

PROGRAMMING MANUAL
CNC E 560

C.B.Ferrari

TABLE OF CONTENTS

PROGRAMMING	405
DEFINITION OF PROGRAMMING FORMAT	406
G... FUNCTIONS	409
G0 RAPID POSITIONING	415
G1 LINEAR INTERPOLATION	416
G2 CIRCULAR INTERPOLATION CLOCKWISE	417
G3 CIRCULAR INTERPOLATION COUNTER CLOCKWISE	417
G4 STOP CYCLE	418
G5-G7-G9 RAPID FUNCTIONS	418
G17XY INTERPOLATION PLANE	419
G18XZ INTERPOLATION PLANE	419
G19YZ INTERPOLATION PLANE	419
G26-G27 FEEDRATE ON THE TOOL TIP	420
G30-G31 AUTOMATIC VELOCITY REGULATION.	421
G31D.. RADIUS COMPENSATION IN SPACE	423
G30-G32 NO AUTOMATIC VELOCITY REGULATION.	423
G39 SPINDLE QUILL KEY POSITIONING	423
G40-G41-G42 RADIUS COMPENSATION	424
G43 - G44 GRINDING CYCLE	425
G45-G45/ PICK UP TOOL CYCLE	427
G47 TOOL RELEASE CYCLE	429
G49 [.....] PROGRAM LINK	432
G50 G51 ROTOTRANSLATION	433
G55 HEAD ROTATION	434
G56 HEAD AND ORIGIN ROTATION	435
G58 ROTARY TABLE REVOLUTIONS RESET	437
G59 RETURN FROM A MACRO	437
G60-G61 SCALE FACTOR	437
G63 HIGH SPEED MILLING OF PROFILES AND SURFACES (OPTIONAL)	438
G70 G71 INCH-METRIC MODES	440
G80 DRILLING CYCLE RESET	441
G81 - G81/ DRILLING CYCLE	441
G82 DRILLING CYCLE	441
G83 DEEP DRILLING CYCLE	443
G84 TAPPING CYCLE	444
G85 DEEP HOLE DRILLING CYCLE	446
G86 BORING CYCLE	447
G87 BORING CYCLE	447
G88 DRILLING CYCLE FOR SPACED WALLS	449
G89 REVERSE BORING CYCLE	450
G94 FEEDRATE IN mm/min	452
G95 FEEDRATE IN micron/rev	452
G96 BORING CYCLE	453
G97 MILLING A BORE AND MILLING A THREAD CYCLES	455
G99 AXES STOP	459
Gxxx MACRO (Subroutine)	462
FUNCTIONS FOR PRESETTING	464
TOOL LIFETIME MANAGEMENT	467
MEASURE AND VERIFY CYCLES PROGRAMMING	469
G300 - TOOL LENGTH MEASUREMENT	470
G301 - TOOL LENGTH VERIFY WITH STOP IF OUT OF TOLERANCE	470
G302 - TOOL LENGTH VERIFY WITH SUBSTITUTION IF OUT OF TOLERANCE	471
G303 - TOOL RADIUS MEASUREMENT	471
G304 - TOOL RADIUS VERIFY CYCLE WITH STOP IF OUT OF TOLERANCE	472
G305 - TOOL RADIUS VERIFY CYCLE WITH AUTOMATIC TOOL SEARCH	472
G306 TOOL LENGTH AND RADIUS MEASUREMENT	473
G315-G319-G350-G354 CNC AXES DEFINITION	474
G320-G321 Q INCREASE SUM ABILITATION	475

G322-G323 HELICOIDAL WORK IN GRINDING ABILITATION	475
G327 TOOLS TABLE AND ORIGINS MODIFY AND SAVE FROM A PROGRAM.....	476
G327 FILE DEFINITION WITH NAME = TOOL POSITION	477
G328 G329 DL AND DR AUTOMATIC CORRECTION.....	478
G340-G341 - RADIUS TOOL CORRECTION SELECTION	479
G342-G343 ACTIVATION OR DESACTIVATION DL ON THE TOOL LENGTH.....	479
G344-G345 ACIVATION OR DESACTIVATION DR ON THE TOOL RADIUS.....	479
G349[] SUBROUTINE	480
G351 ROTO-TRANSLATION.....	481
G355 HEAD ROTATION WITHOUT ADJUSTMENT OF THE WORK PLANE.....	482
G536 SPECIAL HEAD ROTATION	483
G358 TABLE AXIS MOUVEMENT AS A SPINDLE	484
G359 RETURN FROM SUBROUTINE TO MAIN PROGRAM.....	484
G360 INTERPOLATION WITH N.U.R.B.S.....	485
G377 SETTING OF THE INSTRUMENT FOR THE TOOLS LENGTH PRESETT.....	486
G378 SETTING OF THE INSTRUMENT FOR THE TOOLS RADIUS PRESETT.....	486
G384 RIGID TAPPING CYCLE (start with spindle stopped).....	487
G390-G391 ABSOLUTES QUOTES PROGRAMMING.....	487
G399 Z axis rapid move to 5 mm from the positive end of course	488
G428/ G429/ PICK-UP-RELEASE PIECE CYCLE	488
G500 TOOLS LENGTH MEASUREMENT WITH MANUAL TOOL-CHANGE	489
G503 TOOLS RADIUS MEASUREMENT WITH MANUAL TOOL-CHANGE.....	489
G506 TOOLS LENGTH AND RADIUS MEAS. WITH MANUAL TOOL CHANGE	489
G550 CONFIGURATION AUTOMATIC RE-ESTABLISHMENT	490
G551 CONFIGURATION AUTOMATIC CHANGE.....	490
G690 BEARINGS CAGE REALIGNEMENT	491
G731 D.. TOOL RADIUS COMPENS. IN SPACE ZERO ON THE TOOL POINT.....	492
G740 CANCELS G746 and G748	493
G746 AXES SYSTEM ROTATION ASSOCIATED TO THE TABLE ROTATION.....	493
G748 AXES SYSTEM TRANSL. ASSOCIATED TO THE TABLE ROTATION	494
G751 TRANSLATION.....	495
G752 ROTATION.....	497
G755 HEAD ROTATION WITH SPHERICALS TOOLS	499
H ROUTINE.....	500
M FUNCTIONS	503
M10-M11 FUNCTIONS	504
M40-M42 SPINDLE GEAR	505
M58{ rotary axis } ROTARY TABLE REVOLUTIONS RESET	505
M184 M185 PIECES MAGAZINE.....	505
M200 WORK TIME	506
M320-M327 STOP IN MICRO.....	506
M... MEMORY - DISK FILES MANAGEMENT.....	507
M364 M300 {.....} TOOLS MODIFY FROM A PROGRAM.....	508
M382 M385 {.....} MESSAGES FROM A PROGRAM.....	508
M365 M300 {.....} ORIGINS ZEROSET FROM PROGRAM.....	509
M704-M704/1-M705 M708 M709 TAIL-STOCK MANAGEMENT	510
M728 M729 COOLANT SELECTOR MANAGEMENT	510
P PARAMETER	511
S S/ FUNCTIONS.....	517

