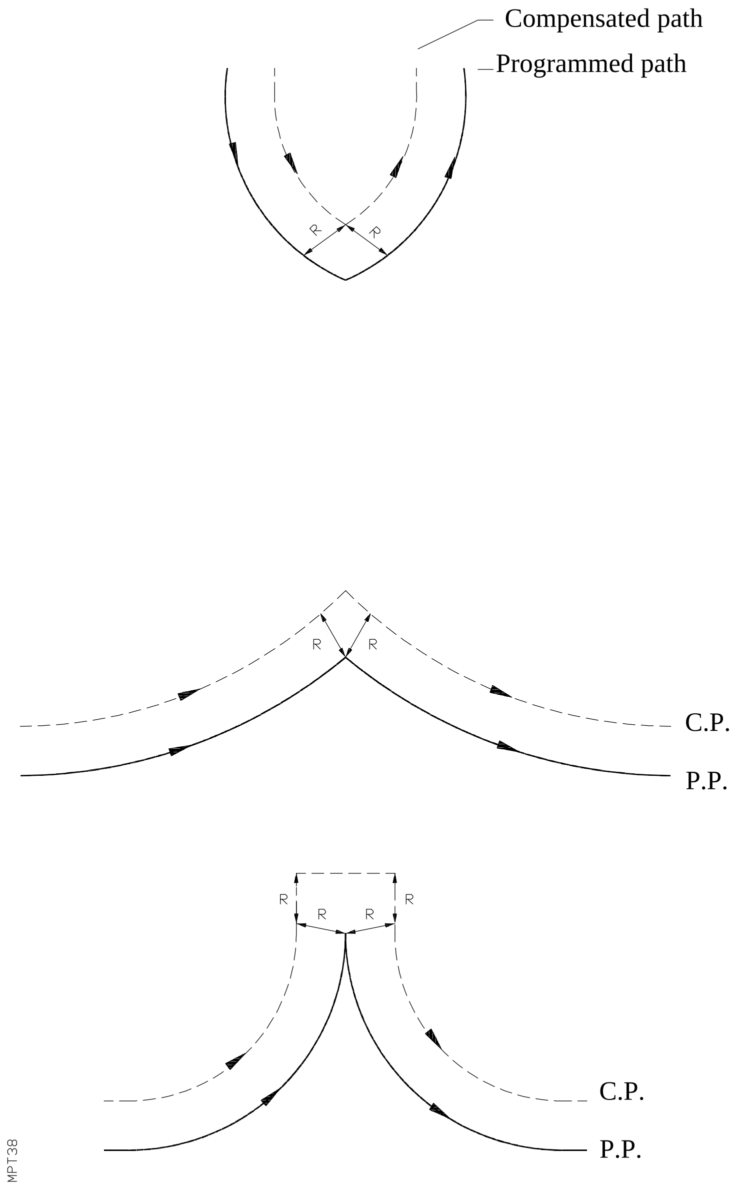
MPT35

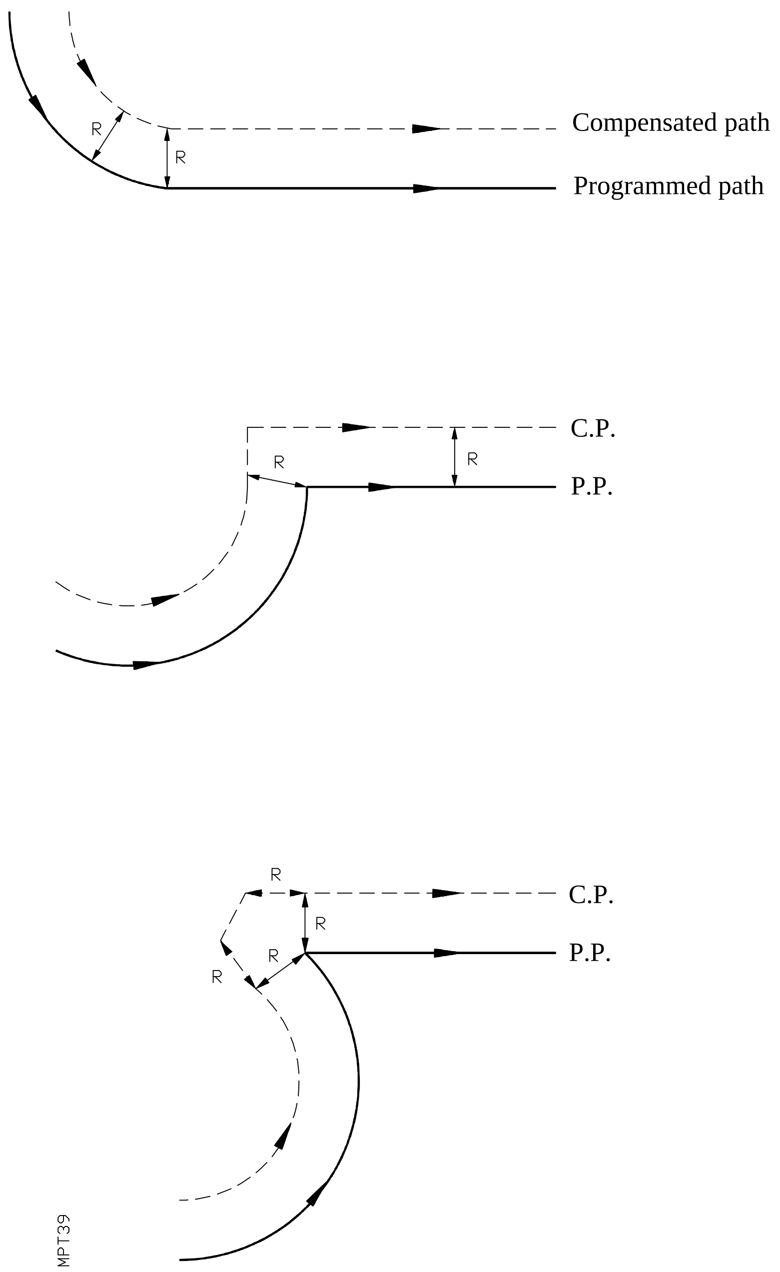
6.15.2. Operating with tool radius compensation

The graphs below illustrate the various paths followed by a tool controlled by a CNC programmed with radius compensation.



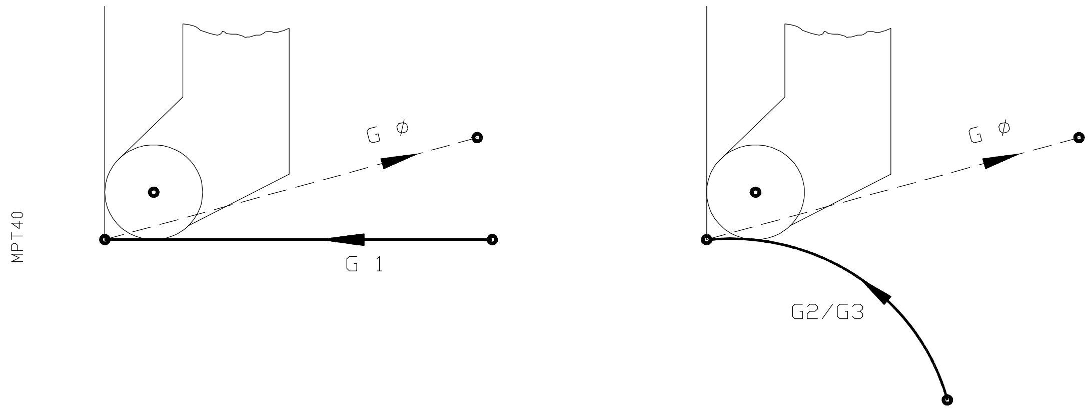

Compensated path
Programmed path
R
C.P.
P.P.
R
R
R
R
R
R
R
P.P.
C.P.
MPT37


Compensated path
Programmed path
R
R
R
R
C.P.
P.P.
R
R
R
R
C.P.
P.P.
MPT38


Compensated path
Programmed path
R
R
C.P.
P.P.
R
R
C.P.
P.P.
R
R
R
R
MPT39

6.15.3 Tool radius compensation freeze with G00

When a change from **G01,G02,G03** to **G00** is detected by the CNC, the tool is positioned tangent to the line perpendicular to the path at the final point of the block previous to the one in which **G00** is programmed.



The same precess is applied when a block with **G40** without movement is programmed.

The following **G00** movements are carried out without tool radius compensation.

When a change from **G00** to **G01,G02,G03** is detected the CNC applies the same process as when the tool radius compensation is initiated.

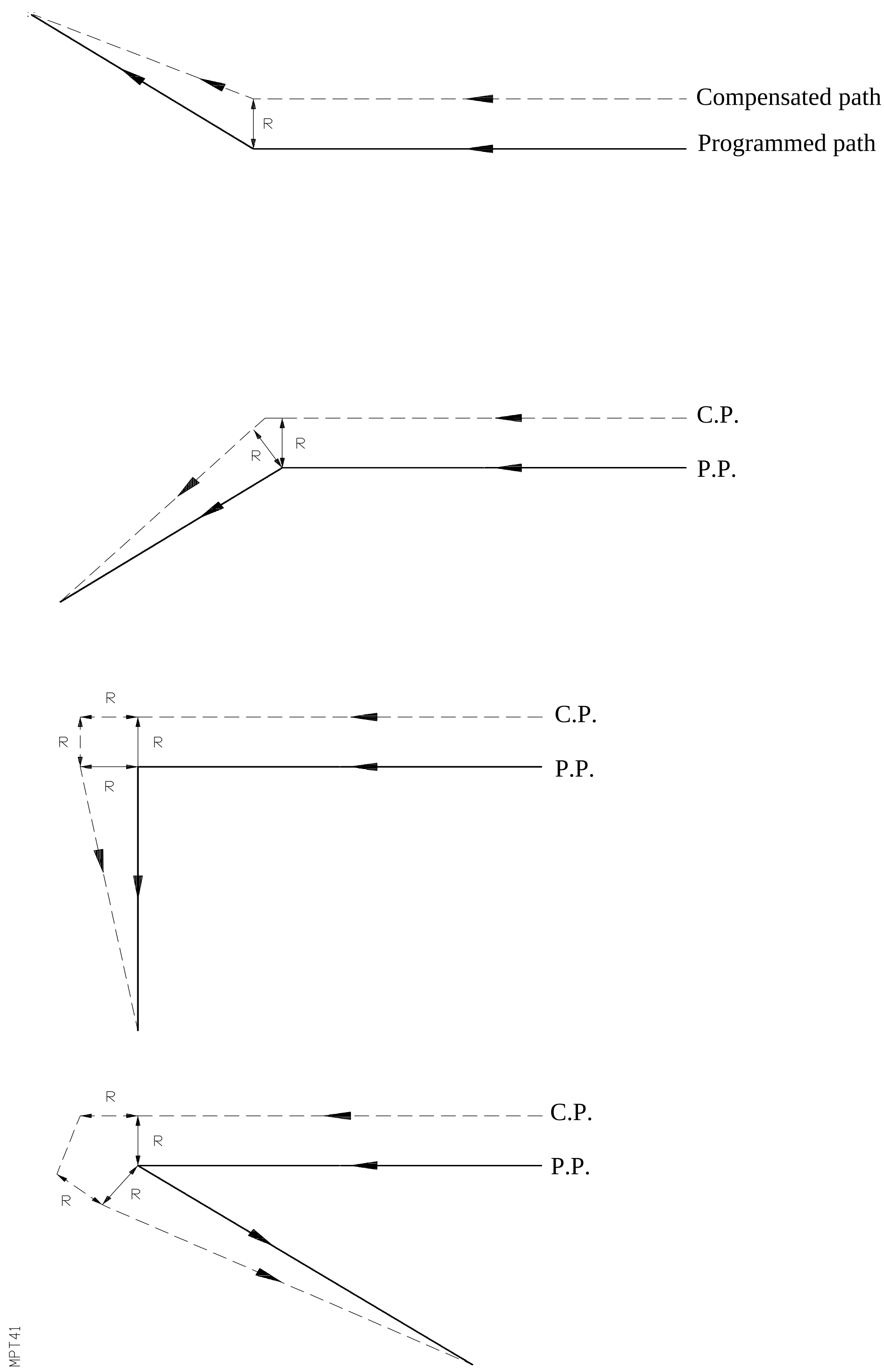
6.15.4. Cancellation of radius compensation

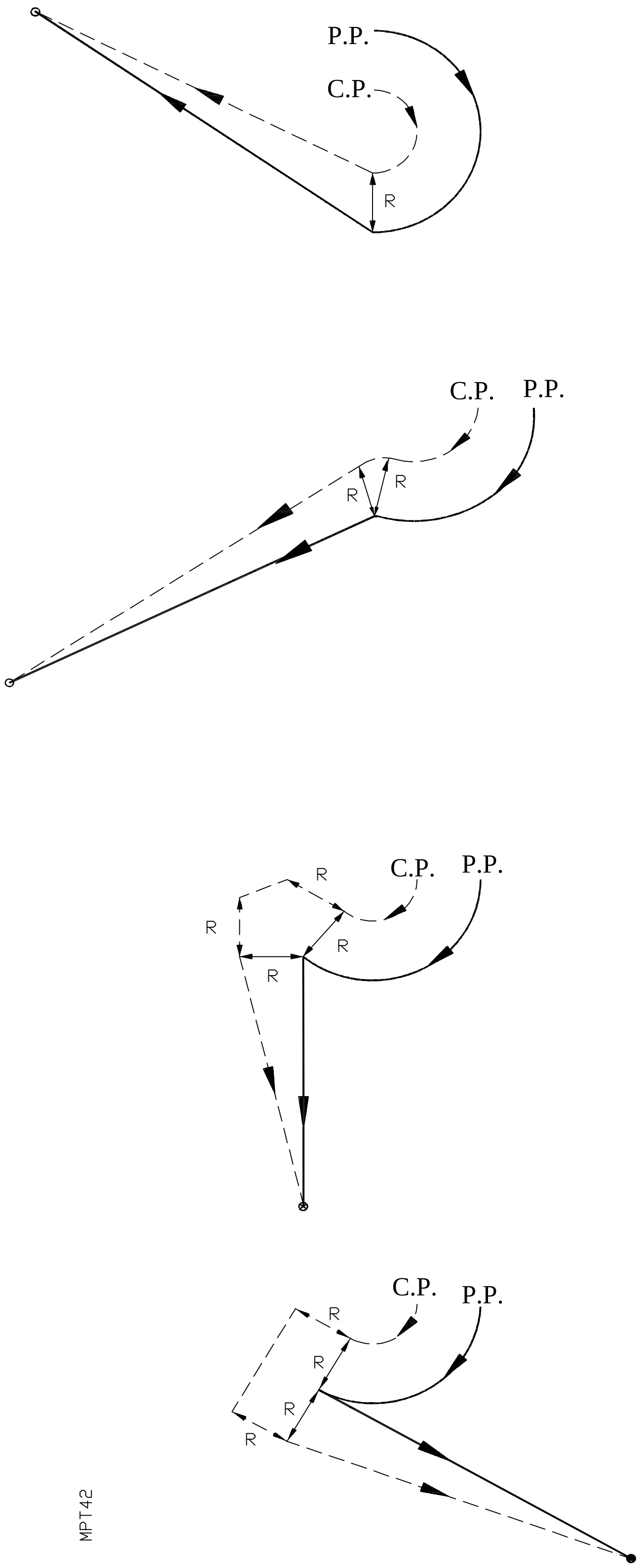
Radius compensation cancellation is achieved by function **G40**.

It should be borne in mind that 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 error **48**.

The following is a table of various cases of cancellation of compensation.


Compensated path
Programmed path
R
C.P.
P.P.
R
R
C.P.
P.P.
R
R
R
R
C.P.
P.P.
R
R
R
R
MPT41


P.P.
C.P.
R
C.P.
P.P.
R
R
C.P.
P.P.
R
R
R
R
C.P.
P.P.
R
R
R
R
MPT42

6.16. 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 machine parameter 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.

6.17. 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.

6.18. G50. LOADING OF THE VALUES IN THE TOOL OFFSET TABLE

The different tool values can be either altered or entered in the table by using **G50**.

There are many method to program the function **G50**:

a) Entering of all the values

By means of the block N4 G50 T2 X+/-4.3 Z+/-4.3 F1 R4.3 I+/-2.3 K+/-2.3 mm X+/-3.4 Z+/-3.4 F1 R2.4 I+/-1.4 K+/-1.4 inches

The values defined by X,Z,F,R,I,K are loaded in the tool offset table direction identified by **T2**.

N4	- Block number
G50	- Tool offsets loading code
T2(T01-T32)	- Tool offset table direction
X+/-4.3 (X+/-3.4)	- Tool length along **X** axis
Z+/-4.3 (Z+/-3.4)	- Tool length along **Z** axis
F1 (F0-F9)	- Location code of the tool
R4.3 (R2.4)	- Tool nose radius
I+/-2.3 (I+/-1.4)	- Tool wear offset along **X** axis (diameters)
K+/-2.3 (K+/-1.4)	- Tool wear offset along **Z** axis

The values of **X,Z,F,R,I,K** replace the values previously existing in the **T2** direction.

b) If only one or some of the values are to be altered, program the mentioned values following G50 T2. The other values won't be altered.

Programming this way, the following aspects must be taken into account:

- When **X** or **Z** both are programmed without programming (I,K), the lengths (X,Z) are replaced in the table by the new values and the relevant wear offset values, **I** or **K** or both are reset.

- When **I+/-2.3** or **I+/-2.3 K+/-2.3** are programmed following **G50 T2**, they are added or subtracted from the previous values recorded.

No more information can be programmed in the block containing **G50**.

6.19. G51. ALTERATION OF THE I AND K VALUES OF THE ENGAGED TOOL

By means of the **G51** function the **I**,**K** values of the tool engaged may be artificially altered but the values recorded in the table are not affected.

The block N4 G51 I+/-2.3 K+/-2.3 (mm)
I+/-1.4 K+/-1.4 (inches)

artificially alters the values of **I**,**K**.

N4 - Block number

G51 - Tool dimensions alteration code

I+/-2.3 (I+/-1.4) - Value to be added to or subtracted from the value of **I** being actually used by the CNC to offset the engaged tool.

K+/-2.3 (K+/-1.4) -Value to be added to or subtracted from the value of **K** being actually used by the CNC to offset the engaged tool.

These values do not modify the table; i.e. next time this particular tool is programmed the CNC will again assume the values recorded in the table disregarding the modification entered via **G51**.

No more information can be programmed in the block containing **G51**.

6.20. G52. COMMUNICATION WITH THE FAGOR LOCAL AREA NETWORK

The communication between the CNC and the rest of the LAN NODES is carried out to 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:

Due to any error at the FAGOR LAN occurring during the execution, the CNC will display the corresponding error code.

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.

6.21. 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 8**, then key **8** and finally key **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 4th, 3rd,X,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).

4th+/-4.3 : Zero offset value referred to the machine
4th+/-3.4 reference zero on the 4th axis.

3rd+/-4.3 : Zero offset value referred to the machine
3rd+/-3.4 reference zero on the 3rd axis.

X+/-4.3 : Zero offset value referred to the machine
X+/-3.4 reference zero on the X axis.

Z+/-4.3 : Zero offset value referred to the machine
Z+/-3.4 reference zero on the Z axis.

G53 only : Called by part program
Made active by program

G53 X, Z, etc.: Modified and made active by the part program.

. Incremental loading of the values

Block N4 G5? (H+/-4.3) L+/-4.3 H+/-4.3 I+/-4.3 J+/-4.3 K+/-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 L+/-3.4	:	Amount added or subtracted to the V value previously stored in the table.
H+/-4.3 H+/-3.4	:	Amount added or subtracted to the W value previously stored in the table.
I+/-4.3 I+/-3.4	:	Amount added or subtracted to the X value previously stored in the table.
K+/-4.3 K+/-3.4	:	Amount added or subtracted to the Z 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) P616(4)=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):

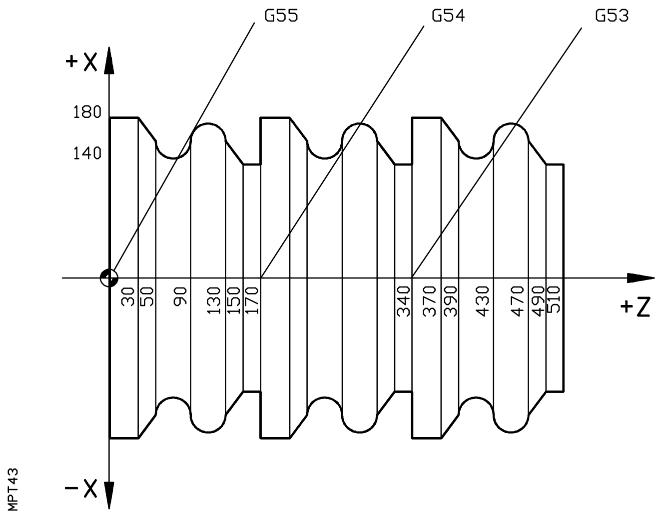
(Memory address in which the zero offset values are stored.)

Case 2) P616(4) = 1

When a function of the type G54 .... G58 is executed, the zero offset applied to each axis will be the value indicated in the table (G54... G58) plus the value indicated in position G59 of the table.

It does not affect G53.

Example:



The tool is located in X200 Z530. X axis in radius and the machine-reference point is X0 Z0.

In the G53/G59 table we will enter:

G53 X0 Z340
G54 X0 Z170
G55 X0 Z0

The programming of the theoretical path will be:

```
N10    G90   G01  F250
N20    G53
N30    X140 Z170
N40    Z150
N50    X160 Z130
N60    G03   X160 Z90 I0 K-20
N70    G08   X160 Z50
N80    G01   X180 Z30
N90    Z0
N100   X140
N110   G54
N120   G25   N30.100.1
N130   G55
N140   G25   N30.90.1
N150   G00   X200 Z530
N160   M30
```

6.21.1. G59 as additive zero offset

If P616(4) =1 When a **G64-G59** function is executed the zero offset applied to each axis will be the value indicated in the table (G54....G59) plus the value indicated in position **G59** on the table. It does not affect **G53**.