PROGRAMMING

The programming is executed by instruction lines (blocks) contained in a work program.

An Instruction block is defined as all information included between an E.O.B. character and another one. The E.O.B. (End Of Block) character in phase of items input is automatically inserted by CNC at any time Enter key is pushed and doesn't appear visually on the monitor.

A whole of instruction blocks between two characters %, is a working program.

When a work program is written by keyboard the % character is automatically inserted by the CNC, the final one is automatically inserted only if the user forgets it.

Work programs written out of the CNC have to contain the two % sign at the start. This character appears on the monitor.

The informations contained in the block are composed normally by words and numbers; in particular cases, as in parametrical programming, special characters are used.

Information can be written inside the block in any order remembering that the repetition of the same word automatically cancels the one previously inserted.

Only in some cases it is possible to insert the same letters inside a block (routine H, M, G, QX characters etc.).

When programmed, an instruction is valid until another is programmed, except for L word that is zeroresetted by G fixed cycles programming.

Some letters could take different meanings according to the programming context.

Every block contains at the most 256 characters. The block numeration is obligatory only for routine blocks, for blocks called by them, for blocks inserted in the conditioned jump in the parametrical programming.

The block numeration can't be consecutive.

The execution of the blocks, that are not inside the two "%" is possible only using the routine "H". Example:

```
%N0                                ⇐ Start program
N10 G0X0Y0Z0PA1M3S1000F2000
N20 H10N1000N1001QX-3QY-4QZ-5
N30 G0X0Y0Z0
N40 H5N1000N1001QX3QY4QZ5
N50 .....
.....
N99999%                             ⇐ End program
N1000 G81 X10Y10Z-50R3L2           ⇐ Routine block
X20Y20
N1001 X-20Y-20
```

DEFINITION OF PROGRAMMING FORMAT.

These are the instruction that CNC can recognize followed by the quantity of the numerical characters admitted.

In the following table the first number represent the programmable digits at the left of the decimal point, the second number, the ones on the right. In the column on the right is indicated the meaning of every instructions. In some cases the instruction can have more than one meaning dependently on the contest where it is programmed.

ADDRESS	N. CHARACTERS	MEANING
A	+ / - 5.4	Rotary axis quota
B	+ / - 5.4	Rotary axis quota
C	+ / - 5.4	Rotary axis quota
D	+ / - 4.4	Tool Radius Value
E	+ / - 4.4	G51E Rotation Angle
		G61E Scale Factor.
		Angle executed in Geometry
		E Digitizing Chord Error
F	5	Work in Feedrate mm/min or Micron/rev
G	3	Preparatory Function
H	4	Routine Cycle
I	+ / - 5.4	Radius Centre X (*)
		Digitizing Increment
J	+ / - 5.4	Radius Centre Y (*)
K	+ / - 5.4	Radius Centre Z (*)
L	+ / - 4.4	Axis quota Orthogonal to the Plane
	0 or 1	Passing Limitation in Copying
	+ / - 4.4	Origin Distance in Geometry
M	4	Auxiliary functions
m	3	Spindle rotary axis
N	7	Block number
PA	2	Absolute Origin point
P	1.8	X Tool Disallignement correction
Q	1.8	Y Tool Disallignement correction
Q.	+ / - 4.4	QX QY QZ QA QB QC QD QE Increment addresses in Routines and in cycles
R	1.8	Z Tool Disallignement correction
	+ / - 4.4	Axis Quota Orthogonal to the Plane in Cycles
		Geometrical Radius
S	5	Spindle Speed in rpm
S/	5/2	Spindle Speed in rpm with control of the Spindle Absorption.
s	3	Time in tenths of second
T	2	Tool and Correctors of tool
U	+ / - 4.4	Axis X Limit Area in Digitizing

V	+ / - 4.4	Axis Y Limit Area in Digitizing
W	+ / - 4.4	Axis Z Limit Area in Digitizing
X	+ / - 4.4	X Axis Quota
		Axis X Limit Area in Digitizing
Y	+ / - 4.4	Y Axis Quota
		Axis Y Limit Area in Digitizing
Z	+ / - 4.4	Z Axis Quota
		Axis Z Limit Area in Digitizing

(*) N.B. This quotas have is displayed with 5.4 digits, but the maximum value imposable is 10 E+-38. Remember that in programming phase all quotas have to be imposed in millimetres or in degrees with the point (.) to separate whole from decimal numbers as well for metrical quotas as for degrees.

Examples:	12	12 Millimetres or Degrees
	42.337	42.337 Millimetres or Degrees
	0.05	0.05 Millimetres or Degrees

G FUNCTIONS

- G0** Rapid positioning of the axes in linear interpolation on the plane and in the space in the programmed point.
- G1** Work in linear interpolation in the plane and in the space in the programmed point.
- G2** Circular interpolation clockwise on the chosen plane or helix
- G3** Work in circular interpolation anticlockwise on the chosen plane or helix.
- G4** Dwell Time programmed with spindle revolutions (G4/n)
or in tenths of a second (G4/s time in sec./10)
- G5-G7-G9** G functions recognized only for reasons of compatibility with CNC 132 160.
They are executed like G0.
- G10-G11** Given to the geometrical programming (see the specific chapter).
- G13-G16** Given to the geometrical programming (see the specific chapter).
- G17** Specifies X Y as the machining plane (where Z is the perpendicular axis) active at the start.
- G18** Specifies X Z as the machining plane (where Y is the perpendicular axis).
- G19** Specifies Y Z as the machining plane (where X is the perpendicular axis).
- G20-G25** For the geometrical programming (see the specific chapter).
- G26** Feedrate on the tool tip also for rotary axes.
- G27** Real feedrate of the axes (for rotary DEGREES/MIN).
- G30** Cancel G31 function
- G31** Active working way (active at the start) Feed on the tip of tool.
- G38** Defines ways of copy alternately at parameters given in VCP table (OPTION)
- G39** Spindle positioning with angle M from 0° to 360°.
- G40** Positioning with Cancel Cutter Radius compensation
- G41** Tool Approaching during the work with introduction of radius tool correction on the left of the programmed path.
- G42** Tool approaching during the work with introduction of radius tool correction on the right on the programmed path.

- G43** Grinding cycle, like G41 and introduction of an oscillatory movement on the axis orthogonal at the work plane.
- G44** Grinding cycle like G42 with introduction of an oscillatory movement on the orthogonal at the work plane axe.
- G45** Pick up automatic tool cycle (subroutine).
- G47** Release automatic tool cycle (subroutine).
- G49** Function of connection programs in memory.
- G50** Cancel G51.
- G51** Rotation-translation according to programmed point (X,Y,E)
- G55** Rotation cycle rotary head without adjustment of the plane with the rotation angle.
- G56** Rotation cycle rotary head with adjustment of the plane with the rotation angle
- G58** Rounding off 360° on rotary axes with absolute measuring system
- G59** Return from subroutine
- G60** Cancel G 61.
- G61** Scale factor for X Y Z axes.
- G62** Cancel G63
- G63** High speed milling with smooth trajectory
- G65-G69** Given to digitizing cycles
- G70-G71** Inch-Metric Modes
- G75-G77** Given to digitizing cycles
- G79** Given to the digitizing cycles
- G80** Cancel canned cycles from G81 to G 89.
- G81** Drilling cycle
- G82** Drilling cycle (the same as G81)
- G83** Deep hole drilling cycle
- G84** Tapping cycle

- G85** Deep hole drilling cycle with high return
- G86** Boring cycle with spindle stop.
- G87** Boring cycle (the same as G86)
- G88** Drilling cycle for more walls
- G89** Reverse boring cycle
- G90-G93** G function not recognized
- G94** Feedrate in mm/min. (Active at the start)
- G95** Feedrate in micron/spindle revolution.
- G96** Boring cycle with detachment from wall
- G97** Macro instruction for milling circle or a thread
- G99** Input M functions or messages with Stopped axes
- G300** Measuring tool length cycle (option).
- G301** Verifying tool length cycle with stop (option).
- G302** Verifying tool length cycle with automatic research of substitutive tool (option).
- G303** Measuring tool radius cycle (option).
- G304** Verifying tool radius cycle with stop (option).
- G305** Verifying tool radius cycle with automatic research of substitutive tool (option).
- G306** Measuring tool length and radius cycle (option).
- G311** Feeler adjustment of the coefficients (option).
- G312** Zero recovery on a pin (option).
- G313** Zero recovery on a hole (option).
- G314** Zero recovery on Z (option).
- G316** Redefinition axes CNC <-> IND function
- G317** Redistribution axes function
- G318** Definition tern axes feeler function

- G319** Assignment synonyme axes function
- G320** Disable the G321.
- G321** Q increase sum abilitation.
- G322** Disable the grinding cycle.
- G323** Enable the grinding cycle.
- G324** Exclusion of the axes repositioning with LNS.
- G325** Zero recovery on a square (option).
- G326** Zero recovery on a edge (option).
- G327** Origins and tools lengths management.
- G328** Delta length correction on the tools table.
- G329** Delta radius correction on the tools table.
- G330** GPF repositioning.
- G331** Recovery G31 function with GPF.
- G332** Recovery G32 function with GPF.
- G335** Enable the registration of the machine quotas on disk.
- G336** Disable the registration of the machine quotas on disk.
- G340** Radius correction in profile from part-program.
- G341** Radius correction in profile from table.
- G342** Activation of the Delta L on the tool length.
- G343** Cancels G342.
- G344** Activation of the Delta R on the radius tool.
- G345** Cancels G344.
- G346** Automatic substitution of the tool with time expires.
- G349** Subroutine.
- G350** Dividing table axis initials modification.