If P616(4)=0 In this case, the zero offset which is applied to each axis will be the value indicated on the table.

6.22. 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 Y100 F1
N10  G01 X10 Z5 F1000
N20  G01 X20
```

When executing block "N0", the Y axis starts moving at a feedrate of F1. Then, block "N10" starts executing the XZ interpolation at F1000 while the Y 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 (Y 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".

6.23. 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**.

6.24. G72. SCALING

Code **G72** allows the machining of parts of similar shape but different size using the same program. **G72** must be programmed alone in a block.

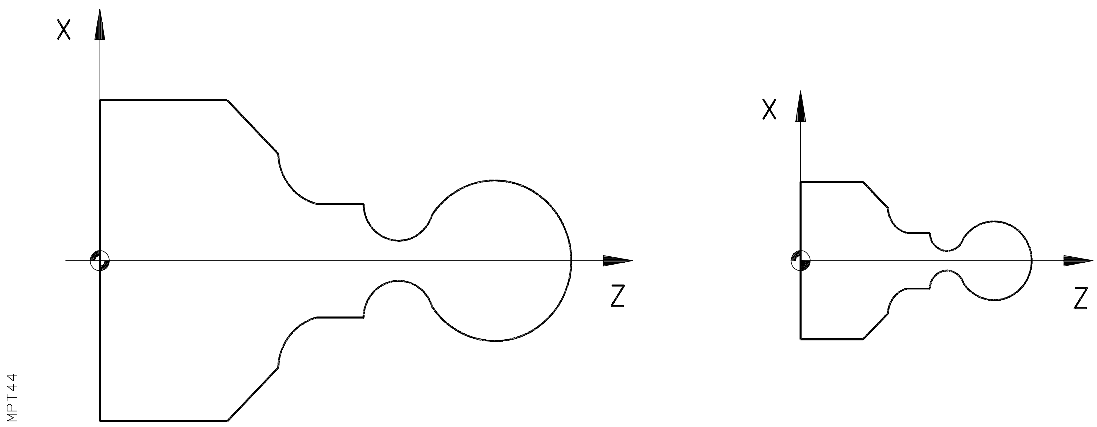
Format: N4 G72 K2.4

N4 : Block number
G72 : Scaling code
K2.4: Value of scaling factor

Min. value K0.0001 (X0.0001)
Max. value K99.9999 (X99.9999)

All coordinate values programmed after **G72** will be multiplied by **K** until the scaling is cancelled.

To cancel the scaling factor one only has to define another **K1** scaling factor or after **M02,M30, EMERGENCY** or **RESET**.



6.25. G74. MACHINE -REFERENCE SEARCH

When **G74** is programmed in a block, the CNC moves the axes to the machine-reference point.

There are two possible cases:

a) Two axes standard referencing (X Z). Only **G74** programmed in the block. The CNC moves first the **X** axis and then the **Z** axis.

b) One or two axes referencing (Z X). 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.

When the axis moved reaches the machine-reference point, the CRT displays the distance between the mentioned point and the last part's zero programmed, minus the tool dimension along the relevant axis (X or Z).

6.26. PROBES

6.26.1. 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.

6.26.2. Characteristics

Probes are of modular construction for better adaptation to the needs of the user. This system consists of a feeler, probe, transmission system and interface.

The feeler 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 feeler. They are of solid and compact construction in order to protect the feeler. 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.

6.26.3. 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 probe cycles 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 **TS** 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.

6.26.4. G75. Probing

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

Format: N4 G75 X+/-4.3 Z+/-4.3 in mm
N4 G75 X+/-3.4 Z+/-3.4 in inches

The axes will move until the probe signal is received. The CNC will then consider the block to be completed and the real position of the axes will be stored as theoretical position. Neither the feedrate will be changed by turning the **FEEDRATE** knob (frozen at 100%) nor the movement of the axes will be displayed until the probe signal is received.

If the axes arrive in position before the probe pins the part the CNC will give error code **65** if so decided by machine parameter. After executing this block, the values of the different axes can be allocated to parameters. The combination of this feature with mathematical operations with parameters allows the creation of special subroutines to measure parts or tools.

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

The CNC allows the tool lengths (**X,Z**) to be measured using a probe.

Refer to the **OPERATING MANUAL**.

The **TS** offers several probing canned cycles which are described next.

6.26.5. G75 N2. Probing canned cycles

The **TS** model offers several probing canned cycles to measure tool and part dimensions. The programming format is as follows:

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

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

The CNC's probing canned cycles are:

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

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

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

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** to **P99**.

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

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

. Parameter **P3** must be greater than zero.

. Parameter **P5** must be equal or greater than zero.

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

BASIC OPERATION

The movements of the axes during a probing cycle are:

Approach

It is executed in rapid mode **G00** from the starting point of the cycle to a safety distance **P3** away from the theoretical value.

Probing

It is executed at a feedrate determined by **P4** until the CNC receives the probe signal. If before moving a maximum distance of **2P3** the CNC has not received the probe signal, error **65** will be displayed and all axes will be stopped.

The CNC will not display the movement of the axis until it receives the probe signal and the **FEEDRATE** knob will have no effect on the feedrate which will be fixed at 100%

Withdrawal

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.

To access the parameter table while on **AUTOMATIC**, **SINGLE BLOCK**, **TEACH-IN** or **DRY RUN**,

key in: [**PARAMETERS**]

and press the arrow keys until the desired parameter is displayed.

The exit conditions of all probing cycles are: **G00**, **G07**,**G40**,**G90**

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 on the turret (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 **X**,**Z**,**F**,**R**, values entered in the appropriate tool table position.

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 **P116**, which **M** function must the CNC send to activate the probe.

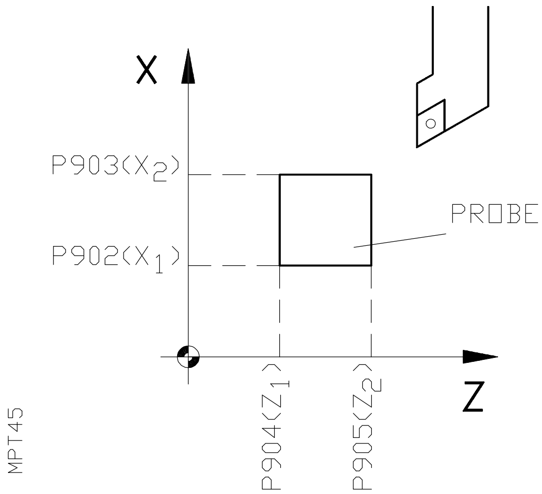
This **M** function will be activated by the CNC at the beginning of the probing cycle and must be cancelled by programming another **M** function.

N0. TOOL CALIBRATION CYCLE

To execute this cycle, 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:

P902 Minimum (X1) value according to the X axis (in radii)
P903 Maximum (X2) value according to the X axis (in radii)
P904 Minimum (Z1) value according to the Z axis
P905 Maximum (Z2) value according to the Z axis



The tool must be previously calibrated with its approximate values already entered in the tool table.

Once the tool has been selected, it can be calibrated by executing this cycle.

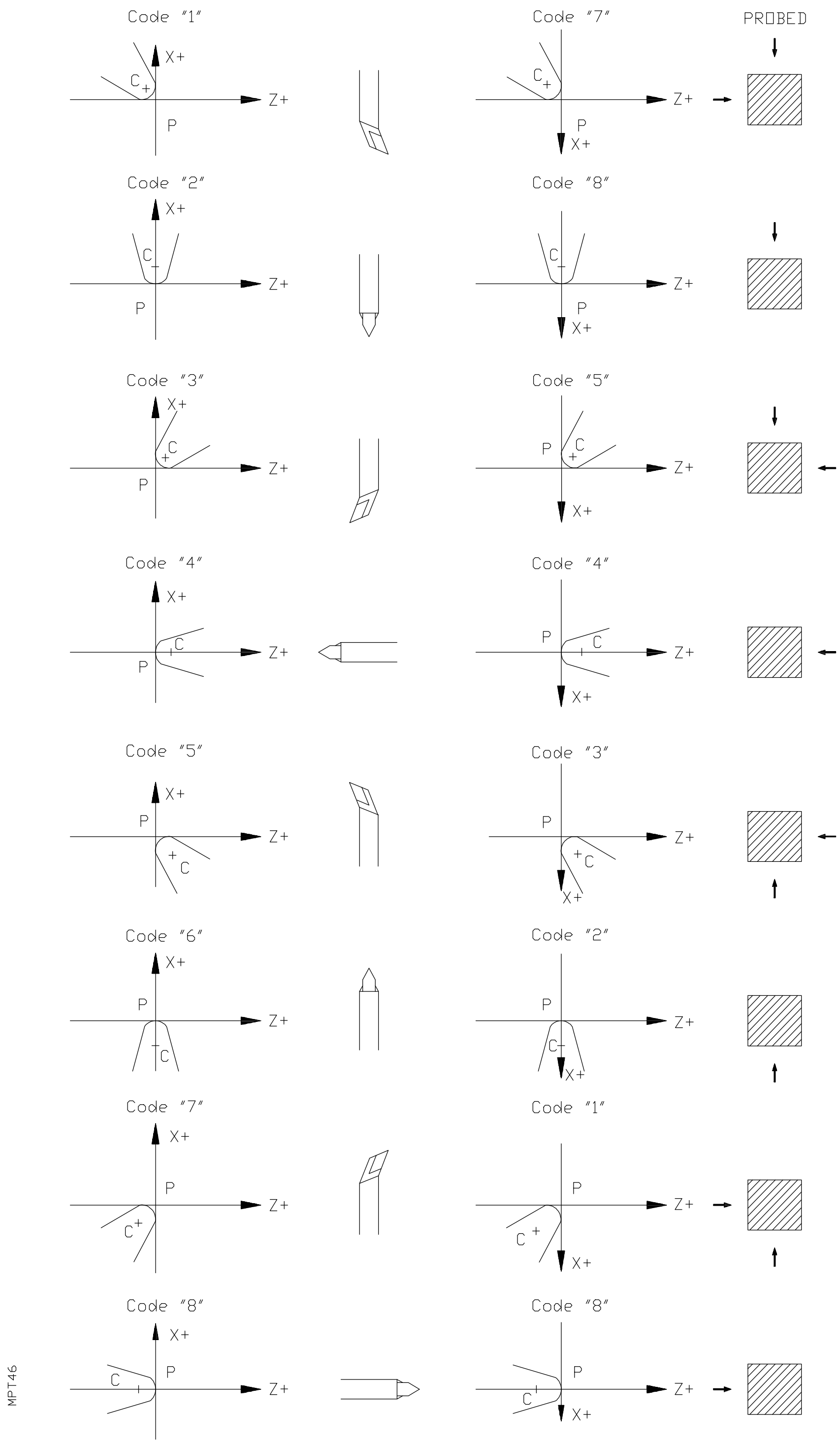
Cycle programming format:

G75 N0 P3=K— P4=K—

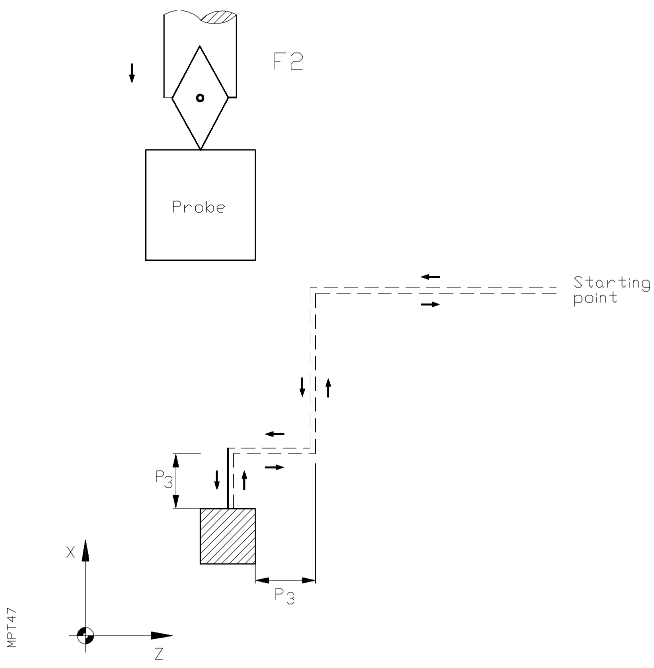
G75 N0 = Tool calibration cycle code.
P3 = Safety distance (in radius).
P4 = Probing feedrate.

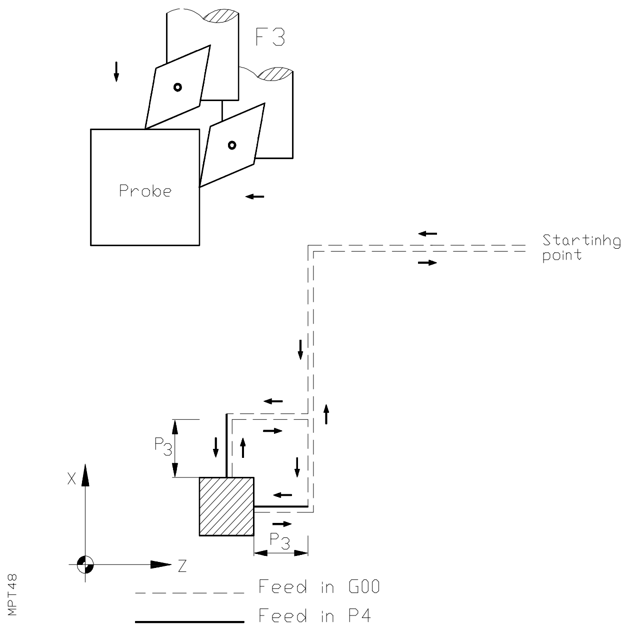
The CNC will execute one or two probings depending on the tool's location code **F**. (see fig.)

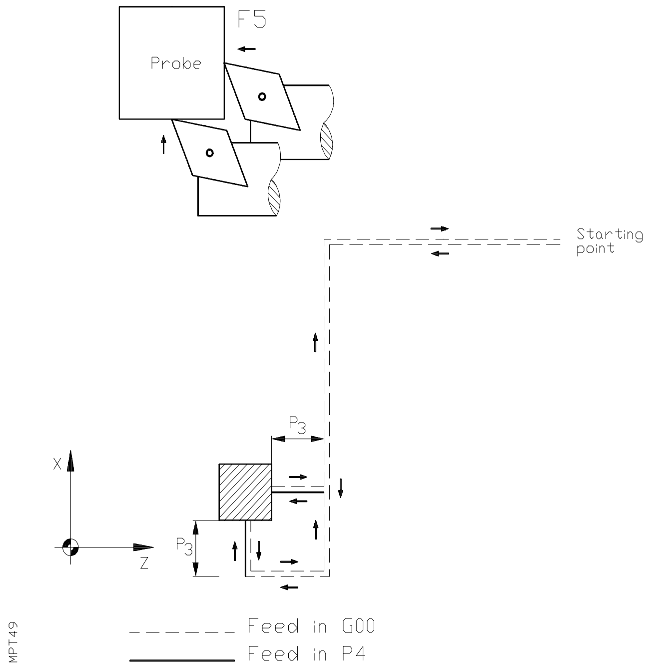
Tools with **F0** or **F9** location codes cannot be calibrated with this cycle, only manually.


Code "1"
Code "7"
PROBED
Code "2"
Code "8"
Code "3"
Code "5"
Code "4"
Code "4"
Code "5"
Code "3"
Code "6"
Code "2"
Code "7"
Code "1"
Code "8"
Code "8"
X+
Z+
C
P
MPT46

The tool movements depending on its location code are described below.







The cycle ends by positioning the tool in the starting point having updated the tool dimensions in the tool table (P6).

The correction values I and **K** are set to zero.

Also, Parameters **P93** and **P95** will indicate:

P93 = Real length minus theoretical length of the tool on the **X** axis (in radius).

P95 = Real length minus theoretical length of the tool on the **Z** axis.

The **P93** value will always be indicated in radii.

N1. PROBE CALIBRATION CYCLE

This cycle is used to calibrate the sides of the probe which is placed in a fixed position on the machine and used to calibrate the different tools.

The approximate values of the sides of the probe are given to the CNC by entering them in the machine parameters **P902**, **P903**, **P904**, **P905**.

A tool will be used whose exact values are entered in the tool table. It must be selected prior to the execution of this cycle.

cycle programming format:

G75 N1 P3=K— P4=K—

G75 N1	= Probe calibration cycle code.
P3	= Safety distance (in radius).
P4	= Probing feedrate.

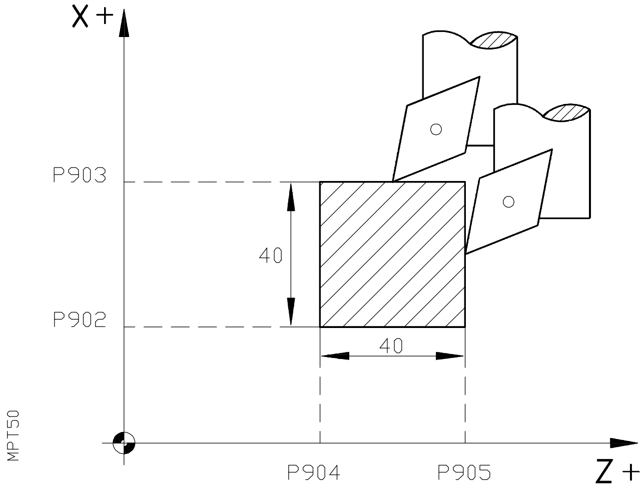
The different movements of this cycle are identical to those described for the tool calibration cycle **N0**.

Once the cycle is ended, the CNC's parameter table will show:

P90 = **X** value of the probe's measurement (in radius).
P92 = **Z** value of the probe's measurement.

Knowing these values and the probe's dimensions, the operator must calculate the values of the other sides of the probe and update, with those values, the machine-parameters **P902**, **P903**, **P904** and **P905**.

Let us suppose that the tool used for this cycle has known dimensions and a location code of **F3** and the probe is a square of **40** mm on each side.

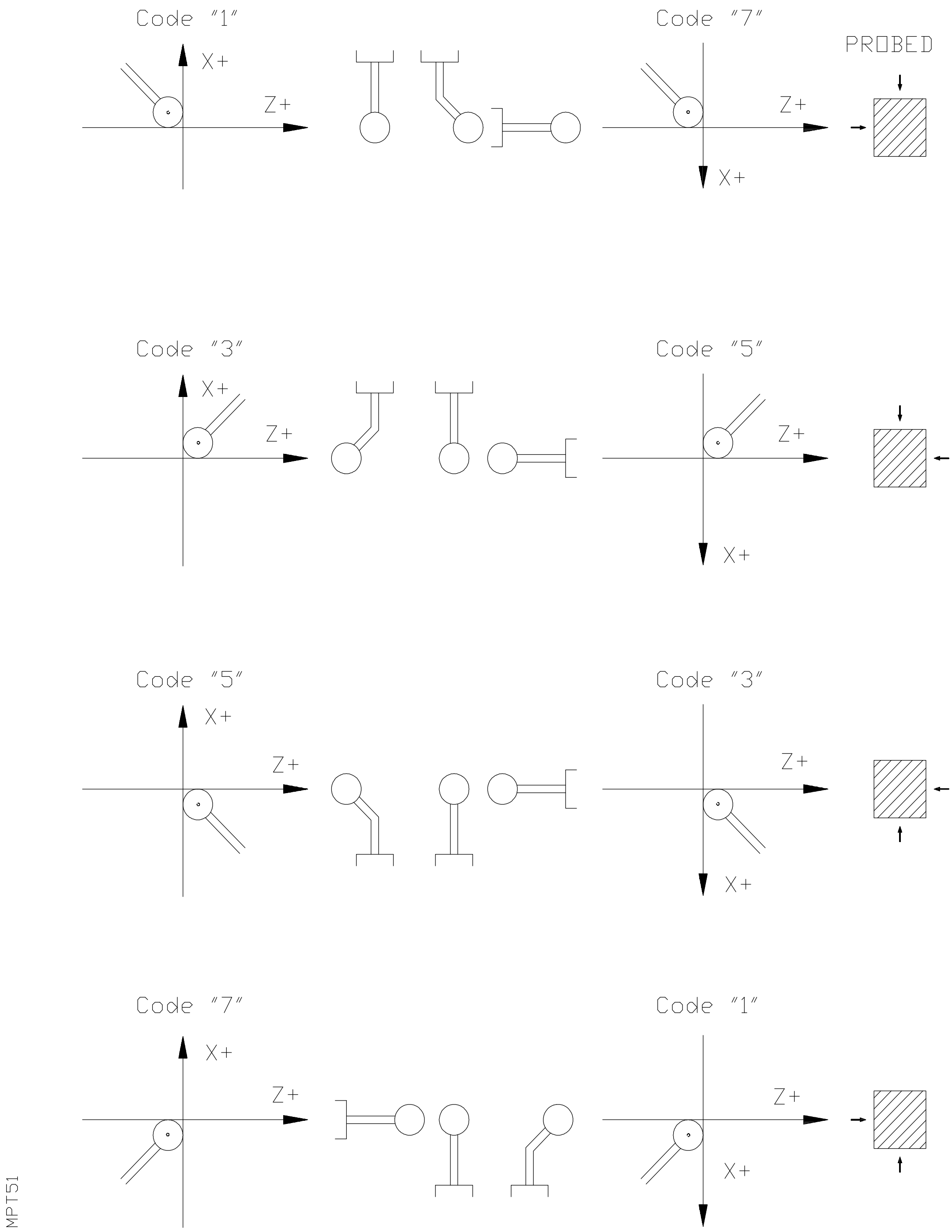


Machine-parameter P902 = P90 - 40
Machine-parameter P903 = P90
Machine-parameter P904 = P92 - 40
Machine-parameter P905 = P92

To run the probing cycles **N2**, **N3**, **N4**, and **N5** described next, a probe will be used placed on the tool turret. The probe must be previously calibrated, by means of **N0** probing cycle, for example, and its dimensions entered in the pertinent tool table.

X - Length in the X axis.
Z - Length in the Z axis.
F - Location code.
R - Radius of the probe’s tip (ball).

The location code to be entered in the tool table will depend on which sides were used to calibrate the probe.


Code "1"
X+
Z+
Code "7"
PROBED
Z+
X+
Code "3"
X+
Z+
Code "5"
Z+
X+
Code "5"
X+
Z+
Code "3"
Z+
X+
Code "7"
X+
Z+
Code "1"
Z+
X+
MPT51

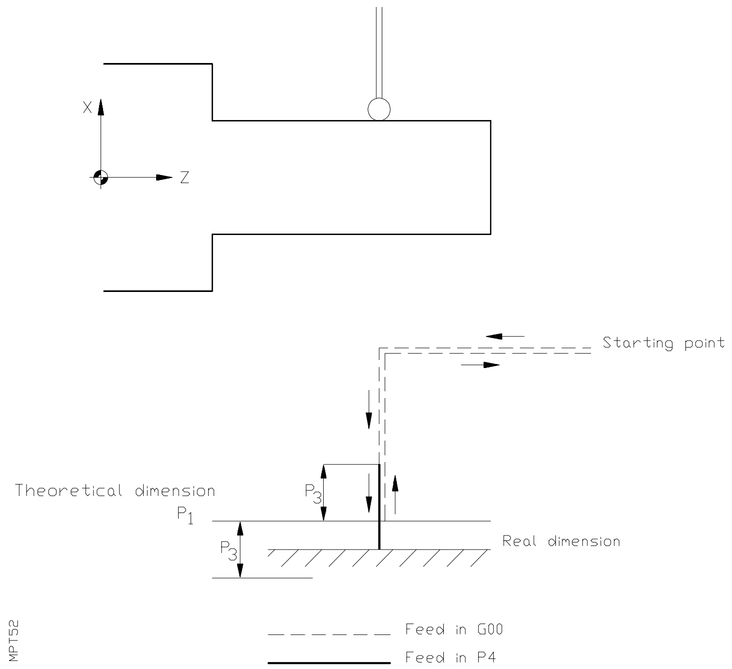
N2. PART MEASUREMENT CYCLE FOR THE X AXIS

Cycle programming format:

G75 N2 P1= K— P2= K— P3= K— P4= K—

G75 N2 = X axis measurement cycle code.
P1 = Theoretical X value of the point to be probed.
P2 = Theoretical Z value of the point to be probed.
P3 = Safety distance (in radius).
P4 = Probing feedrate.

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



Once the cycle is ended, The CNC's parameter table will show:

P90 = Real value measured on the **X** axis.

P93 = Measurement error.

The values of **P90** will be in radius or diameters depending on the setting of the machine-parameter **P11**.

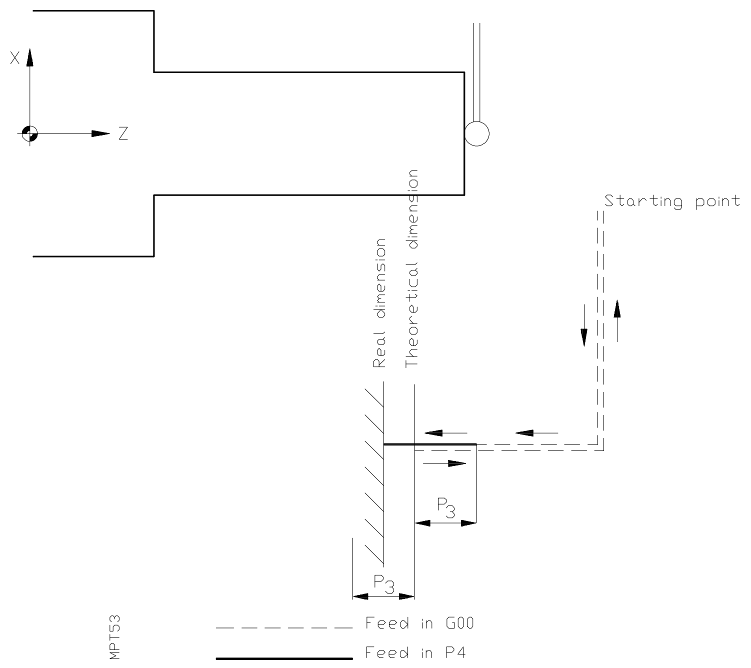
The values of **P93** will always be in diameters.

N3. PART MEASUREMENT CYCLE FOR THE Z AXIS.

Cycle programming format:

G75 N3 P1= K— P2=K— P3=K-- P4=K—

G75 N3 = Z axis measurement cycle code
P1 = Theoretical **X** value of the point to be probed
P2 = Theoretical **Z** value of the point to be probed
P3 = Safety distance
P4 = Probing feedrate



Once the cycle is ended, The CNC’s parameter table will show:

P92 = Real value measured on the **Z** axis.

P95 = Measurement error.

N4. PART MEASUREMENT AND TOOL CALIBRATION CYCLE FOR THE X AXIS

Cycle programming format:

G75 N4 P1= K— P2= K— P3=K— P4= K— P5= K— P6=K—

G75 N4 = Part measurement and tool calibration cycle code for the X axis.
P1 = Theoretical **X** value.
P2 = Theoretical **Z** value.
P3 = Safety distance (in radius).
P4 = Probing feedrate.
P5 = Tolerance (in radius).
P6 = Corrector number for the tool to be calibrated.

With this cycle, besides doing everything described before for the part measurement cycle for the **X** axis (N2), The CNC will correct the **I** value of the corrector number specified by **P6**.

This correction will only take place when the measurement error (P93/2) is equal to or greater than the tolerance specified by **P5**.

N5. PART MEASUREMENT AND TOOL CALIBRATION CYCLE FOR THE Z AXIS

Cycle programming format:

G75 N5 P1=K— P2=K— P3=K— P4=K— P5=K— P6=K—

G75 N5 = Part measurement and tool calibration cycle code for the **Z** axis.
P1 = Theoretical **X** value.
P2 = Theoretical **Z** value.
P3 = Safety distance.
P4 = Probing feedrate.
P5 = Tolerance.
P6 = Corrector number for the tool to be calibrated.

With this cycle, besides doing everything described before for the part measurement cycle for the **Z** axis (N3), The CNC will correct the **K** value of the corrector number specified by **P6**.

This correction will only take place when the measurement error (P95) is equal to or greater than the tolerance specified by **P5**.

6.27. DIGITIZING WITH THE FAGOR 8025/30 TS CNC

6.27.1. 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.

6.27.2. Characteristics of digitizing with the FAGOR 8025/30 TS 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 **TS** 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 , (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).

Also, zero offsets can be applied using **G53...G59** type functions.

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.

6.27.3. G76. Automatic block generation

This feature is only available on the **TS** model.

This function is used to generate blocks that are automatically loaded into the CNC or to a computer (via DNC). If the new program is going to be loaded into the CNC, a block of the type **G76 P5** must be previously written.

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

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

The programming format is:

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

The contents of the block to be created are similar to the normal programming except that the preparatory functions **G22** and **G23** cannot be programmed. After **G76**, the coordinates can be programmed in different ways:

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

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

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

In the same way, if we program in the contents after **G76**:
FP2 or **SP2**, the CNC will load the **F** or the **S** in the new program with the values of the parameter at that moment.

Example: Let us assume that the **X** coordinate of the point where the machine is situated is **78.35**. If we execute 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 N20 the parameter values are: P90=1250 and P55=2500, and in block N40, 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 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 trajectory 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.

6.27.4. 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 is provided 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 probing cycles.

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.

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 (4th), (3rd), **X, 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 communications program, 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.

P606 bit 6 indicates the type of impulse (+ or -).
P710 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).

6.27.5. Examples of using G76.

1. Example G76: PATTERN DIGITIZING

Creation of a program by copying the points of a part with a measuring probe (G75).

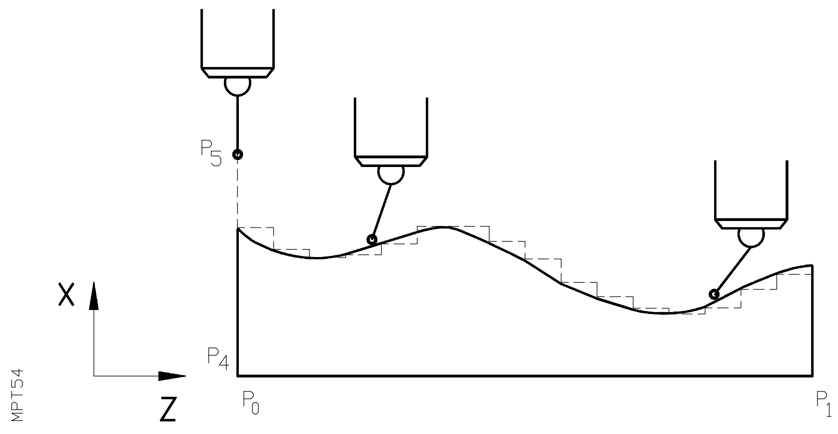
Calling parameters:

P0 = Minimum Z value to sweep.
P1 = Maximum Z value to sweep.
P4 = Minimum X value to sweep.
P5 = Maximum X value to sweep.
P6 = Maximum step value on Z.

Parameters used for calculations

P8 = X limit for G75.
P9 = Number of steps on Z.
P11= Starting point's Z value.
P13= Starting point's X value.
P14= Step counter for Z axis.