- G351** Roto-translation.
- G352** Save the axes locking and lock all the axes.
- G353** Restore the axes locking saved with G352.
- G354** Activation of the matrix rotation on the programmed points.
- G355** Rotation cycle rotary head without adjustment of the head length (option).
- G356** Rotation cycle rotary head with adjustment of the plane with the rotation angle (Option).
- G359** Return to the calling program for files activated with G349.
- G360** Work by N.U.R.B.S. with 3 to 5 axes movement.
- G362** Introduction of the polygonal vertex in digitizing (option).
- G363** Introduction of the depth in digitizing (option).
- G364** Introduction of the second polygonal (option).
- G370-G376** Digitizing cycles (option).
- G377** Length tools pre-setting qualification (option).
- G378** Radius tools pre-setting qualification (option).
- G384** Rigid tapping cycle (option).
- G390** Predisposition for the programming in absolutes quotas.
- G391** Cancels the G390.
- G392** ON/OFF feeler zero recovery on a pin (option).
- G393** ON/OFF feeler zero recovery on a hole (option).
- G394** ON/OFF feeler zero recovery in Z (option).
- G395** ON/OFF feeler zero recovery on a square (option).
- G396** ON/OFF feeler zero recovery on an edge (option).
- G398** Piece research with ON/OFF feeler and quotas determination (option).
- G399** Rapid to 5 mm from the positive Z axes limit.

- G428/** Pick-up piece function (option).
- G429/** Pick-up piece function (option).
- G500** Measuring tool length cycle with manual tool change (option).
- G503** Measuring tool radius cycle with manual tool change (option).
- G506** Measuring tool length and radius cycle with manual tool change (option).
- G550** Configuration automatic re-establishment.
- G551** Configuration automatic change.
- G571-G578** Digitizing cycles (option).
- G731** Tool radius compensation in space with zero on the tool point.
- G703** Tail-stock forward with reduced air pressure (optional)
- G704** Tail-stock forward with normal air pressure (optional)
- G705** Tailstock backward (optional)
- G740** Cancels G748.
- G746** Origins translation and rotation associated to the table rotation.
- G748** Origins translation associated to the table rotation.
- G751** 3D Translation.
- G752** 3D Rotation.
- G755** Head rotation in centre of spherical tools .

G FUNCTIONS SPECIFICATION

G0 RAPID POSITIONING

G0 identifies a Linear Interpolation of the controlled axes executed in Rapid speed. In case of moving of more axes together, the Rapid speed will be adapted to the slower axis.

At every moment during of the work it is possible, with the "Rapid Override" potentiometer on the frontal surface of the CNC, to change the speed from 0 to 100%.

The **G0** function allows replacing the followed functions of CNC 132-160 :

- G5 = Rapid A and B auxiliary axes
- G7 = Rapid of the perpendicular axis to the interpolation plane
- G9 = Rapid of the interpolation plane axis

The G5, G7, G9 are however recognized by the CNC as G0

Example: N10 G0X0Y0Z0PA1

The G0 function and consequently the G5, G7 and G9 functions produce the annulment of the Radius Tool Correction previously programmed.

Example: N20
N30 G41X Y D F ⇐ Approach with radius compensation introduction on the left of the profile trajectory

N40 G1X Y
N50 X Y ⇐ Profile
N60 G2X Y I J
N70 G0X Y ⇐ Rapid Tool Centre positioning to the X Y programmed quotas.

N80 G1X Y ⇐ Linear Tool Centre Interpolation to the X Y programmed quotas, without radius compensation.

G1 LINEAR INTERPOLATION.

G1 identifies a Linear Interpolation of the controlled axes on the Plane or in the Space executed in Feed.

The Feed speed is entered in the program with the "F" function.
In each moment of the work it is possible, with the Rapid Override potentiometer on the frontal surface of the CNC, to change the speed from 0 to 120%

Example: %N0
 N1 G0X0Y0Z0PA1M3S1000
 N2 G1F150X10Y10

G2 CIRCULAR INTERPOLATION CLOCKWISE.

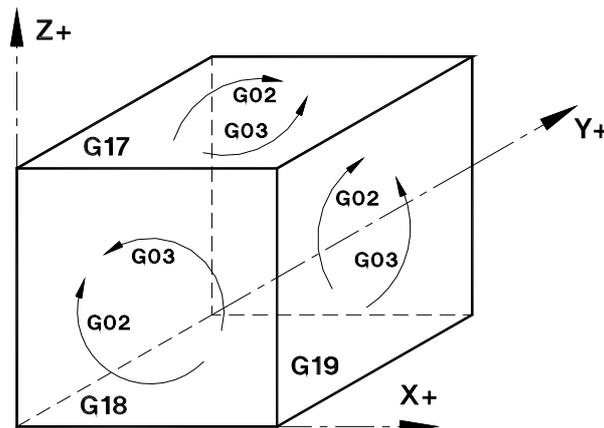
G2 identifies a circular Interpolation clockwise of the controlled axes on the plane or in the space executed in Feed.

It is necessary to input the value of final point and the centre of circle (or arc of circle) that must be executed.

It is accepted a difference 0.1 mm max. between initial and end-radius on the tool radius offset trajectory.

The G2 function will be executed with circular CLOCKWISE moving, starting from the last recorded point.

It is possible to program arc 0.001mm to 99999.999mm radius value.



Example: %N0
 N1 G0 X0 Y0 Z0 PA1M3S1000F150
 N2 G2 X20 Y0 I10 J0

It is possible also to program a helix using the K (in G17) function. Example:
Complete clockwise helix with Radius=30 mm and the step=25 mm

```
N10 G0 X0 Y0 Z0 PA1 M. S.... F....  
N20 G2 I30 J0 K-25
```

The values of the last K remain activated in the entire program; the followed G2 and G3 functions will be executed like helical interpolation. Program K0 for delete it.

G3 CIRCULAR INTERPOLATION COUNTER CLOCKWISE.

The G3 function is the similar to G2, but the moving is Circular Counter clockwise.

Example: %N0
 N1 G0 X0 Y0 Z0 PA1M3S1000F150
 N2 G3 X-20 I-10 J0

G4 STOP CYCLE.

G4 identifies a Stop Cycle in program with the length equal to the number of Spindle revolutions entered in the same function or a stop cycle express in tenth of second.

Example: %N0
N1 G0X0Y0Z0PA1M3S1000F150
N2 G4/250 Wait for about 15 seconds (250 spindle revolutions)
N3 G0X10Y10
N4
N5 G0X15
N6 G4/s150 Wait for 15 seconds (express in tenth of sec.)

G5-G7-G9 RAPID FUNCTIONS.

Same functions as "G0", taken for the compatibility with the old Elexa NC (E132, E132V and E160).

Example : %N0
N1 G0X0Y0Z0PA1
N2 G5A10B25

Example: %N0
N1 G9X0Y0PA1

G17 XY INTERPOLATION PLANE.

The G17 function indicates the Horizontal XY Interpolation Plane (Z will become the Orthogonal to this plane).

At the beginning the CNC automatically selects the G17 plane and all the following considerations will be reported to this plane.

G18 XZ INTERPOLATION PLANE.

The G 18 function indicates the Longitudinal XZ Interpolation Plane (Z will become the Orthogonal to this plane).

The selection of G18 plane can be executed whether on the head or during the program and it is erased only by G17 or G19.

Example :

%N0		%N0	
N10 G0X0Y0Z0PA1		N10 G18	↩
N20 G1Z30F300M3S1000		N20 G0X10Y10Z30PA1	
N30 G18	↩	N30 G1M3S1000F150Y-10	
N40 G0X25Y0		N40 G0Y50	
N50 G1F150Y-10		N50	
N60 G0Y10			
N70 G17			
N80 G0X50Y50Z50			
N90			

G19 YZ INTERPOLATION PLANE.

G19 indicates the Transverse YZ Interpolation Plane (X will become the Orthogonal to this plane).

The selection of G19 Plane can be executed whether on the head or during the program and it is erased only by G17 or G18.

Example :

%N0		%N0	
N10 G0X0Y0Z0PA1		N10 G19	↩
N20 G1Z30F300M3S1000		N20 G0X0Y0Z30PA1	
N30 G19	↩	N30 G1M3S1000F150X-10	
N40 G0Y25Z0		N40 G0X50	
N50 G1F150X-10		N50	
N60 G0X10			
N70 G17			
N80 G0X50Y50Z50			
N90			

G26-G27 FEEDRATE ON THE TOOL TIP

G26 - The programmed speed is the real speed on the tool tip.

Ex. N100 G26

After this function the programmed speed is applied on the trajectory of the tool tip also for rotary axes movements (the G26 function is active at the start).

G27 - The programmed speed is real and doesn't account the distance of the tool from the Tables / Heads rotation centres.

Ex. N120 G27

After this function the programmed speed is applied on the axes trajectory (for rotary axes is degrees/min).

G30-G31 AUTOMATIC VELOCITY REGULATION.

The G30 function erase the G31 function, the G31 is active at the start.

The G31 function is used for normal machining .
With G30 the CNC executes a stop at the end of each block. With the G31 function the numerical control reads up to 32 following blocks in order to calculate in advance the path of the cutting tool.

In the case where there is an abrupt change of direction the CNC makes an automatic reduction in feedrate.

With G31 the movements of rotary axis A B C is computed in order to have the programmed Feed on the tip of tool, depending of distance of rotary centre.

With G30 the movements of rotary axis A B C is computed with Feed value in degree/minutes.

It is necessary to program the M11 {axes name} to unlock the axes during the work execution in G31 mode.

The G31 function checks if the angle between two followings elements and modify the feedrate proportionally to the angle of the two directions. For angle greater than 90° the feedrate on the intersection point of the two elements is always 0.

With the E function programmed on the G31 block it is possible to program the angle under which the automatic feedrate regulation is not activated, the intersection point between the two elements that have an angle smaller than the one programmed will be executed with the programmed feedrate.