```
%00076

N10  G76 N12345 ............................................     (Program to be loaded into
    computer)

N20  G76 G1 F500

N30  P0=K— P1=K—         .....................................      (Parameter definition)
         P4=K— P5=K— P6=K—

N40  P8=P1F2P0 P9=P8F4P6 P10=F12P9 P9=F11P10

N50  G26 N80

N60  P9=P10F1K1 P6=P8F4P9 ..................  (P6=step Z, P9=No. of steps on Z)

N80  P11=Z P13=X P8=P4F2K1 ...............  (P8=X limit for G75)

N90  G0 G5 G90 XP5 ZP0

N100 P14=K0 .................................................      (P14=Step counter on Z)

N110 G90 G75 XP8 ........................................   (Probing on X)

N120 G76 X Z .................................................     (Load values)

N130 G0 XP5 .................................................      (Withdrawal on X)

N140 P14=P14F1K1 P9=F11P14 ...............  (Check for final point on Z)

N150 G28 N180

N160  GP1 ZP6 ................................................     (Next step on Z)

N170 G25 N110

N180 G0XP13ZP11 ........................................      (Back to initial point)

N190 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 X— Z—
N102 X— Z—
N103 X— Z—
N—  X— Z—
  Etc.
```

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.

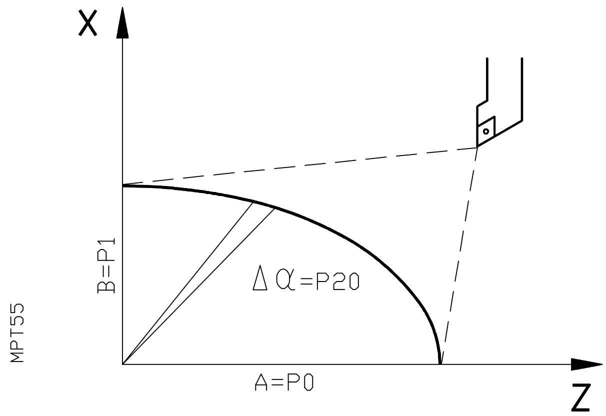
With these functions is possible to smoothen the point-to-point machining profile.

2. Example G76. CALCULATION OF POINTS WHEN THE MATHEMATICAL FUNCTION IS KNOWN.

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 **XZ** coordinates of the various points that compose the ellipse are calculated according to the formula:

Z = P0 SIN P3
X = P1 COS P3

Let us suppose that the tool's starting point is **X27 Z43** and the **X** axis is programmed in radius. The calculation program is **P761**, shown below:

```
N20  G76 P00098
N30  P0=K37 P1=K22 P3=K90 P20=K-0.5
N40  P4=F7P3 P5=F8P3 P6=P0F3P4 P7=P1F3P5
N50  G76 G0 G5 XP7 ZP6 (ellipse's starting point)
N60  P3=P3F1P20 P4=F7P3 P5=F8P3  P8=POF3P4  P9=P1F3P5
N70  P3=P3F1P20 P4=F7P3 P5=F8P3 P10=P0F3P4 P11=P1F3P5
N80  G76 G1 G9 XP11 ZP10 IP9 KP8 F250
N90  P3=P3F1P20 P4=F7P3 P5=F8P3 P10=P0F3P4 P11=P1F3P5
N100 G76 G8 XP11 ZP10
N110 P99=K176
N120 G25 N90.100.P99
N130 G76 G0 X27 Z43
N140 M30
```

When executing this program in **DRY RUN** program **P00098** is generated and loaded into the CNC memory for later machining:

```
N100 G0 G5 X— Z—
N101 G1 G9 X— Z— I— K— F250
N102 G8 X— Z—
N103 G8 X— Z—
N104  "    " N —  "    "
N?     G0 X27 Z43
```

6.28. G90 G91. ABSOLUTE AND INCREMENTAL PROGRAMMING

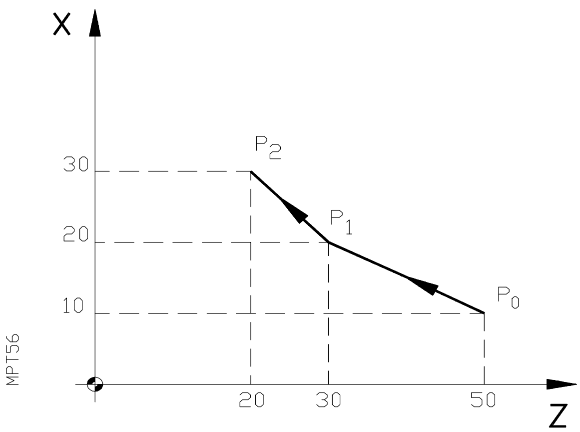
The programming of the coordinates of a point, may be carried out, either in absolute coordinates **G90** or in incremental coordinates **G91**.

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.

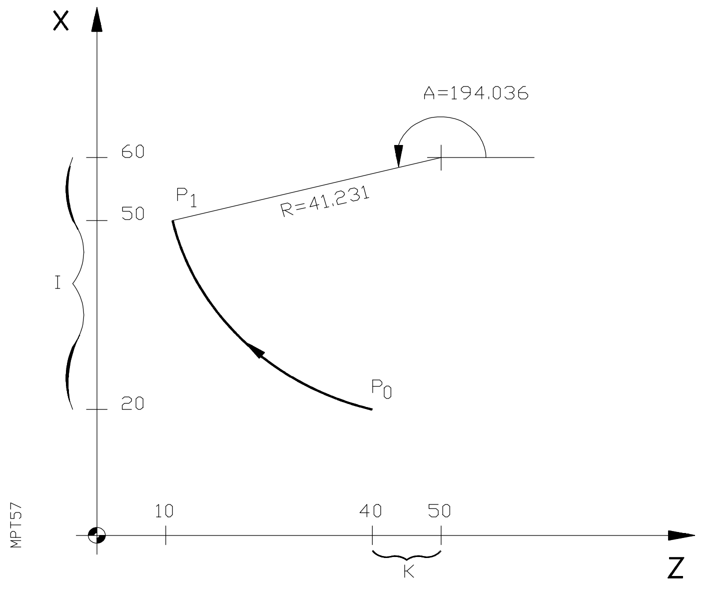


Absolute programming G90

N100 G90 G01 X40 Z30 P0 --> P1
N110 X60 Z20 P1 --> P2

Incremental programming G91

N100 G91 G01 X20 Z-20 P0 --> P1
N110 X20 Z-10 P1 —> P2



Starting point P0(X40 Z40)

Absolute programming G90

N100 G90 G02 X100 Z10 I40 K10

or

N100 G90 G02 X100 Z10 R41.231

Incremental programming G91

N100 G91 G02 X60 Z-30 I40 K10

or

N100 G91 G02 X60 Z-30 R41.231

6.29. G92. PRESETTING OF COORDINATE VALUES AND SETTING OF MAX. S VALUE AT CONSTANT SURFACE SPEED G96

Function **G92** can be used to preselect any value on the axes of the CNC, which involves being able to shift the coordinate origin. It can also set the max. Spindle speed, when operating on **G96** (constant surface speed).

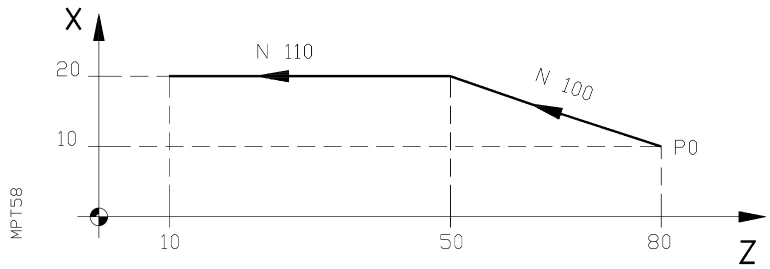
a) Presetting of coordinate values

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.

The order of the axes in the programming will be:

N4 G92 4th __ 3rd__ X__ Z__ .

Example: X in diameters and the starting point is P0(X20 Z80)



The program to describe the path drawn will be:

N100 G01 G90 X40 Z50
N110 Z10

When using the function **G92**, the program will be:

N90 G92 X20 Z0 ...... (The point X20 Z0 replaces the point P0)
N100 G90 X40 Z-30
N110 Z-70

No other function can be programmed in the block in which **G92** is programmed.

Coordinate values presetting by **G92** is referred always to the theoretical position in which the axes are

b) Setting of max. spindle **S** value when working at Constant Surface Speed (G96).

By means of the block **N4 G92 S4** the max. spindle speed is limited to the value set by **S4** (in rpm).

The CNC calculates at all times the spindle rpm required to achieve the programmed constant surface speed in meters/minute or feet/minute.

If the calculated spindle rpm is greater than the maximum set by G92 S4, the CNC will limit it to this maximum value.

6.30. G93. Polar origin preset

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 K+/-4.3 (always absolute coordinate values).
or, G93 I+/-3.4 K+/-3.4

I+/-4.3: Indicates the value of the abscissa
I+/-3.4: of the polar coordinate origin; i.e. the value of X.

K+/-4.3:Indicates the value of the ordinate
K+/-3.4:of the polar coordinate origin; i.e. the value of Z

No more information can be programmed in this block, when programming the presetting of polar origin in this way.

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.

<u>Atention:</u>

When a circular interpolation is programmed with **G02**,**G03**, the CNC assumes the arc’s center as the new polar origin.

On power-up or after **M02**,**M30**,**EMERGENCY** or **RESET**, the CNC takes the point (X0,Z0) as the polar origin.

6.31. G94. FEEDRATE F IN mm/min (inches/min)

When the code **G94** is programmed, the CNC assumes that the values entered by **F4** are in mm/minute (inches/10 minutes).

G94 is modal; i.e. it remains active until **G95**,**M02**,**M30**, **EMERGENCY** or **RESET**, are programmed.

6.32. G95. FEEDRATE F IN mm/rev. (inches/rev.) (Encoder required on the spindle)

When the code **G95** is programmed, the CNC assumes that the values entered by **F3.4** are in mm/rev. In inches the format is **F2.4** and the maximum programmable value is 19.685 inches/rev.

G95 is modal; i.e. it remains active until **G94** is programmed.

The CNC assumes **G95** on being turned on or after **M02**,**M30** or a general **RESET**.

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**.

6.33 G96. S SPEED IN m/min. (feet/min.) AT CONSTANT SURFACE SPEED (Encoder required on the spindle)

When the code **G96** is programmed, the CNC assumes that the values entered by **S4** are in m/minute (feet/minute) and the lathe operates in constant surface speed mode.

The CNC assumes as spindle working range the one currently selected. If no range is selected, the desired range (M41, M42, M43, M44) must be programmed in the same block.

It is recommended that **G96** and **S4** spindle speed be programmed in the same block. If G96 is programmed alone, the CNC assumes as Constant Surface Speed the last one used in that mode. If none was previously used, the CNC will issue error 10.

If the first movement after **G96** is made in rapid (G00), to calculate spindle revolutions, the CNC assumes as part diameter the one at the end of this movement.

If the first movement after **G96** is made in G01, G02 or G03, the CNC assumes as part diameter, the value at the time G96 is executed.

To calculate the number of rev./minute, the CNC will assume as diameter the actual value when starting **G01**,**G02** or **G03**.

G96 is modal; i.e. it remains active until **G97**, **M02**, **M30** is programmed, **EMERGENCY** or **RESET**

6.34. G97. S SPEED IN rev./minute

When the code **G97** is programmed, the CNC assumes that the values entered with **S4** are in rev./minute.

If **S4** is not programmed in the same block as **G97**, the CNC will take as programmed value, the speed at which the spindle is actually running.

G97 is modal; i.e. it remains active until **G96** is programmed.

The CNC assumes **G97** on being turned on, after **M02**,**M30**, **EMERGENCY** or **RESET**.

7. COORDINATE PROGRAMMING

A point can be programmed in the CNC by using:

. Cartesian coordinates
. Polar coordinates
. Two angles
. One angle and one cartesian value

7.1. CARTESIAN COORDINATES

7.1.1. Linear axes.

The format for linear axes dimensions is:

. In mm
4th +/- 3rd+/-4.3 X+/-4.3 Z+/-4.3

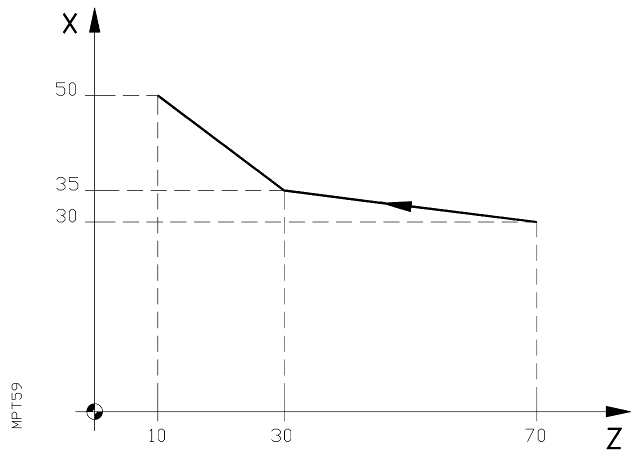
. In inches
4th +/- 3rd+/-3.4 X+/-3.4 Z+/-3.4

In other words, the axis coordinate values are programmed by the letters which correspond to the 4th and 3rd axes, i.e., **W,Y,C**, defined in the machine parameter, as well as the letters **X,Z** followed by the coordinate values.

The coordinate values programmed will be absolute or incremental depending on whether **G90** or **G91** is programmed.

There is no need to write the **+** sing in the case of positive coordinate values. The leading and trailing zeros of coordinate values may be omitted.

Example: X in diameters and the starting point is (X60 Z70)



Absolute coordinate values

N100 G90 X70 Z30
N110 X100 Z10

Incremental coordinate values

N100 G91 X10 Z-40
N110 X30 Z-20

7.1.2. Rotary axis

By means of machine parameters it is possible to determine whether the 4th axis or the 3rd axis or both, are Rotary or Linear.

Likewise, should they be a Rotary axis it is possible to define whether they are a Rollover Axis or not (programming between +/- 360 degrees).

Type	4th axis	3rd axis
ROTARY	P 615(1) = 1	P 613(1) = 1
ROLLOVER	P 615(2) = 1	P 613(2) = 1

4th axis

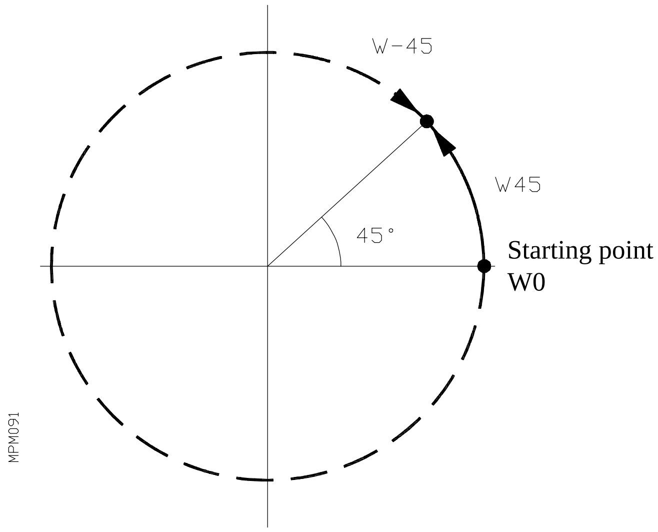
If the 4th axis is rotary P615(1)=1 and parameter P615(2) is set to 0 a max. value of +/- 8388.607 degrees can be programmed both in absolute coordinates **G90** and relative coordinates **G91**. Lower limits can be set by the machine parameters.

Programming is identical to linear axes.

If P615(2)=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.

(Let us assume that the 4th axis is called W)



7.2. POLAR COORDINATES

The format of the axis coordinate values is as follows:

. In mm
R+/- 4.3 A+/-3.3

. In inches:
R+/-3.4 A+/-3.3

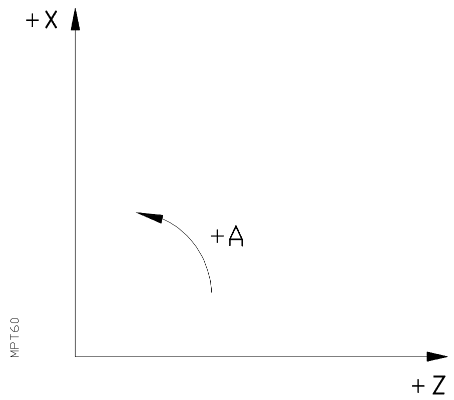
R being the value of the radius and **A** the value of the angle (A in degrees), referred to the polar center.

When turning on and after **M02,M30 EMERGENCY** or **RESET**, the CNC takes the point **X0 Z0** as polar origin. The polar origin can be altered by **G95**.

The values of **R** and **A** will be absolute or incremental depending on whether **G90** or **G91** are active.

When programming rapid (G00) or linear interpolations (G01), the values of **R** and **A** must be entered.

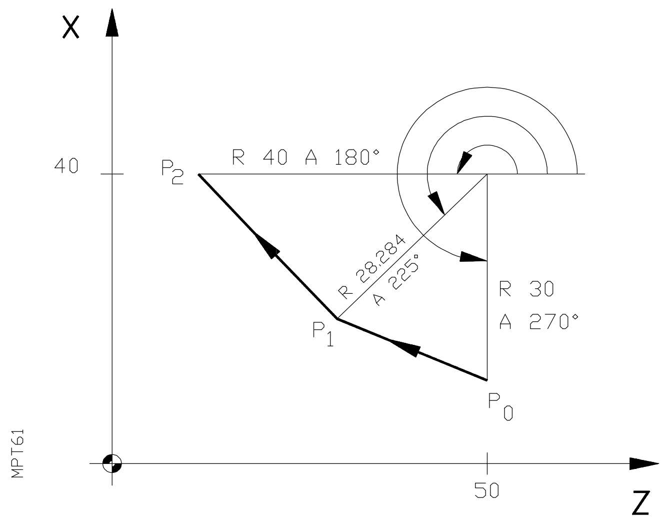
When a circular interpolation (G02,G03) is programmed, the values of the angle **A+/-3.3** of the arc's final point and the values of the arc's center referred to the starting point must be entered.



When working with polar coordinates, the center of the circle on circular interpolations (G02,G03) is defined with **I,K** same as working with cartesian coordinates.

When a circular interpolation (G02,G03) is programmed the CNC takes the arc's center as the new polar origin.

Example 1: X in diameters



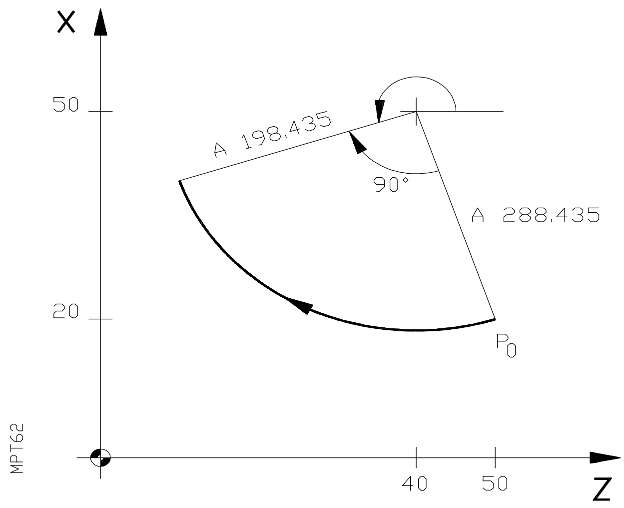
In absolute coordinate values G90

N100 G93 I80 K50 .... Presetting of polar origin
N110 G01 G90 R30 A270 .... P0
N120 R28.284 A225 ....P1
N130 R40 A180 .... P2

In incremental coordinate values G91

N100 G93 I80 K50 .... Presetting of polar origin
N110 G01 G90 R30 A270 .... P0
N120 G91 R-1.716 A-45 .... P1
N130 R11.716 A-45 .... P2

Example 2. Let us suppose that the starting point is X40 Z50.7



In absolute coordinate values G90

N100 G90 G02 A198.435 I30 K-10

or

N100 G93 I100 K40
N110 G90 G02 A198.435

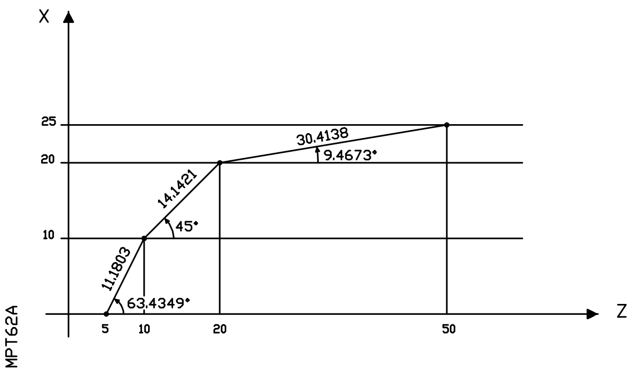
In incremental coordinate values G91

N100 G91 G02 A-90 I30 K-10

or

N100 G93 I100 K40
N110 G91 G02 A-90

Example 3: X in radius



In absolute coordinate values G90