Example con XYZ	Example con XYZAB	Example con XYZAC
%N0	%N0	%N0
N1 M10	N1 M10	N1 M10
N2 M11{XYZ}	N2 M11{XYZAB}	N2 M11{XYZAC}
N3 M6 T1	N3 G45T1	N3 G45 T1
N5 G0 X..Y..PA1	N4 G0X Y A B PA1	N4 G0X Y A C PA1
N6 G0Z.. M3 S1500	N5 Z.. M3 S3000 F1500	N5 Z.. M3 S12000 F3000
N7 G1X..Y..Z..F1000	N6 G1X..Y..Z..A..B..	N6 G63 E0.01
N8 X..Y..Z..	N8 X..Y..Z..A..B..	N7 G1X..Y..Z..A..C..
N9 X..Z..	N9 X..Y..A..B..	N8 X..Y..Z A C
N10 G0Z100	N10 G47	N9 X Y Z A C
N11 G99PA2	N11 G45T2	N10 X Y Z A C
N12 G0X..Y..	N12 G0X..Y..A..B..PA1	N11 X Y Z A C
N13 Z..	N13 Z..M3 S1900F1200	N12 X Y Z A C
N14 G0X..Y..	N14 G1X..Y..Z..	N13 X Y Z A C
N15	N15	N14

Q1 PARAMETER INSIDE THE G31 BLOC

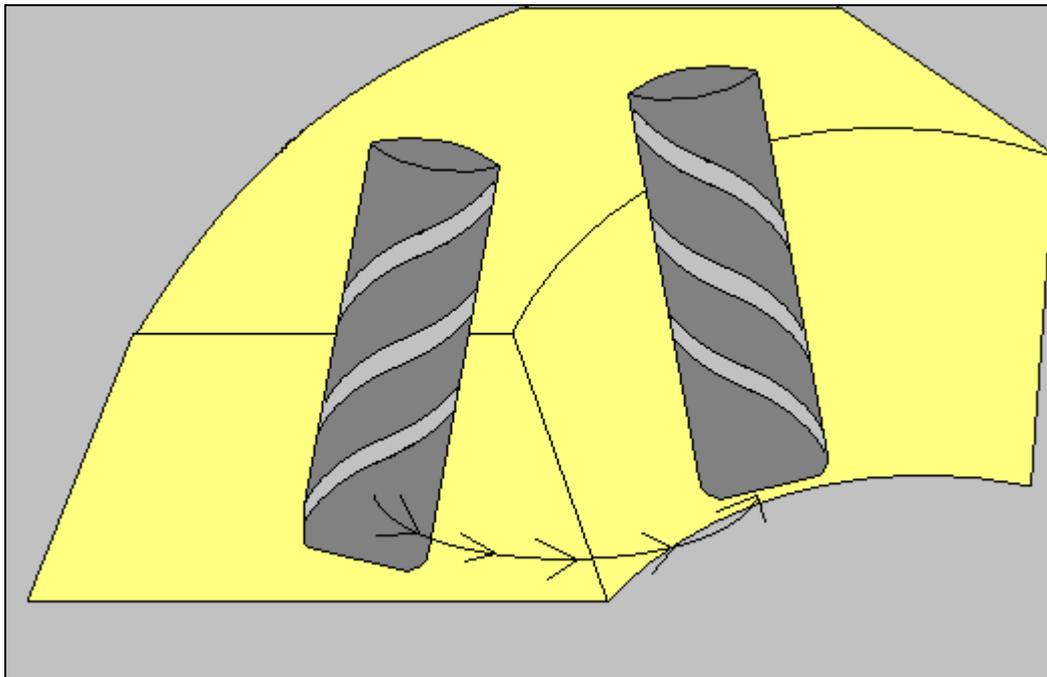
Input the parameter **Q1** in the **G31** bloc means to choose a particular mode to interpret the feed F of program.

This mode can be delete with: **G31Q0** or **G30**, also the Shift+F11 produce the delete of this mode.

The **G31 Q1** generates a continuous movement of axes in presence of discontinuous coordinates on the program.

With this mode the CNC uses the smaller feedrate between the G26 and G27 functions.

A classic example is a 5 axes milling (with the head) of a wall of a surface, when moves to the other side of the surface in correspondance of the passage between the two faces there is a jump of some degrees of the axis of the head, where the CNC made an acceleration caused by the component of cut of the movement of the tool in relation to the piece goes to the 0 (because of the compensation:



In G31Q1 mode, when the component of cut goes toward the 0 (information of the G26), the CN takes like speed of reference the vectorial sum of all speed components (information of the G27) for a regular movement everywhere..

G31D.. RADIUS COMPENSATION IN SPACE

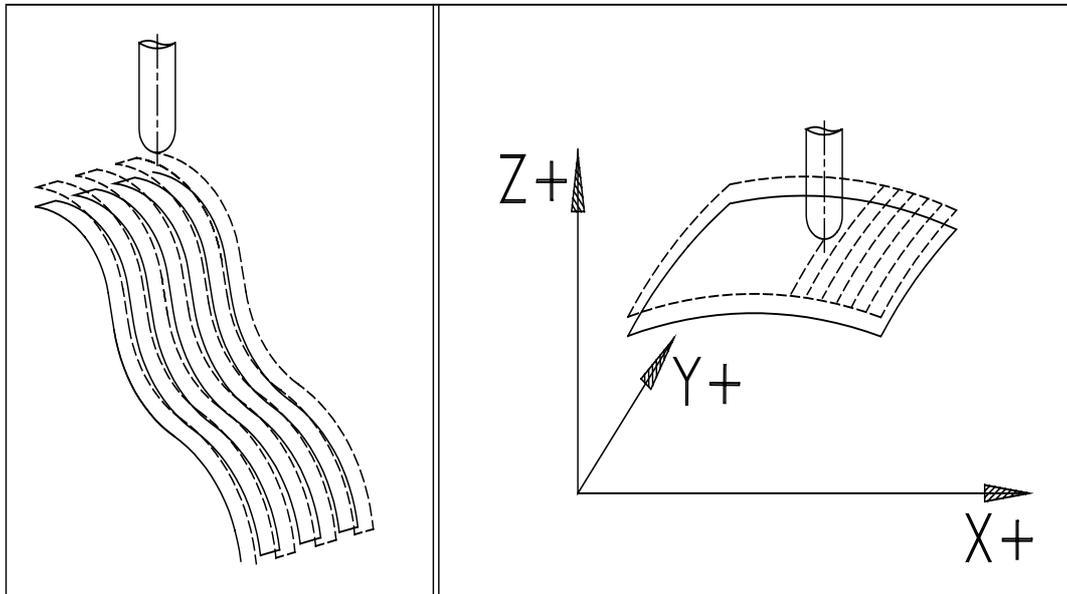
This function is activated by programming: **G31D ..** where D is the tool radius correction.

The 3D program must be composed by the 3 axes (X, Y and Z) and by the correction factors (P, Q and R) ex: **N10 X.. Y.. Z.. P.. Q.. R..**

These correction factors P, Q and R can change from -1 to 1.

No tool collision checks are done on the points of the surface.

No possible to use together G63E.. function



G30-G32 NO AUTOMATIC VELOCITY REGULATION.

The G30 function erase the G32 function and it is active at the start.

The G32 function can be used for the execution of each program.

By using this function all the feedrate movements (G1, G2, G3) are executed, by the CNC, by excluding the velocity command for the axes to move.

In this way the axes cover the programmed trajectory in continuous motion by ignoring the angle between the adjacent trajectory elements.

The worked trajectory will be the one programmed with a little rounding on the programmed trajectory corners.

This rounding is proportional to the work speed. The CNC gives an alarm if the work speed value exceeds the 2000 mm/min.

It is necessary to program the M11 {axes name} to unlock the axes during the work execution in G32 mode.

G39 SPINDLE QUILL KEY POSITIONING.

The G39 function allows to program an "m" angle from (x+), between 0 and 360 degrees, that is the spindle quill key positioning.

Example: N105 G39m35

G40-G41-G42 RADIUS COMPENSATION.

The G40 function erases the G41 and G42 functions and executes a feedrate move to the programmed point.

The G41 function identifies an approach to the piece on the left side of the programmed path with the input of the value of the tool radius D.

The working speed of the approach can be programmed in the G41 block or before it.

The G42 is as the G41 but the D radius compensation is inserted on the right of the profile trajectory to execute.

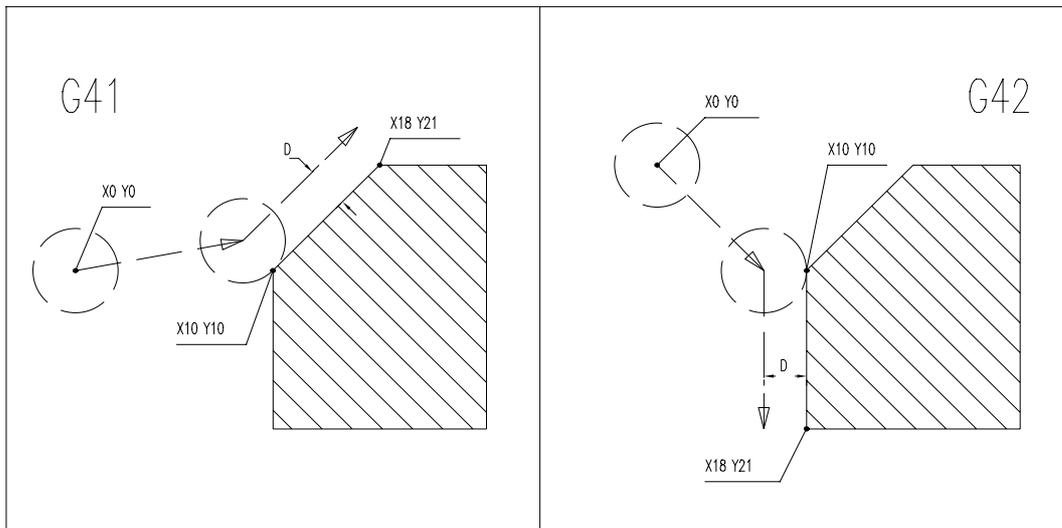
The G40, G rapid, fixed cycle G, change tool G, G50, G60 functions erase the G41 or G42.

The G42 is as the G41 but the D radius compensation is inserted on the right of the profile trajectory to execute.

Example:

```
N10 G0X0Y0Z0M3S1000F150PA1  
N20 G41X10Y10D3  
N30 G1X18Y21  
N40 G40X0Y0
```

```
N10 G0X0Y0Z0M3S1000F150PA1  
N20 G42X10Y10D3  
N30 G1X18Y21  
N40 G40X0Y0
```



G43 - G44 GRINDING CYCLE.

The G43 function identifies the Grinding cycle with approaching of the piece from the left side and introduction of the D tool radius.

Also in this case are valid the same consideration for G41.

The G44 function is equal to G43 excepted when the approach of the piece is from the right side.