```
N10 G00 X0 Z5
N20 G93 G01 F1000 R11.11803 A63.4349
N30 G93 R14.1421 A45
N40 G93 R30.4138 A9.4623
N50 M30
```

7.3. TWO ANGLES (A1,A2)

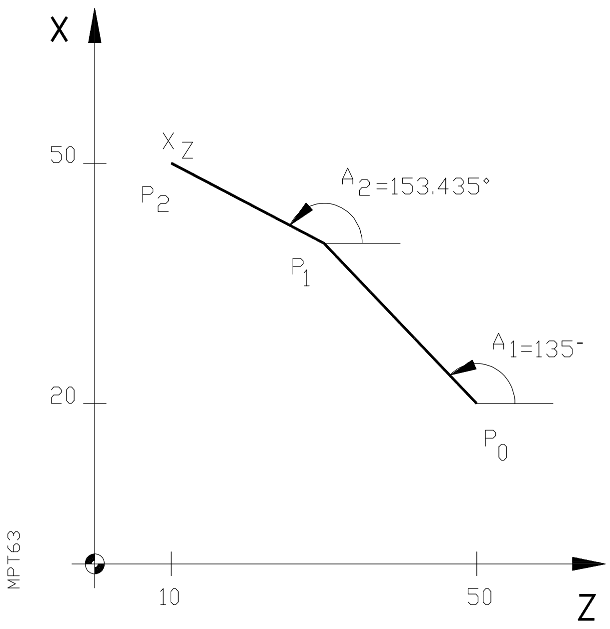
An intermediate point in a path can also be defined by: A1 A2 (X,Z).

Where **A1** is the departure angle from the starting point of the path **P0**.

A2 is the departure angle of the intermediate point **P1**.

(X,Z) are the coordinates of the final point **P2**.

The CNC calculates automatically the coordinates of **P1**.

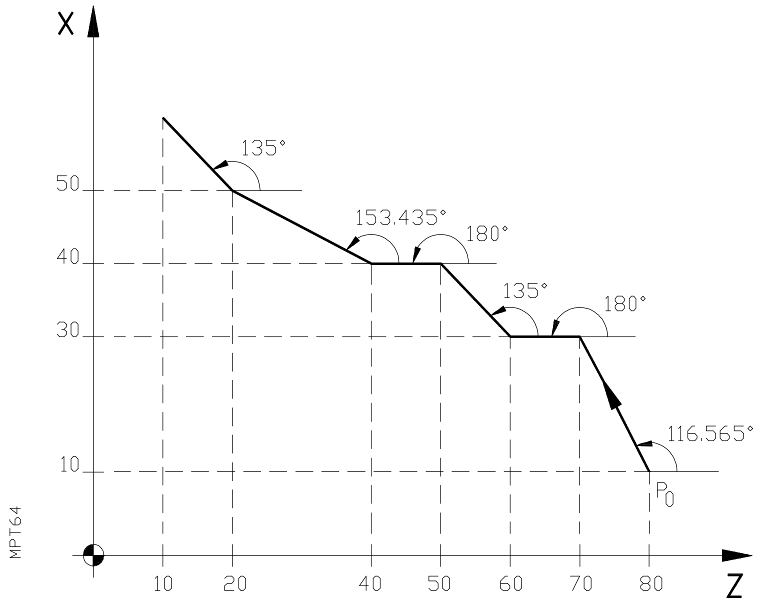


Let us suppose that the starting point is **P0** (X40 Z50) and the **X** axis is in diameters.

N100 A135 A153.435
N110 X100 Z10

7.4. 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.

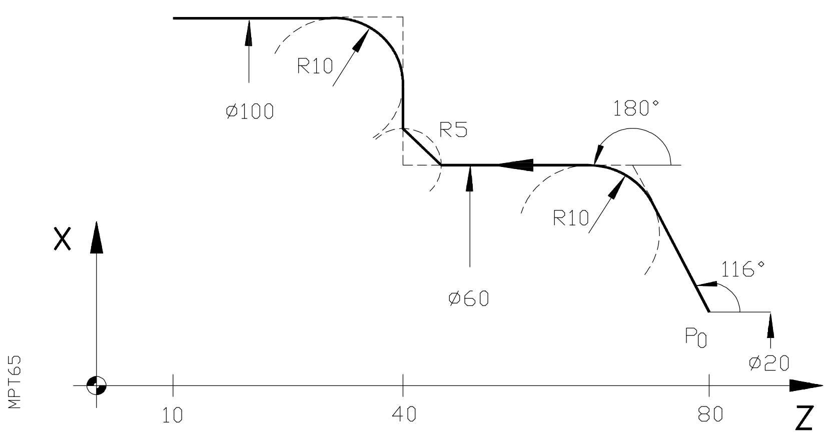


Let us suppose that the starting point is **P0** (**X20 Z80**) and the **X** axis in diameter.

In absolute coordinate values In incremental coordinate values

N100 G90	N100 G91
N110 A116.565 X60	N110 A116.565 X40
N120 A180 Z60	N120 A180 Z-10
N130 A135 X80	N130 A135 X20
N140 A180 Z40	N140 A180 Z-10
N150 A153.435 X100	N150 A153.435 X20
N160 A135 Z10	N160 A135 Z-10

In the definition of the points by two angles or one angle and one coordinate value, it is possible to insert roundings, chamfers, tangential entries and exits.



Starting point P0 (X20 Z80)

N100 G01 G36 R10 A116 A180
N110 G39 R5 X60 Z40
N120 G36 R10 A90 X100
N130 A180 Z10

Note: On a cylindrical grinder. Put the spindle encoder on the part rotation to obtain feedrate per revolution of the part.

8. (F) PROGRAMMING THE FEEDRATE

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:

	Programming format	Programming unit	Minimum value	Maximum value
G94	F 4	F1= 1mm/min	F1 (1 mm/min)	F9999 (9999 mm/min)
G95	F3.4	F1= 1mm/rev.	F0.001 (0.001 mm/rev.)	F500.0000 (500 mm/rev.)

Inch programming:

	Programming format	Programming unit	Minimum value	Maximum value
G94	F 4	F1= 0.1 inch/min	F1 (0.1"/min)	F3937 (393.7 inch/min)
G95	F 3.4	F1= 1 inch/rev.	F0.0001 (0,0001"/rev.)	F19.6850 (19,6850 inch/rev.)

When operating in inches and with rotary axes, we recommend setting machine parameter P618(2) to "1" so the programming units in G94 are in degrees/minute.

	P618(2)	Only rotary axis	Interpolation of rotary and linear axes
G94	P618(2)=0	F1= 2.54°/min	F1= 1 inch/min
	P618(2)=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**. For example: On a machine with a maximum programmable working feedrate of 1000 mm/min. it makes no difference whether **F1000** or **F0** is programmed.

The programmed feedrate **F** is effective when operating on linear interpolation (G01) or circular interpolation (G02/G03).

Should the **F** function not be programmed, the CNC will assume the **F0** feed.

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 **P600(3)** by means of the knob on the front panel of the CNC as long as it is not executing a threading operation with any of the functions **G33, G86, G87** during probing movements (G57).

Notes: If a very slow feedrate is required, use G95.

. If no spindle encoder has been installed (such as on a surface grinder), connect an encoder to a 1 rpm gear motor (clock motor for example) to obtain feedrate per minute in G95.

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 as long as threadcutting with any of the functions **G33**, **G86** or **G87** as not being carried out.

When working with **G96** the possible **S** values are:

S0-S3047 (0 mm/min, 3047 m/min)
S0-S9999 (0 mm/min, 9999 m/min)

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 **P606(2)** and **P702** until the point identified by **S4.3** is reached.

10. (T) TOOL PROGRAMMING

The tool to be used is programmed by means of code T2.2

- **Tool number**. The two digits to the left of the decimal point 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 turret, and may be limited to a value lower than 99 according to the machine parameter. This value is used to select the required tool.

- **Tool compensation** (tool offset table). The two digits to the right of the decimal point may have any value between 01 and 32. With these figures the desired values are selected in the tool offset table.

As soon as T2.2 is read, the CNC applies the **X,Z,I,K** values stored in the table except when P604(5) is 1, in which case these values will be applied after an **M06**.

When **G41** or **G42** is programmed, the CNC applies the value stored at the programmed T address (01-32) as radius compensation value.

If no **T** has been programmed, the CNC applies the address T00.00, which corresponds to a tool of zero dimensions.

The following values are recorded in every tool offset table address (01-32).

X : Tool length along **X** axis.
Z : Tool length along **Z** axis.
F : Location code.
R : Tool nose radius.
I : Tool wear offset along **X** axis. This value must always be entered in diameters.
K : Tool wear offset along **Z** axis.

The max. values are:

X,**Z** (tool length) +/-8388.607 mm (+/-330.2599 inches).

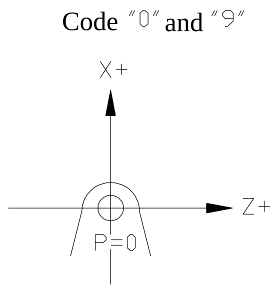
I,**K** (tool length offset) +/-32.766 mm (+/-1.2900 inches).

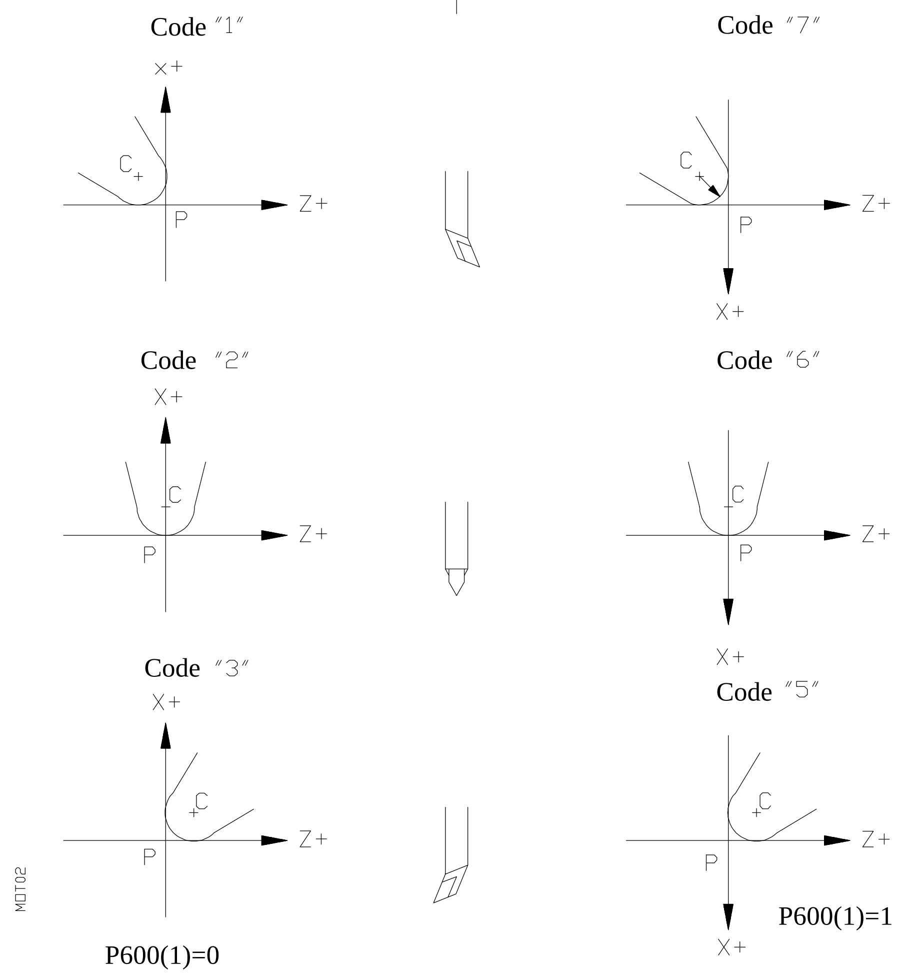
R (Radius) 1000.000 mm (39.3700 inches).

To offset the tool nose radius the location code (F) of the tool must also be stored. Possible codes are: **F0-F9** (see figure).

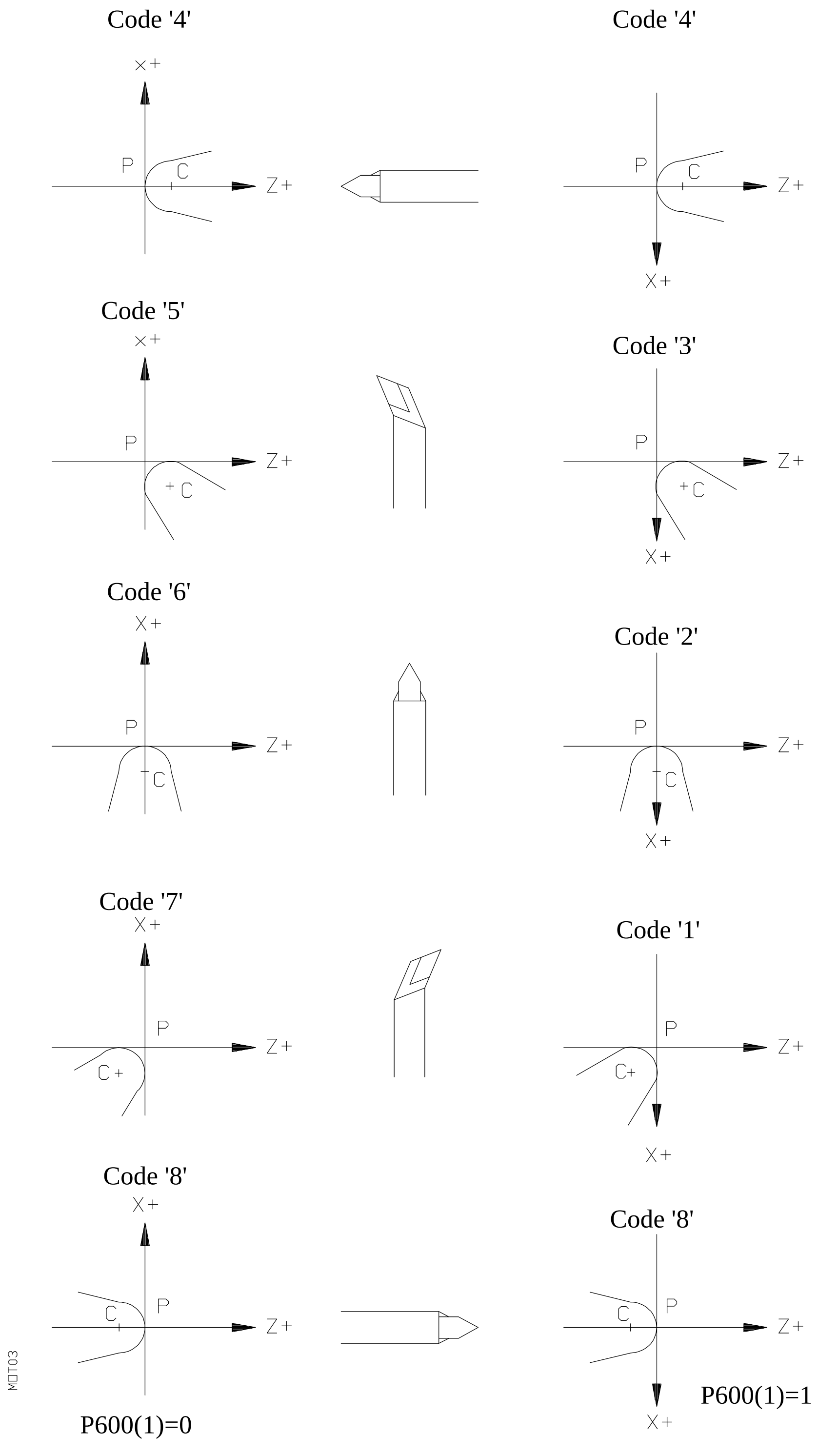
LOCATION CODES

Refer to section 6.15 on page 48 to review the axis orientation, parameter P600(1) will advise you which set of tables to use.





P: Tool tip
C: Tool center



Code '4'
Code '4'
Code '5'
Code '3'
Code '6'
Code '2'
Code '7'
Code '1'
Code '8'
Code '8'
X+
Z+
P
C
MOT03
P600(1)=0
P600(1)=1

11. M. MISCELLANEOUS FUNCTIONS

The miscellaneous functions are programmed by means of code **M2**.

96 different miscellaneous functions (M00-M99) can be programmed except **M41,M42,M43,M44** which are outputs automatically based on the spindle speed programmed and the spindle ranges defined by parameters P7, P8, P9, P10 if P601(1) is set to 1. If the parameter is set to **0, M41,M42,M43** must be programmed. The miscellaneous functions come out in **BCD** code.

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 which 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 an 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 then consecutively in the order in which they are programmed.

Some of the 100 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. Press the **Cycle Start** key to continue.

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 conditional 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**.

11.5. M03. CLOCKWISE START OF THE SPINDLE

This code means that the spindle starts running clockwise. 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-CLODWISE 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. M19. SPINDLE ORIENTATION

If M19 S4.3 is programmed, the spindle will rotate at the speed and direction set by machine-parameters P606(2) and P706 and will stop at the point identified by S4.3 in degrees, referred to the reference zero marker.

When the spindle is positioned within the dead band (P707), the CNC sends out the spindle blocking signal (decoded M15) and the spindle is kept in closed loop; gain as per P708, min. analog voltage as per P709.

When programming in a block M19 S4.3, more information is not allowed in that block.

Machine parameters P906 and P907 determine the lower and upper limits, respectively, of the spindle travel, with **M19**.

11.9. M41,M42,M43,M44 SPINDLE RANGE SELECTION

When P601(1) has been set to 1, these codes are automatically generated by the CNC, when an **S** function is programmed. If this parameter is **0, M41,M42,M43** and **M44** must be programmed.

When operating at constant surface speed (G96) these functions must necessarily be programmed even if P601(1) is set to 1.

11.10. M45. SELECTION OF ROTATION SPEED OF THE LIVE TOOL AND THAT OF THE SYNCHRONIZED TOOL.

There are two formats to program an **M45** function:

a) <u>Live tool</u>

Programming format: **N4 M45 S+/-4**

S+/-4 defines the direction and rotation speed of the live tool.

The **+/-** sign defines the direction of turn, with **S+4**, it will turn in one direction and with **S-4** will turn in the opposite direction.

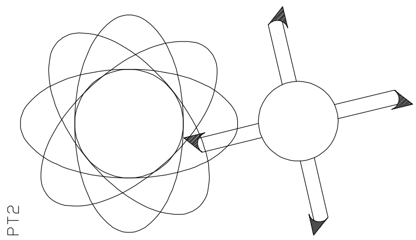
The programmable value range is between **S0** and S9999 (0-9999 r.p.m.).

b) Synchronized tool with the rotation speed of the spindle

A live tool is slaved and synchronized to the spindle encoder.

A synchronized tool is defined as being one formed by several blades, in a specific relation. This tool is rotary and turns with respect to the speed of the spindle.

Its primary use is for making polygon shapes on the part, such as changing the part into a cuadrangular prism, hexagon, octagon, etc.



EXAMPLES:

. TOOL WITH FOUR BLADES AND PART/TOOL TURNING RATIO OF 1:2.
THE RESULTING POLYGON SHAPE IS AN OCTAGON (8 SIDES).

. WITH A RATIO OF 1:1 THE RESULT IS A SQUARE (4 SIDES).

. WITH A RATIO OF 2:3 THE RESULT IS A HEXAGON (6 SIDES).

The formula for calculating the number of the resulting polygon is as follows:

number of sides = number of blades x **K** factor.

Another way to say it is to deal with a second spindle by means of which, and by synchronizing the speeds of the heads, we can transfer the part from one spindle to the other.

To take advantage of this feature, an encoder must be connected to the live tool.

Programming format: **N4 M45 K+/-3.**4

The +/- sign defines the direction of turn, with **K+4**, it will turn in one direction and with **K-4** will turn in the opposite direction.

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

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

Example:

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

N - P1=K1 F4 K3
N - M45 KP1

If the rpm's are greater than the limit established by machine- parameter **P802**, the CNC will display error **17**. Also, when the following error of the synchronized tool is too large, the CNC will display error **71**.

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

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

c) Machine parameters associated with the synchronized or live tool.

Parameters to be borne in mind as as follows:

P802 indicates the maximum number of rpm of the synchronized tool(when exceeded, the CNC produces error 17). Its maximum number is 9999.

P803 indicates the number of impulses/encoder rotation of the live tool.

P609 bit 8 indicates whether we can vary the speed of the live tool by means of the **SPEED RATE** (between 50% and 120%).

P711 defines the gain of the synchronized tool. (acceleration/ deceleration).

P607 bit 2 indicates the counting direction of the synchronized tool.

P607 bit 1 indicates the analog sign of 1 to the synchronized tool.

Atention:

If these last two parameters are not well coordinated the CNC immediately produces the following error as soon as the synchronized tool begins to move.

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 (standard or parametric) 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. PARAMETRIC SUBROUTINES

It is basically similar to an standard subroutine except for the fact that values can be assigned to up to 15 parameters in the call block (G21).

When the execution of the parametric subroutine is finished (G24), the values assigned to the parameters in the call block are recovered even though different values might be allocated to them during the subroutine.

12.3.1. Identification of a parametric subroutine

A parametric subroutine always starts by **G23**.

The structure of the first block is:

N4 G23 N2

N4 : Block number

G23 : Defines the start of a parametric subroutine

N2 : It identifies the parametric subroutine (it may be any number between **00** and 99).

Atention:

Two parametric subroutines with the same number cannot co- exist in the CNC's memory, even if they are included in different programs. Two subroutines one standard the other parametric can be identified by the same number.

A parametric subroutine must necessarily end with a block of the format: **N4 G24**.

N4 : Block number
G24 : It defines the end of a subroutine (standard or parametric).

No additional information can be programmed in this block.

12.3.2. 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 P3=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.

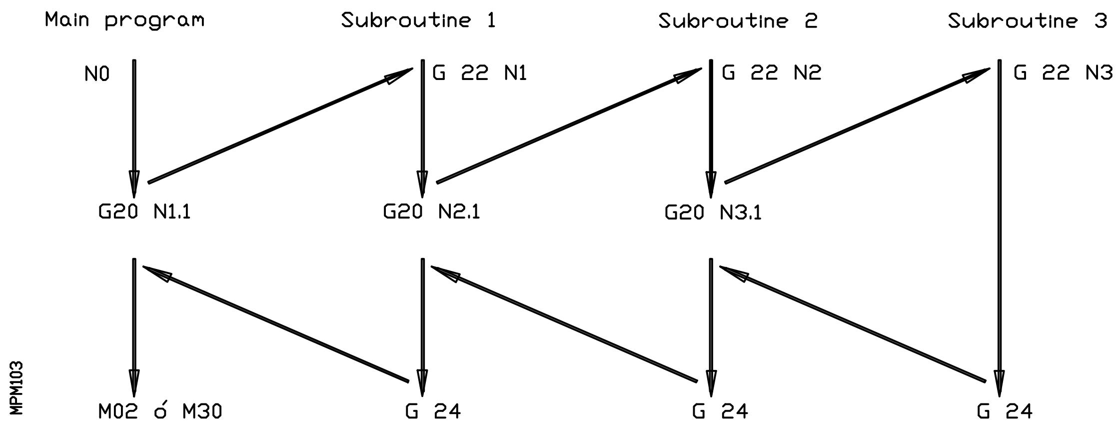
P3 : Value of the parameter.

K+/-5.5 : Value assigned to the arithmetic parameter.

12.4. 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



12.5. EMERGENCY SUBROUTINE

If machine parameter **P716** 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 **P716**.

13. PARAMETRIC PROGRAMMING. OPERATIONS WITH PARAMETERS

The CNC has 255 parameters (P0-P254) with which the programming of parametric blocks can be performed as well as different types of operations and jumps within a program. 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 $\sqrt{A^2+B^2}$
F7 : Sine
F8 : Cosine
F9 : Tangent
F10 : Arc tangent
F11 : Comparison
F12 : Entire part (Integer value without decimals)
F13 : Entire part plus one
F14 : Entire part minus one
F15 : Absolute value
F16 : Complementation (invert sign)
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
F34 : Special functions.
F35 : Special functions.

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

<u>Assignments</u>

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 = C

P1 takes the theoretical value of the actual position of the C axis.

d) N4 P1 = X

P1 takes the theoretical value of the actual position of the X 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 = Y

P1 takes the theoretical value of the actual position of the Y axis.

h) N4 P1 = R

P1 takes value 1 if the machine parameter P11 (rad/diameter) is in radii and vale 2 is in diameters.

i) N4P1 = T

P1 takes the actual value of the clock (execution time) in hundredths of a second. This assignation causes the cancellation of the radius compensation (G41 or G42).

j) N4 P1 - OX

P1 takes the theoretical coordinate of the X axis, with respect to the machine zero where the CNC is situated.

k) N4P1 = OC

P1 takes the theoretical coordinate of the C axis, with respect to the machine zero where the CNC is situated.

l) N4P1 = OZ

P1 takes the theoretical coordinate of the Z axis, with respect to the machine zero where the CNC is situated.

m) N4P1 = OW

P1 takes the theoretical coordinate of the 4th axis, with respect to the machine zero where the CNC is situated.

n) N4P1 = OY

P1 takes the theoretical coordinate of the 3rd 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 P611(6).

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.

o) 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. he 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 Sine

N4 P1 = F7 P2 —> P1 = Sine P2

The angle has to be programmed in degrees.

N4 P1 = F7 K5 —> P1 = Sen 5 degrees

F8 Cosine

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 a constant, 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 integer value of P2 as its value
N4 P1 = F12 K5.4 —> P1 = 5

F13 Entire part plus one

N4 P1 = F13 P2 —> P1 takes the integer value 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 integer value 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-F29 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 Z 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 memory address of the block before the one at P2.

F20 does not accept a constant as operand.

Example: P1 = F20 K4 is not valid.
P1 = F17K13 (for example 32492)
P2 = f20 P1 (this makes P2 = 32465, for example)
N12 Address: 32465
N13 Address: 32492

F21

N4 P1 = F21 P2

P1 takes the value of the I coordinate in the block located at P2.

F21 does not accept a constant as operand.

Example: P1 = F21 K2 is not valid.

F22

N4 P1 = F22 P2

P1 takes the value of the K coordinate in the block previous to the one defined by P2.

F22 does not accept a constant as operand.

Example: P1 = F22 K3. 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 X value of the tool table in the position 2.

Example b) N4 P8 = F24 P12

Parameter P8 takes the X 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 Z value of the tool table in the position 16.

Example b) N4 P13 = F25 P34

Parameter P13 takes the Z 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 P6 = F26 K32

Parameter P6 takes the F value of the tool table in the position 32.

Example b) N4 P14 = F26 P15

Parameter P14 takes the F value of the tool table in the position indicated by parameter P15.

F27

This function can be programmed in two different ways:

Example a) N4 P90 = F27 K13

Parameter P90 takes the R value of the tool table in the position 13.

Example b) N4 P28 = F27 P5

Parameter P28 takes the R value of the tool table in the position indicated by parameter P13.

F28

This function can be programmed in two different ways:

Example a) N4 P17 = F28 K10

P17 takes the I value of the position 10 in the tool table.

Example b) N4P19 = F28 P63

P19 takes the I value of the tool table position indicated by parameter 63.

F29

This function can be programmed in two different ways:

Example a) N4 P15 = F29 K27

P15 takes the K value of the position 27 in the tool table.

Example b) N4 P13 = F29 P25

P13 takes the K value of the tool table position indicated by parameter 25.

Any number of assignments and operations can be programmed in a block provided, however, that no more than 15 parameters are modified.

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 of 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

Special functions F34-F35

These functions do not affect the jump indicators.

F34

N4P1 = F34P2

P1 takes the value of the 3rd axis which appears in the block with the address P2.

F34 does not accept operating constant.

Example: P1 = F34K2 Is not valid.

F35

N4P1 = F35P2

P1 takes the value of the 4th axis which appears in the block with the address P2.

F35 does not accept constant operand.

Example: P1 = F35K3 Is not valid.

F36

N4P1 = F36

P1 takes, the value of the tool number being used at the time.

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.

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 (P1=k0 has no effect on the flag).

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 not equal to 0

When the CNC reads a block with G27, if the condition **is not 0** is met, it jumps to the block identified by N4 or N4.4.2, if the condition **is not 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.

G28 Conditional jump/call if smaller

When the CNC reads a block with the code G28, if the condition of smaller than 0 is met, it jumps to the block identified by N4 or N4.4.2. If the condition of smaller 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 of equal or greater is met, it jumps to the block identified by N4 or N4.4.2. If the condition of equal or greater 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 error code (see page 186, section 15.0 for possible error codes and messages).
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.

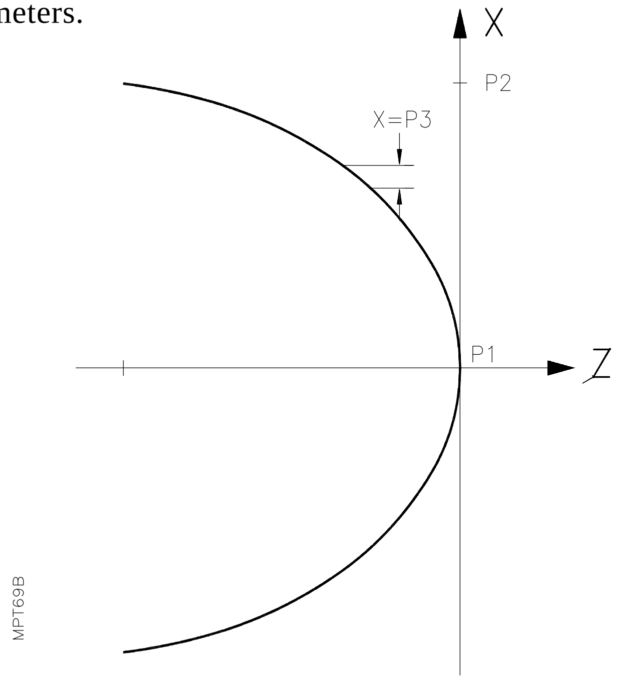
Remember also that the user can write comments in the program which will be displayed when the corresponding block is executed.

PROGRAMMING EXAMPLE TO DEFINE A PARABOLIC PATH

The formula for a parabolic path is:

$$\mathbf{Z = -KX^{2}}$$

The programming of the X axis is in diameters.



The call parameters are:

P0—> K
P1—> Initial X dimension
P2—> Final X dimension
P3—> Increase in X

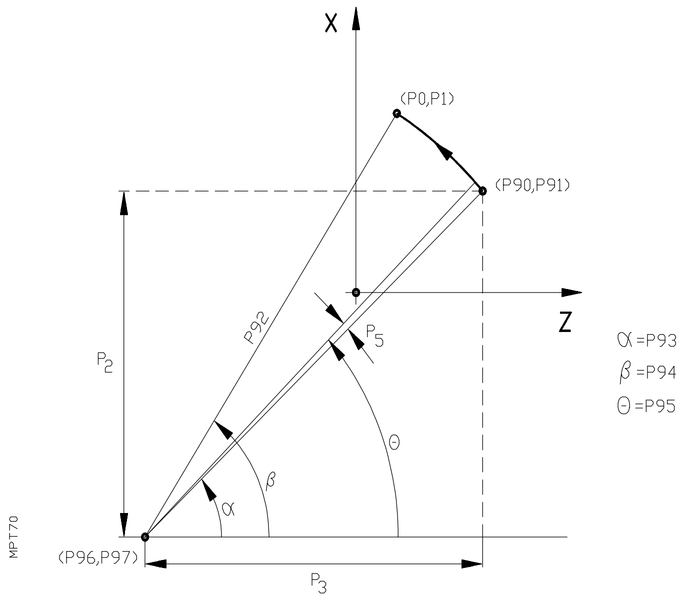
Calculated parameters:

P4—> X dimension
P5—> Z dimension

```
N80 G21 N56.1 P0=K0.01 P1=K00 P2=K100 P3 = K1
N90 M30
N110 G23 N56
N120 P4=P1 .................................. (X=X initial)
N130 P4=P4 F1 P3 P4=F11 P2
N140 G28 N160
N150 P4=P2
N160 P5=P4 F3 P4 P5 = P5 F3 P0 P5 = F16 P5
N170 G01 XP4 ZP5 ....................... (movement block)
N180 P4 = F11 P2
N190 G27 N130
N200 G24
```

EXAMPLE OF PROGRAM OF AN ARC WHOSE RADIUS IS GREATER THAN 8388.607 mm

If starting point is X2000 Z3000 and the following arc is programmed: G03 X3774.964 Z1000 I-7000 K-8000, 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

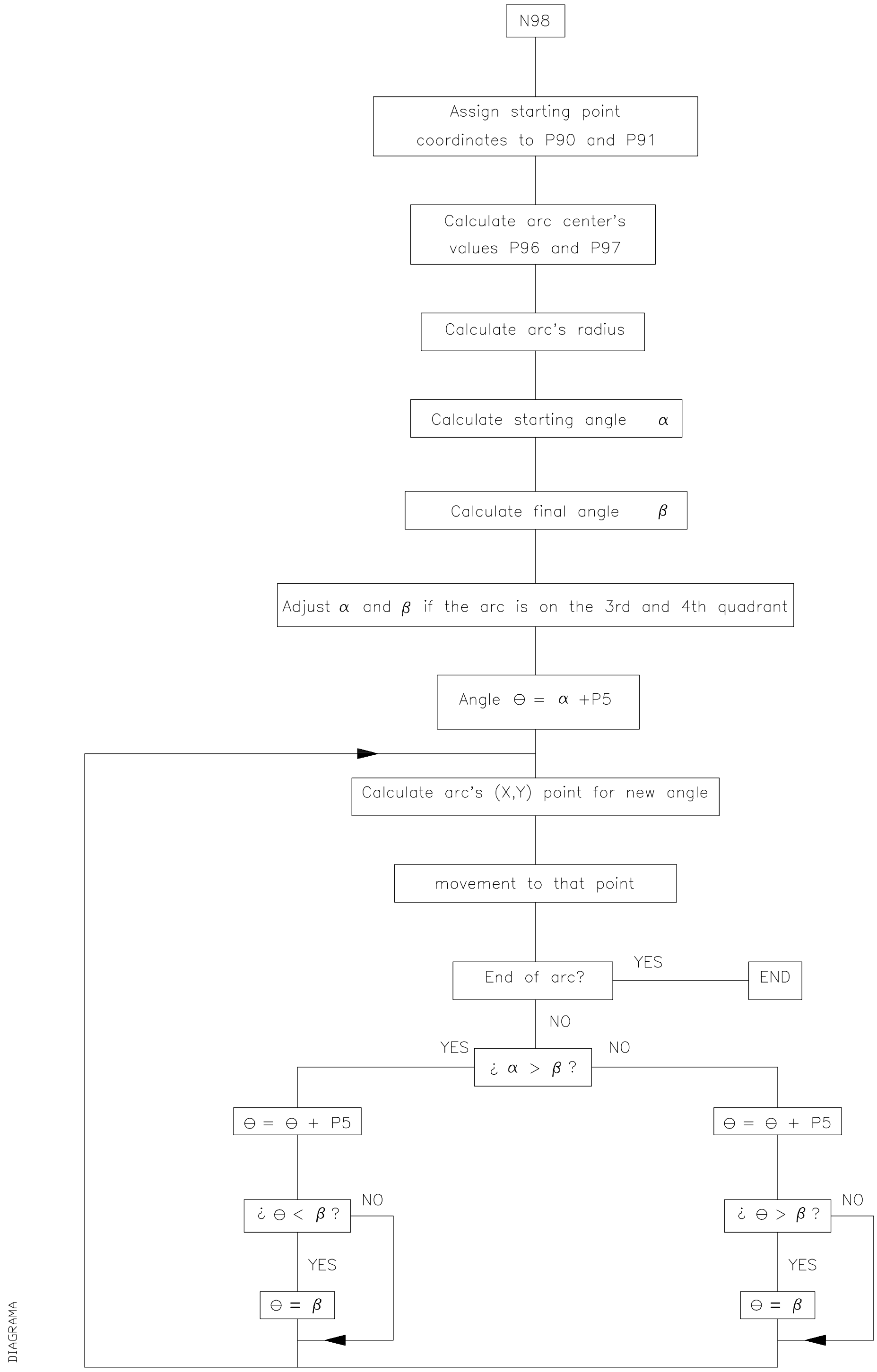
Call parameters

P0: X value of the final point
P1: Z 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 Z 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 Z value
P98: Calculations
P99: Calculations

Subroutines flow chart:



SUBROUTINE N98

```
N00  G23 N98
N01  P90=X P91=Y ........................................  (Takes point valu)
     P96=P90 F1 P2 P97=P91 F1 P3 ........................  (Calculates center)
     P92=P2 F6 P3 .......................................  (Calculates radius)
     P98=P2 F4 P3 P93=F10 P98 ...........................  (Calculates angle α)
     P98=P91 F2 P97 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 N21 ............................................  (Adjusts values of
N12  P94=F11 K0 .........................................    α and β if the
N13  G28 N21 ............................................  arc spares 3rd an4th
N14  P93=P93 F1 K360 ....................................  quadrant)
N15  G25 N21 ............................................
N16  P94=F11 K ..........................................
N17  G29 N21 ............................................
N18  P93=F11 K0 .........................................
N19  G28 N21 ............................................
N20  P94=P94 F1 K360 ....................................
N21  P95=P93 F1 P5 ......................................  (θ angle = α+ P5)
N22  P98=F8 P95 P98=P98 F3 P92 ..........................  (X value of point)
     P98=P98 F1 P96
     P99=F7 P95 P99=P99 F3 P92 P99=P99 F1 P97 ........  (Z 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=a β)
N30  G28 N32
N31  P95=P94 ............................................  (If θ = β as been
                                                           reached or surpased)
N32  G25 N22 ............................................  (calculates new point.)
N33  P95=P95 F1 P5  P94=F11 P95 .........................  (If α> β decrement of θ
                                                           and check whether =a 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:

Programming of X axis in radius

N10 P0=3774.964 P1=K1000 P2=K-7000 P3=K-8000 P4=K100 P5=K0.5
N20 G1 G41 X2000 Z3000 T1.1
N30 G21 N98.01

Programming of X axis in diameters

N10 P0=7549.928 P1=K1000 P2=K-7000 P3=K-8000 P4=K100 P5=K0.5
N20 G1 G41 X4000 Z3000 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.

14. CANNED CYCLES

The CNC includes the following canned cycles:

G66 - Pattern repeating
G68 - Stock removal along the X axis
G69 - Stock removal along the Z axis
G81 - Turning cycle with straight sections
G82 - Facing cycle with straight sections
G83 - Deep hole drilling
G84 - Turning cycle with arc sections
G85 - Facing cycle with arc sections
G86 - Threadcutting (Z axis)
G87 - Threadcutting (X axis)
G88 - Grooving (X axis)
G89 - Grooving (Z axis)

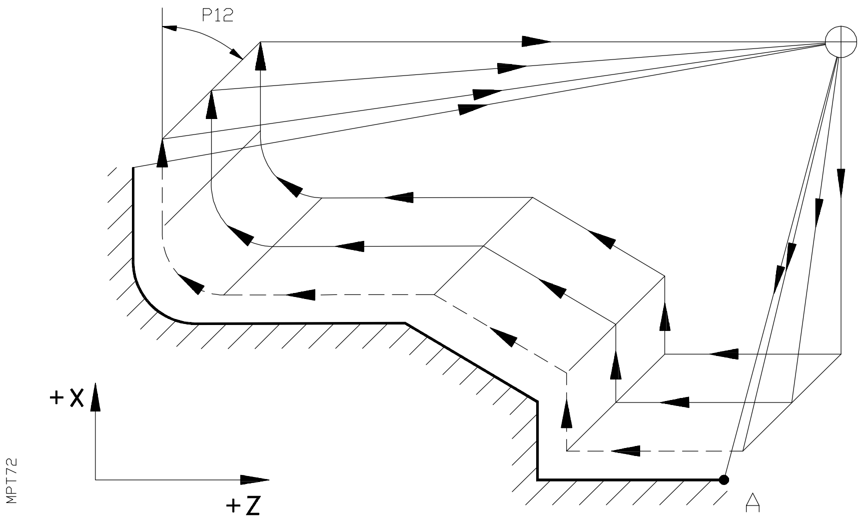
Atención:

The canned cycles do not alter the call parameters which can be used in subsequent cycles, although they alter the contents of the parameters P70 to P99.

If the value of a parameter is a constant, when programming canned cycles, it is necessary to key-in K after =.

Example: N4 G66 P0 = K25 ......

14.1. G66. PATTERN REPEATING



Format:

N4 G66 P0=K P1=K P4=K P5=K P7=K P8=K P9=K P12=K P13=K P14=K

Meaning of the parameters:

P0: X coodinate value of the initial point A, in radius or diameter.

P1: Z coordinate value of the initial point A.

P4: Total amount of stock to be removed. It must be equal to or greater than 0 and equal to or greater than the finishing stock allowance or error 3 will be displayed. According to P12 it will be identified as residual in X or Z.

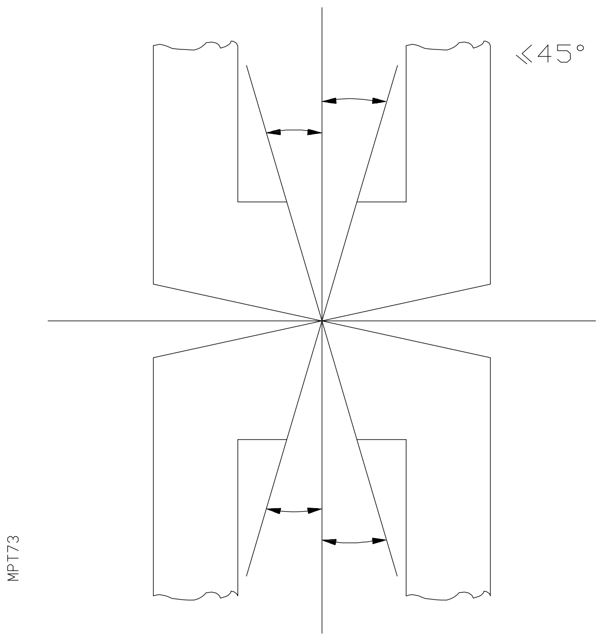
P5: Max. step. It must be greater than zero or error code 3 will be displayed. According to P12 it will be identified as step along X or Z axis. The real step calculated by the CNC will be equal or smaller than the max. step.

P7: Finishing stock allowance on the X axis. It must be equal to or greater than 0, or error 3 will be displayed.

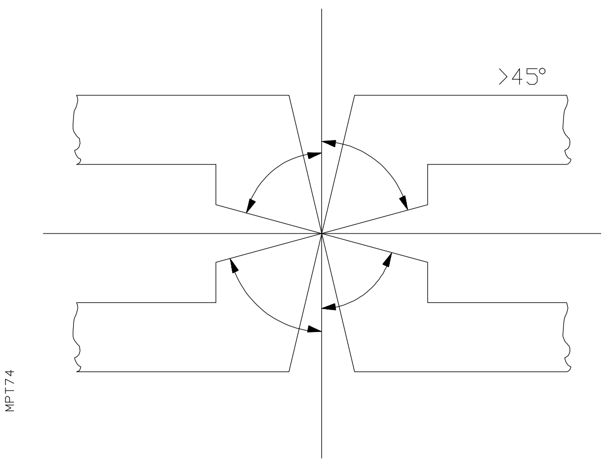
P8: Finishing stock allowance on the Z axis. It must be equal to or greater than 0, or error 3 will be displayed.

P9: Feedrate for the finishing pass. If it is = 0, there is no finishing pass. If it is negative, error code will be displayed.

P12: Tool entry/exit angle. Its value must be comprised between 0 degrees and 90 degrees or error 3 will be displayed. If it is equal or smaller than 45 degree P4 will be taken as residual stock on the X axis and P5 as max. step along X axis.



If it is greater than 45 degree, P4 will be taken as residual stock on the Z axis and P5 as max. step along Z axis.



P13: Number of the first block to define the pattern.

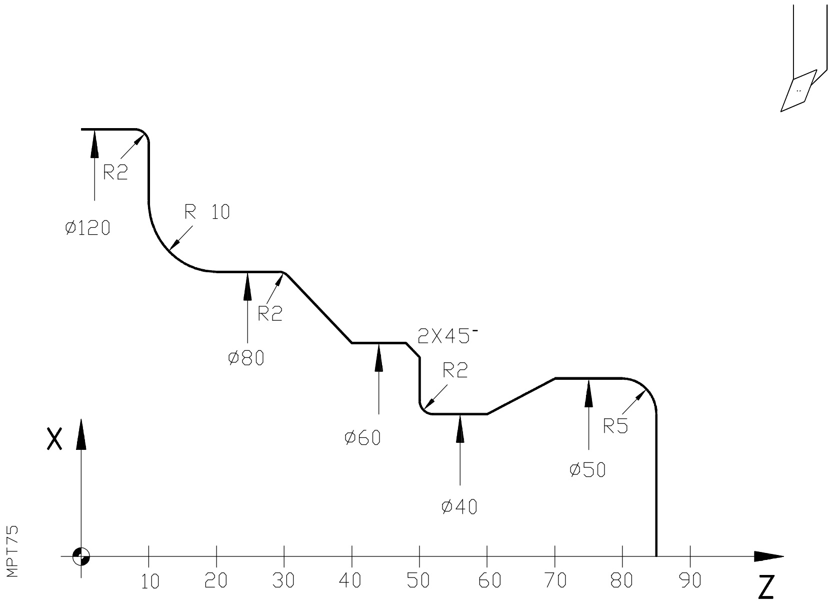
P14: Number of the last block to define the pattern.

The following points should be borne in mind when programming this canned cycle:

1. The definition of the pattern must not include point A because it is identified by P0 and P1.
2. The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle.
3. The parameters must be entered either in the cycle calling block or in previous blocks.
4. The exit conditions of the cycle are G00 and G90.
5. The pattern can be made up by straight lines, circles, roundings, tangential approaches, tangential exits and chamfers.
6. Absolute or incremental programming can be used.
7. In the definition of the pattern no T function can exist.
8. The approaching and withdrawing movements are carried out in rapid and the rest at the programmed feedrate.
9. The cycle is completed on the starting position of the tool.
10. Tool radius compensation (G41,G42) can be used.
11. The coordinates values (X,Z) of the point from where the cycle is called must be different from P0 and P1 respectively. Otherwise error code 4 will be generated.
12. The machining movements will be made at the programmed feedrate.

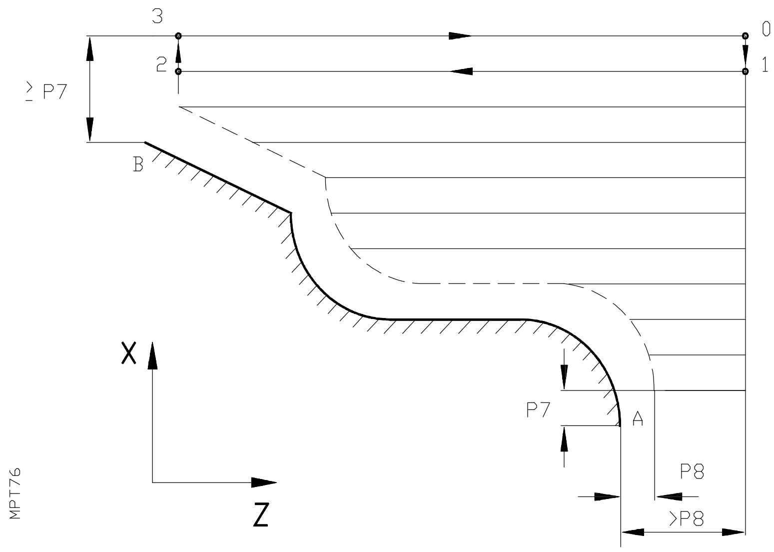
Note: No polar programming allowed. G02/G03 must be programmed with X Z I K.

Example G66: X in diameters.



N100 —
N110 G90 G00 G42 X150 Z115
N120 G66 P0=K0 P1=K85 P4=K20 P5=K5 P7=K1 P8=K1
P9=K100 P12=K40 P13=K200 P14=K290
N130 G40 X160 Z135
N140 M30
N200 G36 R5 X50 Z85
N210 X50 Z70
N220 X40 Z60
N230 G36 R2 X40 Z50
N240 G39 R2 X60 Z50
N250 X60 Z40
N260 G36 R2 X80 Z30
N270 G36 R10 X80 Z10
N280 G36 R2 X120 Z10
N290 X120 Z0

14.2. G68. STOCK REMOVAL ALONG THE X AXIS (TURNING)



Format:

N4 G68 P0=K P1=K P5=K P7=K P8=K P9=K P13=K P14=K

Meaning of the parameters:

P0: Absolute X coordinate value of the starting point (A) in radius or diameters.

P1: Absolute Z coordinate value of the starting point A.

P5: Max. depth of cut per pass (radius). It must be greater than zero or error code 3 will be displayed. The real step calculated by the CNC will be equal or smaller than the max. step.

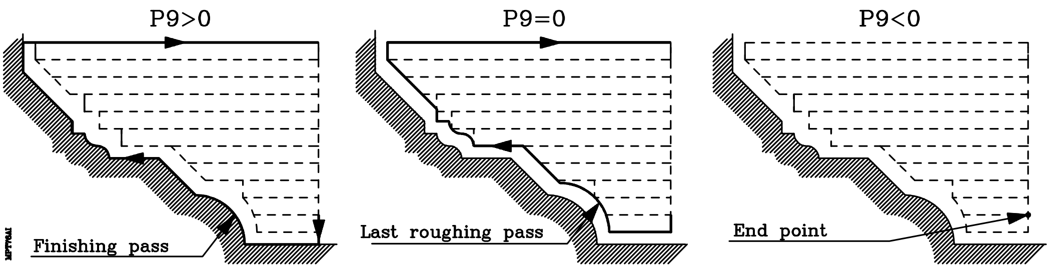
P7: Finishing stock allowance along X axis (radius). It must be greater or equal to zero, or error code 3 will be displayed.

P8: Finishing stock allowance along Z axis. It must be greater or equal to zero or error code 3 will be displayed.

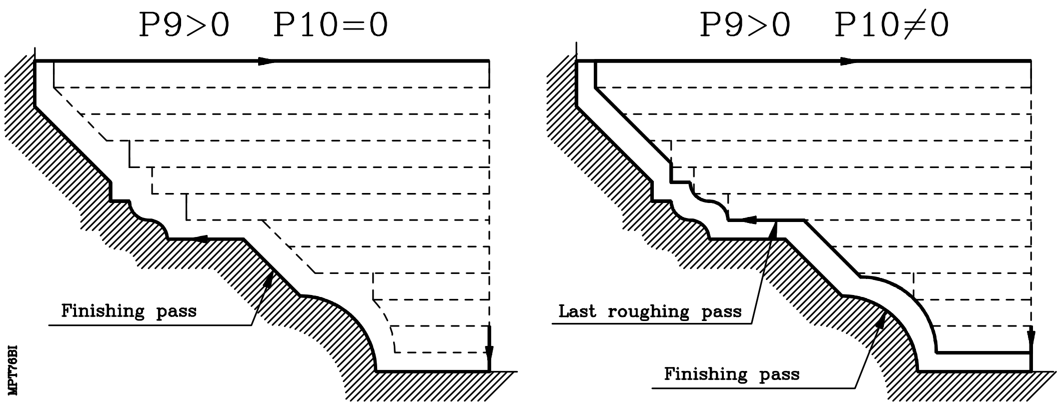
P9: Feedrate of the finishing pass.

If P9=0, there will be no finishing pass; but there will be a final roughing pass maintaining the excess material indicated by P7 and P8.

If it has a negative value, neither a final roughing pass nor a finishing pass will be carried out.



P10: This parameter must be assigned a value other than "0" in order for the CNC to carry out a final roughing pass prior to the finishing pass.



P13: Number of the first block to define the pattern.

P14: Number of the last block to define the pattern. It must be greater than P13. Otherwise error code 13 is generated.

When programming this canned cycle the following should be borne in mind:

1. The distance between the starting point 0 and final point (B) along X generating error 31 when operating with tool compensation the value of this distance (from 0 to B) should be equal to P7+NP5, N being an entire number (any multiple of P5).

2. The distance from 0 to A along Z axis should be higher than P8.

3. The definition of the pattern must not include point A because it is identified by P0 and P1.

4. The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle. The parameters can be programmed in the calling block or in previous blocks. The exit conditions are G00 and G90.

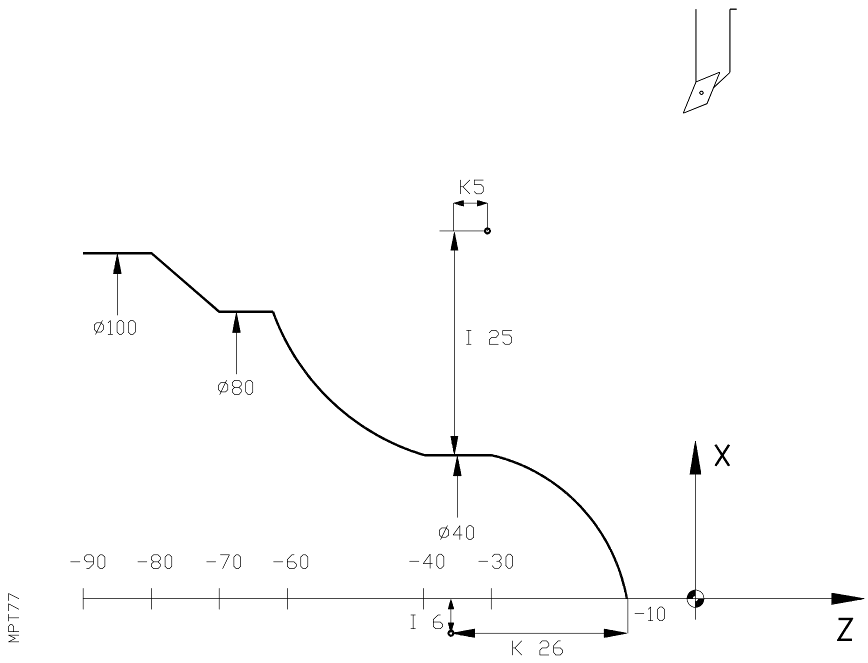
5. The pattern can be made up of straight lines and arcs. All the blocks of pattern definition will be programmed with cartesian coordinates being mandatory to program the two axes in absolute, otherwise, the CRT will display error 21. If arcs are included in the definition, they must be programmed with the center's I,K coordinates, referred to the arc's starting point and with the relevant sign. If functions F,S,T or M are programmed in the definition, they will be ignored except for the finishing pass. No polar definitions can be used.

6. The cycle is completed on the starting position of the tool 0.

7. If the last movement prior to calling the canned cycle (G68) has been executed in G00, tool radius compensation (G41,G42) can be used. Otherwise error 35 will be displayed.

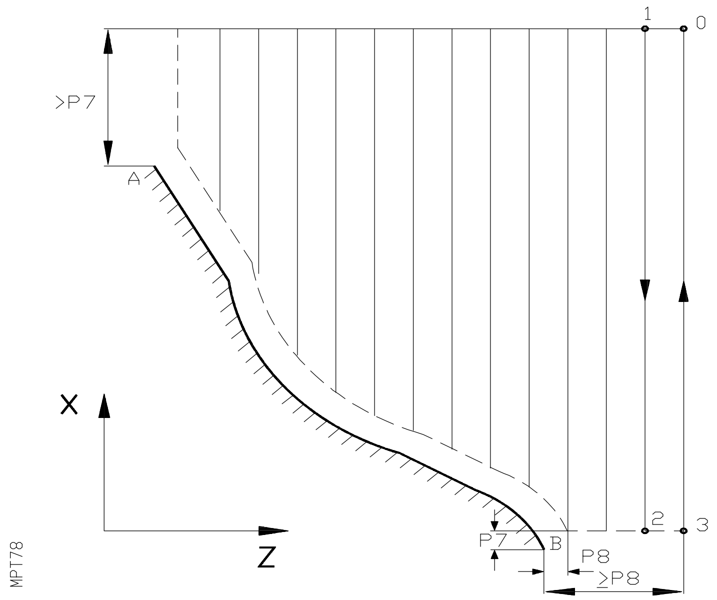
The figure shows the elemental cycle. The movements 1-2 and 2-3 will be performed at the programmed feedrate and the 0-1 and 3-0 in rapid.

Example G68.



N100 —
N110 G42 G00 X120 Z0
N120 G68 P0=K0 P1=K-10 P5=K2 P7=K0.8 P8=K0.8 P9=K100 P13=K200 P14=K250
N130 G40 X130 Z10
N140 M30
N200 G03 X40 Z-30 I-6 K-26
N210 G01 X40 Z-40
N220 G02 X80 Z-60 I25 K5
N230 G01 X80 Z-70
N240 X100 Z-80
N250 X100 Z-90

14.3. G69. STOCK REMOVAL ALONG THE Z AXIS (FACING)



Format:

N4 G69 P0=K P1=K P5=K P7=K P8=K P9=K P13=K P14=K

Meaning of the parameters:

P0: Coordinate X value of the starting point (A) in radius or diameters.

P1: Coordinate Z value of the starting point (A).

P5: Max. step. It must be greater than zero or error code 3 will be displayed. The real step calculated by the CNC will be smaller or equal to the max. step.

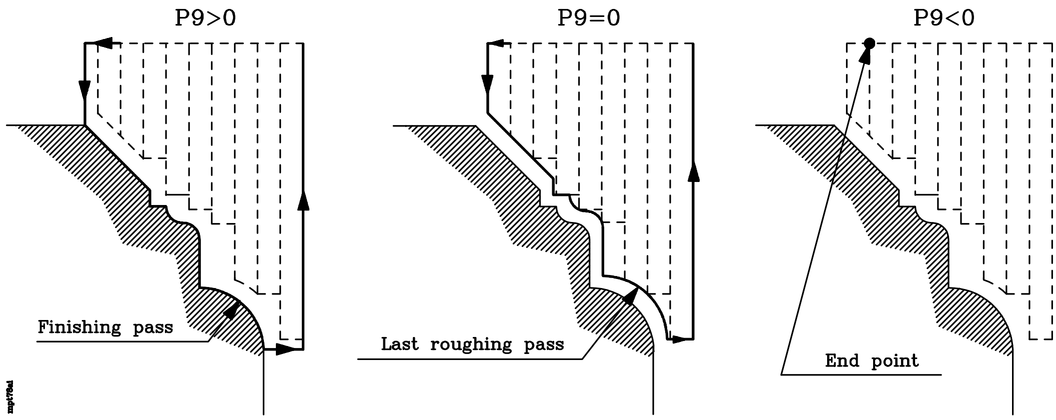
P7: Finishing stock allowance along X axis. It must be greater or equal to zero or error code 3 will be displayed.

P8: Finishing stock allonwance along Z axis. It must be greater or equal to zero or error code 3 will be displayed.

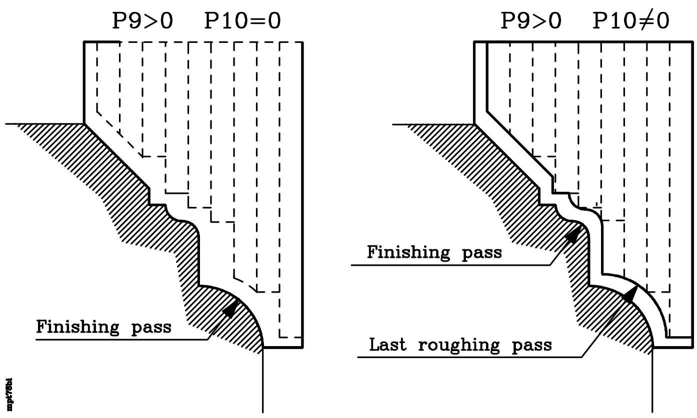
P9: Feedrate of the finishing pass.

If P9=0, there will be no finishing pass; but there will be a final roughing pass maintaining the excess material indicated by P7 and P8.

If it has a negative value, neither a final roughing pass nor a finishing pass will be carried out.



P10: This parameter must be assigned a value other than "0" in order for the CNC to carry out a final roughing pass prior to the finishing pass.



P13: Number of the first block to define the pattern.

P14: Number of the last block to define the pattern. It must be higher than P13 or error code 13 will be displayed.

When programming this canned cycle, the following should be borne in mind:

1. The distance between the starting point 0 and B point along the Z axis must be equal or greater than P8. To avoid passes that are too thin or generating error P31 when operating with tool compensation the value of this distance (from 0 to B) should be equal to P8+NP5, N being an entire number (any multiple of P5).

2. The distance from 0 to A along. The axis should be higher than P7.

3. The definition of the pattern must not include point A because it is identified by P0 and P1.

4. The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle. The parameters can be programmed in the calling block or in previous blocks. The exit conditions are G00 and G90.

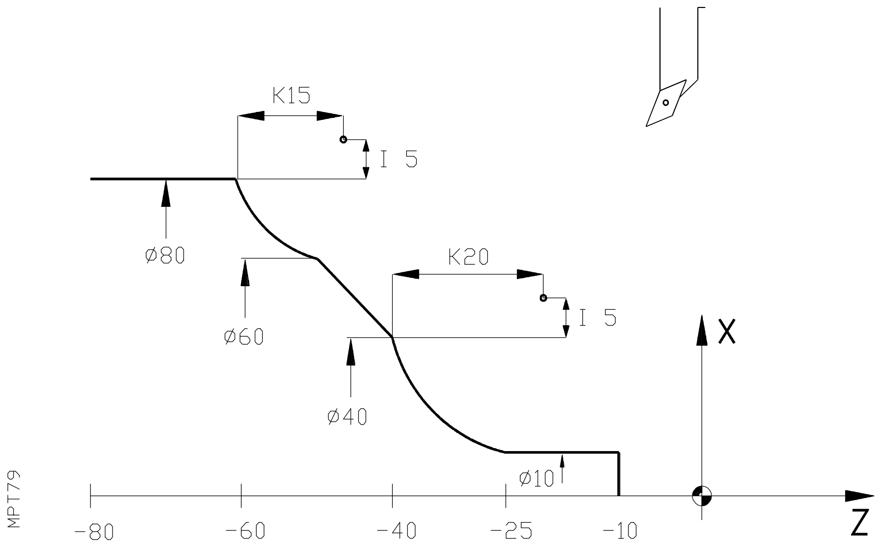
5. The pattern can be made up of straight lines and arcs. All the blocks of pattern definition will be programmed with cartesian coordinates being mandatory to program the two axes in absolute, otherwise, the CRT will display error 21. If arcs are included in the definition, they must be programmed with the center's I,K coordinates, referred to the arc's starting point and with the relevant sign. If functions F,S,T or M are programmed in the definition, they will be ignored except for the finishing pass. No polar definitions can be programmed.

6. The cycle is completed on the starting position of the tool (0).

7. If the last movement prior to calling the canned cycle (G69) has been executed in G00, tool radius compensation (G41,G42) can be used. Otherwise error 35 will be displayed.

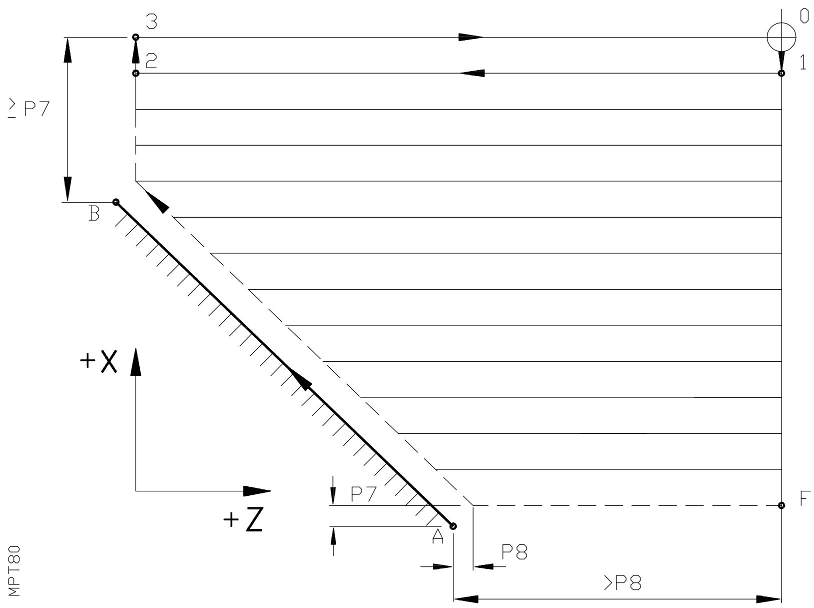
The figure shows the elemental cycle. The movements 1->2 and 2->3 will be performed at the programmed feedrate and the 0->1 and 3->0 in rapid.

Example G69.



N190 —
N200 G41 G0 X90 Z-5
N210 G69 P0=K80 P1=K-80 P5=K2 P7=K0.8 P8=K0.8 P9=K100 P13=K300 P14=K340
N220 G40 X100 Z0
N230 M30
N300 G01 X80 Z-60
N310 G03 X60 Z-50 I5 K15
N320 G01 X40 Z-40
N330 G03 X10 Z-25 I5 K20
N340 G01 X10 Z-10

14.4. G81. TURNING CANNED CYCLE WITH STRAIGHT SECTIONS



EXAMPLE: Let us suppose the coordinate values of the drawing's points are: A(X0 Z0) B(X90 Z-45) 0(X134 Z47) and the programming of X axis is in diameters.

N90 G00 X134 Z47 .... (The tool located in point 0).

N100 G81 P0=K0 P1=K0 P2=K90 P3=K-45 P5=K5 P7=K3 P8=K4 P9=K100

Meaning of the parameters:

P0: X coordinate value of the point A (radius or diameters).

P1: Z coordinate value of the point A.

P2: X coordinate value of the point B (radius or diameters).

P3: Z coordinate value of point B.

P5: Max. step. It must be greater than zero or error code 3 will be displayed. The real value calculated by the CNC will be smaller or equal to the max. step.

P7: Finishing stock allowance on the X axis. It must be greater or equal to zero or error code 3 will be displayed.

P8: Finishing stock allowance along the Z axis. It must be greater or equal to zero or error code 3 will be displayed.

P9: Feedrate of the finishing pass. If it is zero, there will be no finishing pass. If it is negative, error code 3 will be displayed.

When programming this canned cycle, the following should be borne in mind:

1. The distance between the starting point 0 and final point (B) along the X axis must be equal or greater than P7. To avoid passes that are too thin or generating error 31 when operting with tool compensation the value of this distance (from 0 to B) should be equal to P7+NP5, N being an entire number.

2. The distance from 0 to A along Z axis should be higher than P8.

3. The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle. The parameters can be programmed in the calling block or in previous blocks. The exit conditions of the cycle are G00 and G90.

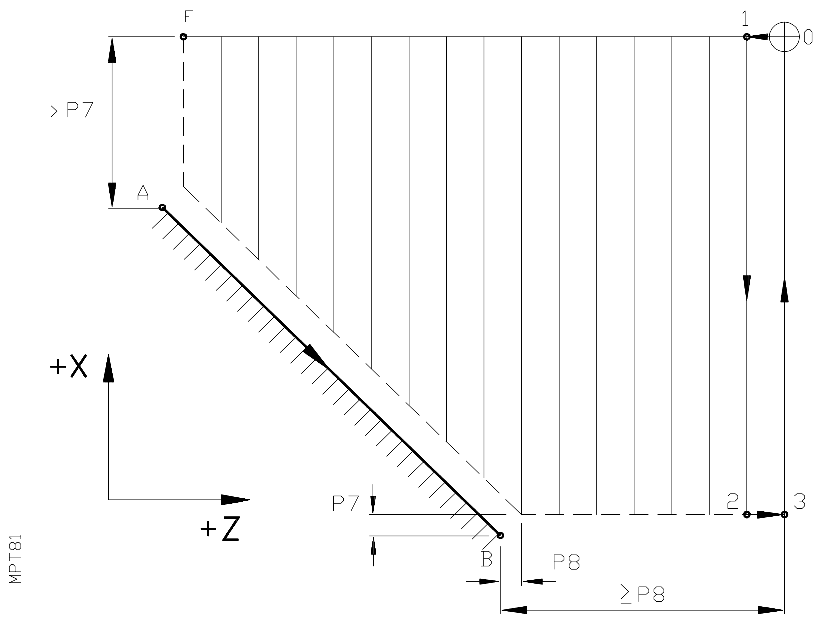
4. If the position of the tool is not correct to execute the cycle, error code 4 will be displayed. If it is correct a horizontal turning will be carried out until it is reached and then the cycle will be executed.

5. If there is a finishing pass the cycle will be completed on the starting position of the tool (0). Otherwise the cycle will end on point F.

6. If the last movement prior to calling the canned cycle (G81) has been executed in G00, tool radius compensation (G41,G42) can be used. Otherwise error 35 will be displayed.

The figure shows the elemental cycle. The movements 1- 2 and 2-3 will be performed at the programmed feedrate and the 0-1 and 3-0 in rapid.

14.5. G82. FACING CANNED CYCLE WITH STRAIGHT SECTIONS



EXAMPLE: Let us suppose the coordinate values of the drawing's points are: A(X90 Z-45) B(X0 Z0) 0(X136 Z39) and the programming of X axis is in diameters.

N90 G00 X136 Z39 .... (The tool located in point 0).

N100 G82 P0=K90 P1=K-45 P2=K0 P3=K0 P5=K5 P7=K3 P8=K4 P9=K100

Meaning of the parameters:

P0: X coordinate value of point A (radius or diameters).

P1: Z coordinate value of point A.

P2: X coordinate value of point B (radius or diameters).

P3: Z coordinate value of point B.

P5: Max. step. It must be greater than zero or error code 3 will be one displayed. The real step calculated by the CNC will be smaller or equal to the max. step.

P7: Finishing stock allowance on the X axis. It must be greater or equal to zero or error code 3 will be displayed.

P8: Finishing stock allowance along the Z axis. It must be greater or equal to zero or error code 3 will be displayed.

P9: Feedrate of the finishing pass. If it is zero there will be no finishing pass. If it is negative, error code 3 will be displayed.

NOTES:

1. The distance between the starting point 0 and final point (B) along the Z axis must be equal or greater than P8. To avoid passes that are too thin or generating error 31 when operting with tool compensation the value of this distance (from 0 to B) should be equal to P8+NP5, N being an entire number.

2. The distance from 0 to A along the X axis should be higher than P7.

3. The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle. The parameters can be programmed in the calling block or in previous blocks. The exit conditions of the cycle are G00 and G90.

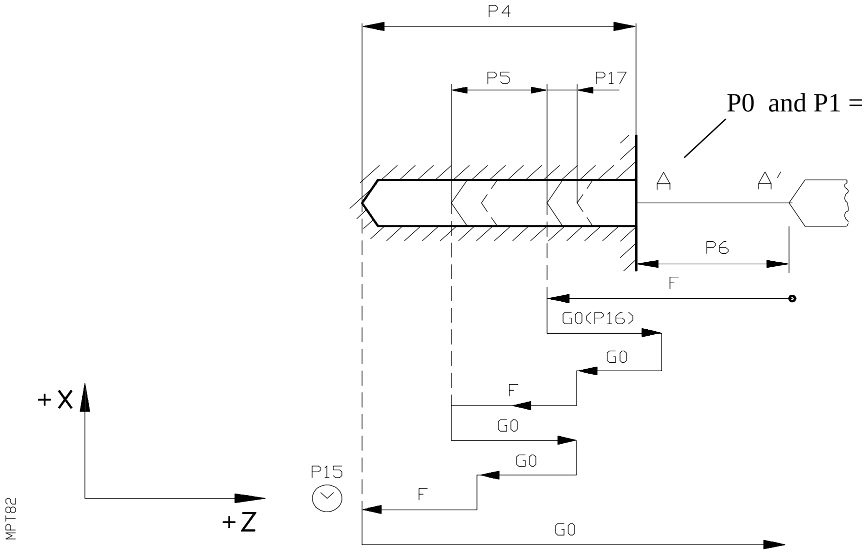
4. If the position of the tool is not correct to execute the cycle, error code 4 will be displayed. If it is correct a vertical facing will be carried out until it is reached and then the cycle will be executed.

5. If there is a finishing pass, the cycle will be completed on the tool's starting position (0). If there is no finishing pass the cycle will end at point F.

6. If the last movement prior to calling the canned cycle (G82) has been executed in G00, tool radius compensation (G41,G42) can be used. Otherwise error 35 will be displayed.

The figure shows the elemental cycle. The movements 1-2 and 2-3 will be performed at the programmed feedrate and the 0-1 and 3-0 in rapid.

14.6. G83. DEEP HOLE DRILLING CYCLE



Format:

N4 G83 P0=K P1=K P4=K P5=K P6=K P15=K P16=K P17=K

Meaning of the parameters:

P0: Absolute X value of the point where the drilling or circular groove is desired (if different from zero) in radius or diameters.

P1: Absolute Z value of the point where the drilling is desired.

P4: Total depth of the hole. It will have positive value when drilled towards the negative direction of the Z axis and viceversa. If it is zero, error code 3 will be displayed.

P5: Max. pass. The CNC will execute the minimum number of equal passes, smaller than P5 until the total depth, defined by P4, is reached. If it is equal to or smaller than zero, error code 3 will be displayed.

P6: Safety distance. It defines the distance to the part from the point where the tool ends the positioning approach. If it is negative, error code 3 will be generated.

P15: Dwell. It identifies the value in seconds of the dwell at the bottom of the hole. If it is negative, error code 3 will be displayed.

P16: It indicates the incremental value of the G00 movement after each pass. If it is zero, this movement will be executed up to the A'point. If it is negative, error 3 will be displayed.

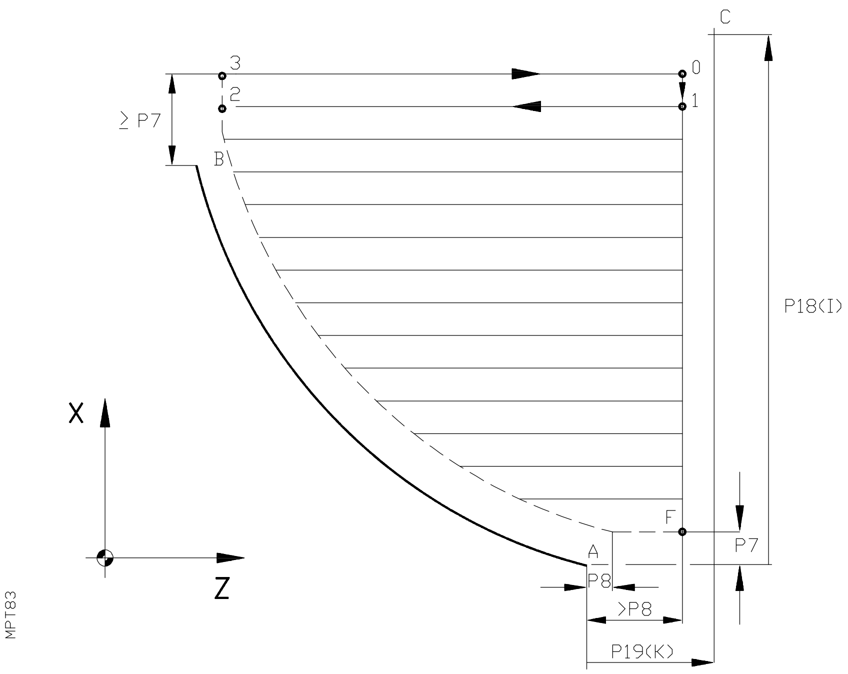
P17: It indicates the safety distance between the bottom of the previous penetration and the point where the tool ends the rapid approach for a subsequent penetration. If it is negative, error code 3 will be displayed.

The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle. The parameters can be programmed in the calling block or in previous blocks.

The cycle does not alter the calling parameters and thus, they can be used for future cycles.The parameters P90 to P96 are altered. The exit conditions are G00,G07,G40 and G90.

The cycle starts with a G00 approach to point A' and ends at A' as well.

14.7. G84. TURNING CANNED CYCLES WITH ARCS



EXAMPLE: Let us suppose the coordinate values of the drawing's point are: 0(X149 Z86) A(X0 Z71) B(X120 Z11) C(X160 Z91) and the programming of X axis is in diameters.

N90 G00 X149 Z86 .... (The position of the tool is the point 0)
N100 G84 P0=K0 P1=K71 P2=K120 P3=K11 P5=K5 P7=K4 P8=K4 P9=K100
P18=K80 P19=K20

Meaning of the parameters:

P0: X coordinate value of point A (radius or diameters).

P1: Z coordinate value of point A.

P2: X coordinate value of point B (radius or diameters).

P3: Z coordinate value of point B.

P5: Max. step. It must be greater than zero or error code 3 will be displayed. The real step calculated by the CNC will be smaller or equal to the max. step.

P7: Finishing stock allowance on the X axis. It must be greater or equal to zero or error code 3 will be displayed.

P8: Finishing stock allowance along the Z axis. It must be greater or equal to zero or error code 3 will be displayed.

P9: Feedrate of the finishing pass. If it is zero there will be no finishing pass. If it is negative, error code 3 will be displayed.

P18: Distance **I** between the point **A** and the arc's center along the X axis. Although the X axis is programmed in diameters, the values of I are always programmed in radius.

P19: **K** distance between the point **A** and the arc's center along the Z axis.

When programming this canned cycle, take into account the following aspects:

1. The distance between the starting point 0 and final point (B) along the Z axis must be equal or greater than P8. To avoid passes that are too thin or generating error 31 when operating with tool compensation, the value of this distance (from 0 to B) should be equal to P8+N P5, N being an entire number.

2. The distance between the starting point 0 and the point (A), along the X axis, should be higher than P7.

3. The machining conditions (feedrate, spindle rotation ...) must be programmed before calling the cycle. The parameters can be programmed either in the cycle calling block or in previous blocks. The exit conditions are G00 and G90.

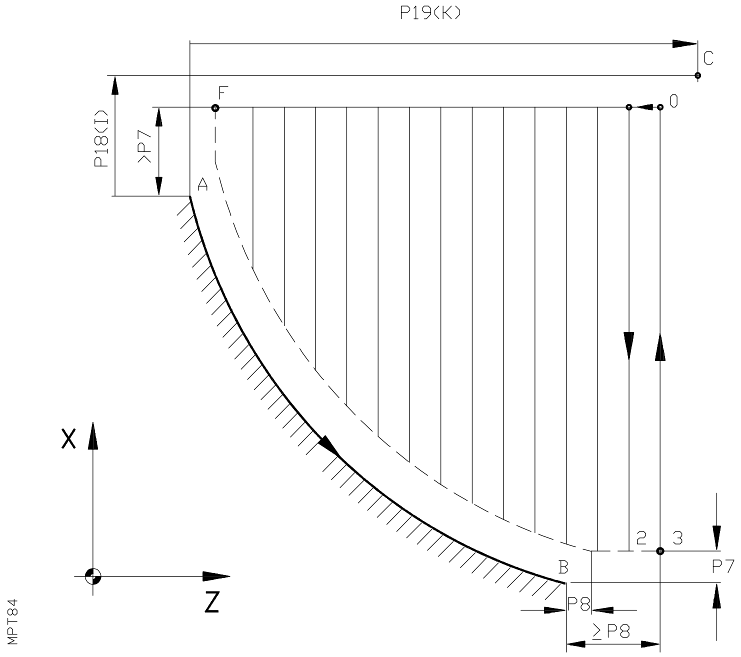
4. If the position of the tool is not correct to execute the cycle, error code 4 will be displayed. If it is correct a horizontal turning will be carried out until it is reached and then the cycle will be executed.

5. If there is a finishing pass, the cycle will be completed on the tool's starting position (0). If there is no finishing pass the cycle will end at point F.

6. If the last movement prior to calling the canned cycle has been executed in G00, tool radius compensation (G41,G42) can be used. Otherwise error 35 will be displayed.

The figure shows the elemental cycle. The movement 1-2 and 2-3 will be performed at the programmed feedrate and the 0-1 and 3-0 in rapid.

14.8. G85. FACING CYCLE WITH ARCS



EXAMPLE: Let us suppose the coordinate values of the drawing's point are: 0(X150 Z85) A(X118 Z11) B(X0 Z70) C(X160 Z91) and the programming of X axis is in diameters.

N90 G00 X150 Z85 .... (The position of the tool is the point 0)

N100 G85 P0=K118 P1=K11 P2=K0 P3=K70 P5=K5 P7=K4 P8=K4 P9=K100
P18=K21 P19=K80

Meaning of the parameters:

P0: X coordinate value of point A (radius or diameters).

P1: Z coordinate value of point A.

P2: X coordinate value of point B (radius or diameters).

P3: Z coordinate value of point B.

P5: Max. step. It must be greater than zero or error code 3 will be displayed. The real step calculated by the CNC will be smaller or equal to the max. step.

P7: Finishing stock allowance on the X axis. It must be greater or equal to zero or error code 3 will be displayed.

P8: Finishing stock allowance along the Z axis. It must be greater or equal to zero or error code 3 will be displayed.

P9: Feedrate of the finishing pass. If it is zero there will be no finishing pass. If it is negative, error code 3 will be displayed.

P18: Distance **I** between the point **A** and the arc's center along the X axis. Although the X axis is programmed in diameters, the values of **I** are always programmed in radius.

P19: **K** distance between the point **A** and the arc's center along the Z axis.

When programming this canned cycle, take into account the following aspects:

1. The distance between the starting point 0 and final point (B) along the Z axis must be equal or greater than P8. To avoid passes that are too thin or generating error 31 when operating with tool compensation, the value of this distance (from 0 to B) should be equal to P8+N P5, N being an entire number.

2. The distance between the starting point 0 and the point (A), along the X axis, should be higher than P7.

3. The machining conditions (feedrate, spindle rotation ...) must be programmed before calling the cycle. The parameters can be programmed either in the cycle calling block or in previous blocks. The exit conditions are G00 and G90.

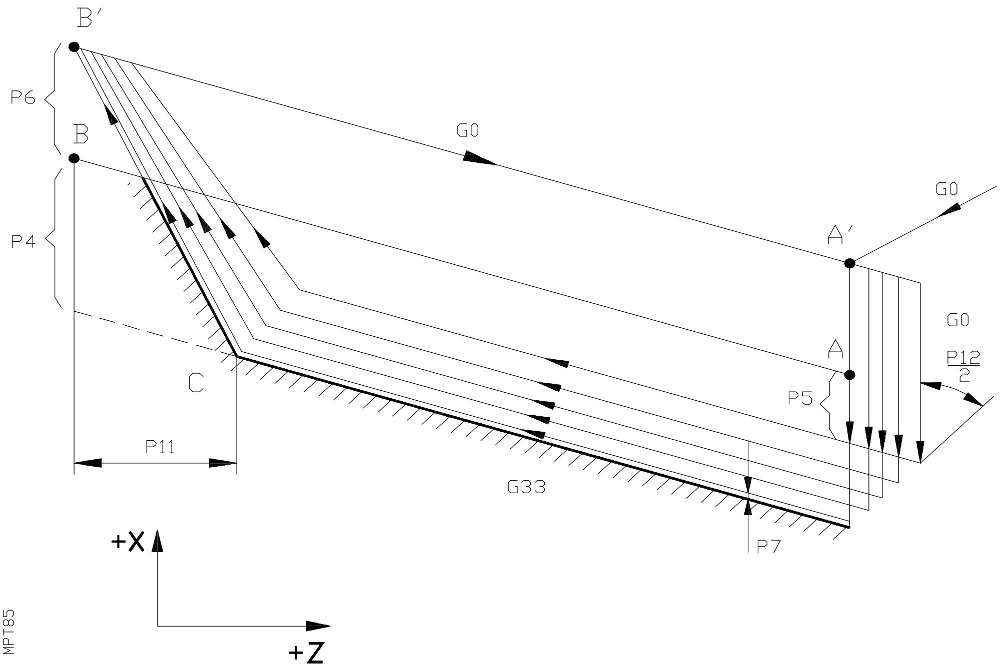
4. If the position of the tool is not correct to execute the cycle, error code 4 will be displayed. If it is correct a vertical facing will be carried out until it is reached and then the cycle will be executed.

5. If there is a finishing pass, the cycle will be completed on the tool's starting position (0). If there is no finishing pass the cycle will end a point F.

6. If the last movement prior to calling the canned cycle has been executed in G00, tool radius compensation (G41,G42) can be used. Otherwise error 35 will be displayed.

The figure shows the elemental cycle. The movement 1-\>2 and 2-\>3 will be performed at the programmed feedrate and the 0-\>1 and 3-\>0 in rapid.

14.9. G86. THREADCUTTING CYCLE (Z axis)



Format:

N4 G86 P0=K P1=K P2=K P3=K P4=K P5=K P6=K P7=K P10=K P11=K P12=K

Meaning of the parameters:

P0: Absolute X value of the starting point of the thread (A) in radius or diameters.

P1: Absolute Z value of the starting point of the thread (A).

P2: Absolute X value of the final point of the thread (B) in radius or diameters.

P3: Absolute Z value of the final point of the thread.

P4: Depth of the thread (in radius). If will have a positive value in external threads and a negative one in internal threads. If it is zero error code 3 will be displayed.

P5: Initial pass (in radius). It defines the depth of the first cutting pass. The subsequent passes will depend on the sign given to the parameter.

- If the sign is **positive**, the depth of the second pass will be P5 $\sqrt{2}$ and the depth of the 11th will be P5 $\sqrt{n}$, until the finishing depth is reached.
- If the sign is **negative**, the penetration increment will be constant and of a value equal to the absolute value of the parameter.
- If the value is 0, error 3 will be issued.

P6: Safety distance in radius. It indicates the distance from B to B'.

- If the value is positive, this movement will be done in G05 (rounded corner). The 0 value is considered positive.
- If the value is negative, this movement will be done in G07 (square corner).

P7: Finishing pass (in radius), this pass is carried out with radial approach.

- If it is 0, the previous pass is repeated.
- If the value is positive, the finishing pass will be carried out maintaining a P12/2 angle with the X axis.
- If the value is negative, the finishing pass will be done with radial entry.

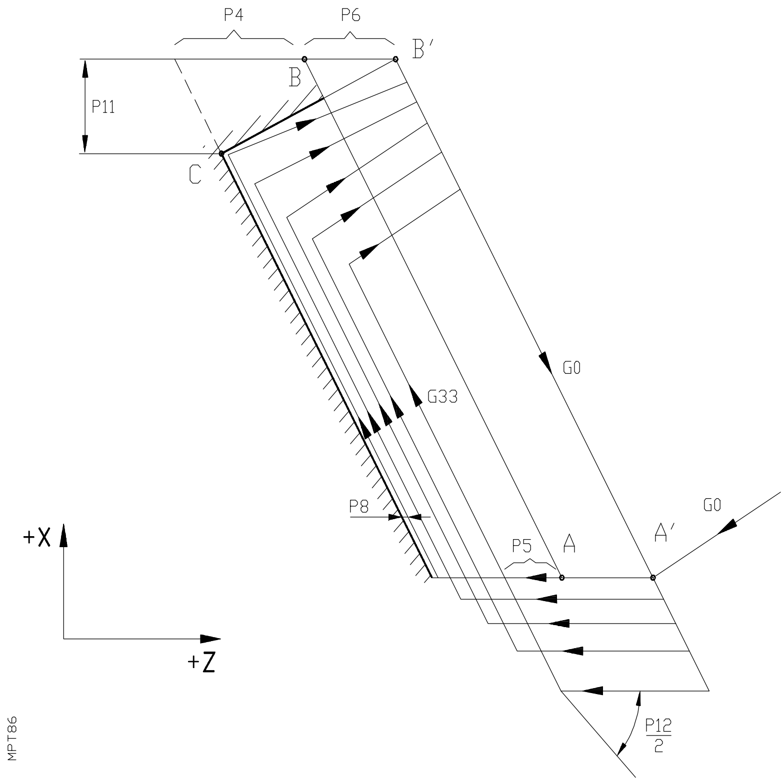
P10: Thread pitch along Z axis in inches or mm per thread.
Note: For threads per inch (or mm) use 1/threads per inch (or mm). Ex.:6 threads per inch = P10 =K1 F4 K6.

P11: Thread exit. It defines the distance from the end of the thread to the point where the exit starts. If it is negative, error code 3 will be displayed. If it is different from zero, the section CB' is a tapered thread whose pitch along Z axis is P10. If it is zero, the section CB' is executed in G00 (X only while Z decelerates).

P12: Angle of the tool's nose. It makes the starting points of the successive passes to be at a P12/2 angle with X axis. Do not forget to adjust the tool angle by 1/2° if you want each pass to shave the thread wall.

The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle. The parameters can be programmed in the calling block or in previous blocks. The parameters P80 to P99 are altered. The exit conditions are G00,G07,G40, G90 and G97. The cycle starts with a G00 approach to point A' and ends at A' as well. When executing the block, the F feedrate speed cannot be altered by turning the **FEEDRATE** knob whose value will be frozen at 100%.

14.10. G87. THREADCUTTING CYCLE (X axis)



Format:

N4 G87 P0=K P1=K P2=K P3=K P4=K P5=K P6=K P8=K P10=K P11=K P12=K

Meaning of the parameters:

P0: Absolute X coordinate value of the initial point of the thread (A) in radius or diameters.

P1: Absolute Z coordinate value of the initial point of the thread (A).

P2: Absolute X coordinate value of the final point of the thread (B) in radius or diameters.

P3: Absolute Z coordinate value of the final point of the thread (B).

P4: Depth of the thread. It will have a position value when the cutting takes place towards the negative direction of the Z axis and viceversa. If it is zero, error code 3 will be displayed.

P5: Initial pass. It defines the depth of the first cutting pass. The subsequent passes will depend on the sign given to the parameter:

- If the sign is **positive,** the depth of the second pass will be P5 $\sqrt{2}$ and the depth of the 11th will be P5 $\sqrt{n}$, until the finishing depth is reached.

- If the sign is **negative**, the deepening increment will be constant and of a value equal to the absolute value of the parameter.

- If the value is equal to zero, error 3 is generated.

P6: Safety distance. It indicates the distance from point B to point B'.

- If the value is positive, this movement will be done in G05 (rounded corner). The 0 value is considered positive.

- If the value is negative, this movement will be done in G07 (square corner).

P8: Finishing pass:

- If it is 0, the previous pass is repeated.

- If the value is positive, the finishing pass will be carried out maintaining a P12/2 angle with the Z axis.

- If the value is negative, the finishing pass will be done with radial entry.

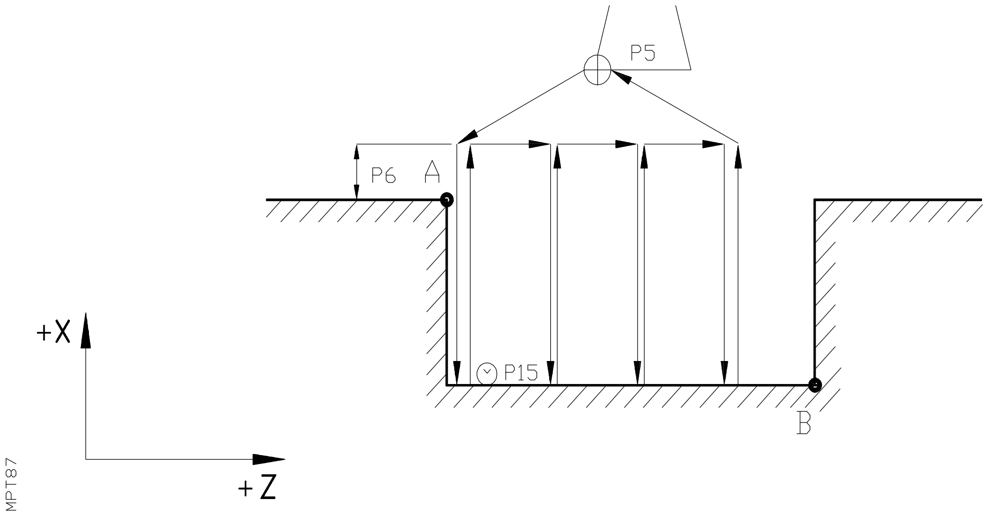
P10: Thread pitch along X axis in radius.
Note: For threads per inch (or mm) use 1/threads per inch (or mm). Ex.:6 threads per inch = P10 =K1 F4 K6.

P11: Thread exit (in radius). It defines the distance from the end of the thread to the point where the exit starts. If it is negative, error code 3 will be displayed. If it is different from zero, the section CB’ is a tapered thread whose pitch along X axis is P10. If it is zero, the section CB’ is executed in G00 (X only while Z decelerates).

P12: Angle of the tool’s nose. It makes the starting points of the successive passes to be at a P12/2 angle with the Z axis. Do not forget to adjust the tool angle by 1/2° if you want each pass to shave the thread wall.

The machining conditions (feedrate, spindle rotation, etc.) must be programmed before the cycle is called. The parameters can be programmed in the call block or in previous blocks. The cycle does not alter the calling parameters and thus, they can be used for future cycles. The parameters P80 to P99 are altered. The exit conditions are G00, G07, G40, G90 and G97. The cycle starts with a G00 approach to point A’ and ends at A’ as well. When executing the block, the F feedrate speed can’t be altered by turning the FEEDRATE knob whose value will be frozen at 100%.

14.11. G88. GROOVING CYCLE ALONG X AXIS



Format:

N4 G88 P0=K P1=K P2=K P3=K P5=K P6=K P15=K

Meaning of the parameters:

P0: X coordinate value of point A (radius or diameters).

P1: Z coordinate value of point A.

P2: X coordinate value of point B (radius or diameters).

P3: Z coordinate value of point B.

P5: Tool nose's width. It must be greater than zero or error code 3 will be displayed. The real step calculated by the CNC will be smaller than the tool's width.

P6: Safety distance. It must be equal or greater than zero or error code 3 will be displayed.

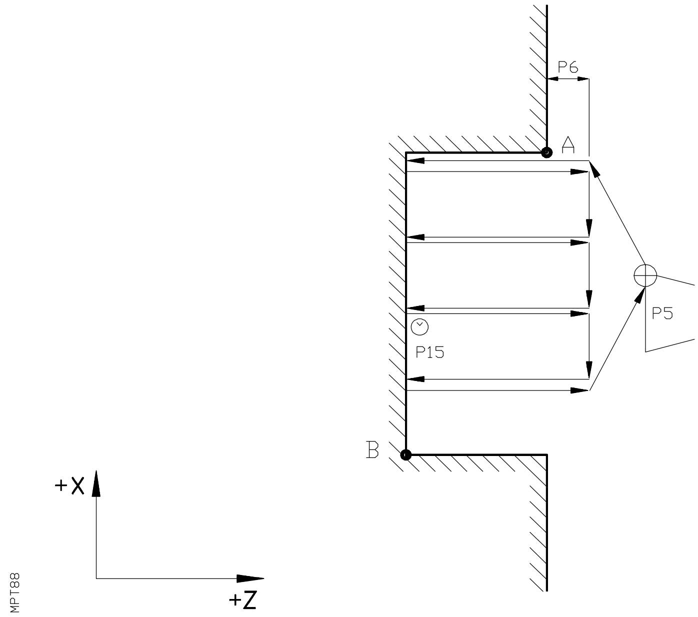
P15: Dwell at the bottom (seconds). It must be equal or greater than 0 and smaller than 655.36 seconds. Otherwise, error code 3 will be displayed.

The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle. The parameters can be programmed in the calling block or in previous blocks. The exit conditions of the cycle and G00,G07,G40 and G90. If the groove's depth is zero, code 3 will be displayed. If the width of the groove is smaller than the tool nose's width, error code 3 will be displayed. If the position of the tool is not correct to execute the cycle, error code 4 will be displayed.

The movement from the safety distance to the bottom of the groove is carried out at the programmed feedrate. The rest of the movements in rapid.

The cycle ends at the tool's initial position.

14.12. G89. GROOVING CYCLE ALONG Z AXIS



Format:

N4 G89 P0=K P1=K P2=K P3=K P5=K P6=K P15=K

Meaning of the parameters:

P0: X coordinate value of point A (radius or diameters).

P1: Z coordinate value of point A.

P2: X coordinate value of point B (radius or diameters).

P3: Z coordinate value of point B.

P5: Tool nose's width. It must be greater than zero, or error code 3 will be displayed. The real step calculated by the CNC will be smaller than the tool's width.

P6: Safety distance. It must be equal or greater than zero or error code 3 will be displayed.

P15: Dwell at the bottom (seconds). It must be equal or greater than zero and smaller than 655.36 seconds. Otherwise error code 3 will be displayed.

The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle. The parameters can be programmed in the call block or in previous blocks. The exit conditions of the cycle and G00,G07,G40 and G90. If the depth of the groove is zero, error code 3 will be displayed. If the width of the groove is smaller than the tool nose's width, error code 3 will be displayed. If the position of the tool is not correct to execute the cycle, error code 4 will be displayed.

The movement from the safety distance to the bottom of the groove is carried out at the programmed feedrate. The rest of the movements in rapid.

The cycle ends at the tool's initial position.

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 * The range or the Constant Surface Speed has not 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 Canned cycle profile 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 G14, G15, G16, 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.

> Synchronization factor K of the synchronized tool too large.

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 More than one spindle range have been defined in the same block.

021 This error will be issued in the following cases:

> There is no block at the address defined by the parameter assigned to F18, F19, F20, F21, F22.
> The corresponding axis has not been defined in the addressed block

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

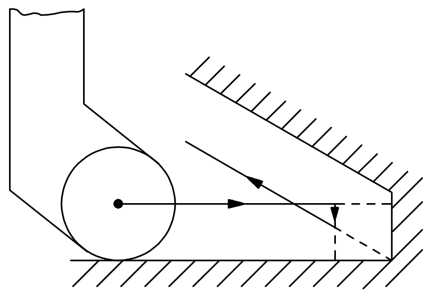
028 An attempt has been made to select a tool offset at the tool table or a non-existent external tool (the number of tools is set by machine parameter).

029 Too large a value assigned to a function.

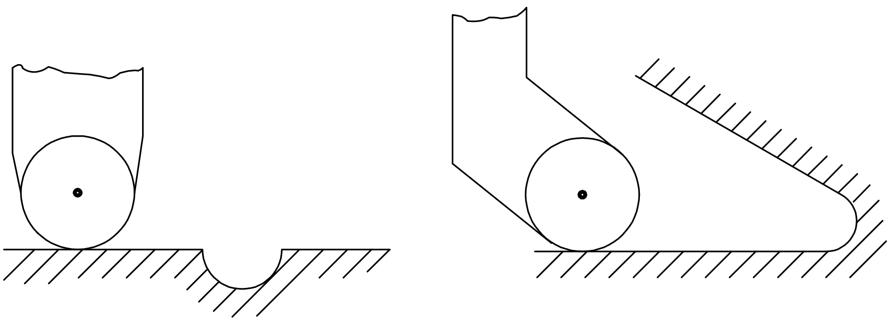
This error is often issued when programming an F value in mm/min (inch/min) and, then, switching to work in mm/rev (inch/rev) without changing the F value.

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 Z axis position Z-5000, if we want to move it to point Z5000, the CNC will issue error 33 when programming the block N10 Z5000 since the programmed move will be: Z5000 - Z-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 Z0          ; 5000 mm move
N10 Z5000       ; 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 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:

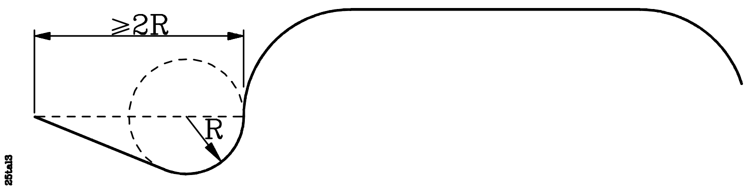
> More than 15 nesting levels when calling subroutines.

> 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:

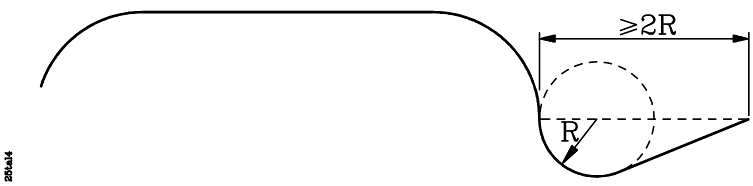
> There is no room to perform the tangential entry. A clearance of twice the rounding radius or greater is required.



> 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 Function M45 S programmed wrong (speed of the live tool).

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 Start or cancel tool radius compensation while in G02 or G03.

049 Chamfer programmed incorrectly.

050 G96 has been programmed while the S output is in BCD as set by machine parameter. (AC spindle).

051 * "C" axis programmed incorrectly

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

055 Parity error when reading or recording a cassette or a floppy disk.

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 tape.

058 Problems in floppy disk movement or sluggish tape movement.

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

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.

070 ** X axis following error.

071 ** Synchronized tool following error

072 ** Z axis following error.

073 ** 4th axis following error.

074 ** This error is issued in the following cases:

> 3rd axis following error

>"C" axis following error

075 ** Feedback error at connector A1.

076 ** Feedback error at connector A2.

077 ** Feedback error at connector A3.

078 ** Feedback error at connector A4.

079 ** Feedback error at connector A5.

081 ** 3rd axis travel limit overrun.

082 ** Parity error in 4th axis parameters. The CNC initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=1.

083 ** 4th axis travel limit overrun.

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 initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=1.

095 ** Parity error in general parameters. The CNC initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=1.

096 ** Parity error in Z axis parameters. The CNC initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=1.

097 ** Parity error in 3rd or "C" axis parameters. The CNC initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=1.

098 ** Parity error in X axis parameters. The CNC initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=1.

099 ** Parity error in M table. The CNC initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=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.

108 ** Error in Z axis leadscrew error compensation parameters. The CNC initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=1.

110 ** Error in X axis leadscrew error compensation parameters. The CNC initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=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 <u>half</u> the time indicated in machine parameter "P729".

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

118 * An attempt has been made to modify an <u>unavailable</u> 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.

<u>Atention:</u>

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.