For the programming it is needed to replace G41 G42 with G43 "G44/n " and input the QZ value, where:

/n = Number of cycles/minute (Max. 999)

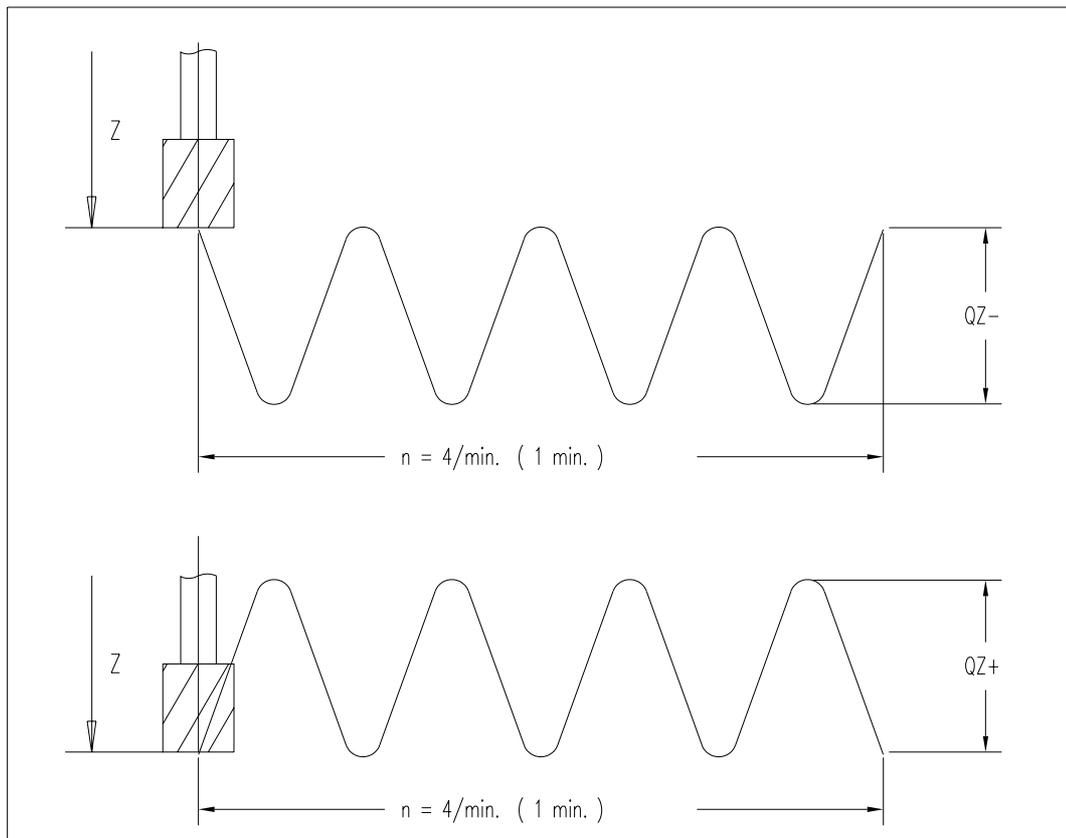
Qz = Z deep excursion

Example:

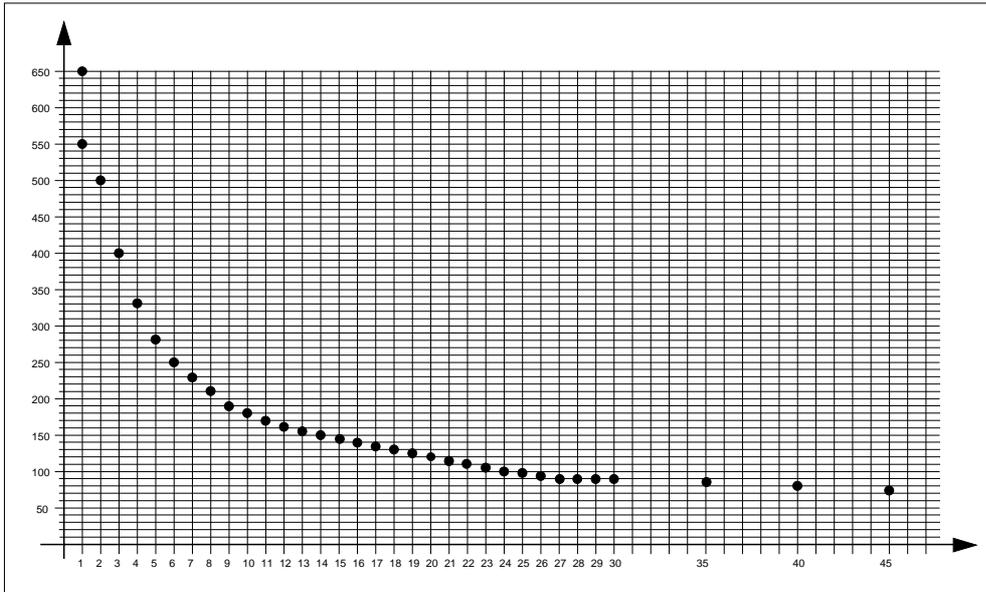
```
N10 G0X0Y0Z0PA1M3S1000F150  
N20 G43/300X10Y10QZ-4D3  
N30 G1X18Y21
```

```
N10 G0X0Y0Z0PA1M3S1000F150  
N20 G44/300X10Y10QZ-4D3  
N30 G1X18Y21
```

The G40, G rapid, fixed cycle G, change tool G, G50, G60 functions erase the G43 or G44.



CYCLES/MINUTE (/n)



DEEP EXCURSION IN Z (mm)

G45-G45/ PICK UP TOOL CYCLE.

It indicates the Pick up of Tool Automatic Cycle.

This function is used only if the Machine Tool has the Tools Magazine.

The tool machine can have two tools magazine, that could be equipped with guarding (see Note 2).

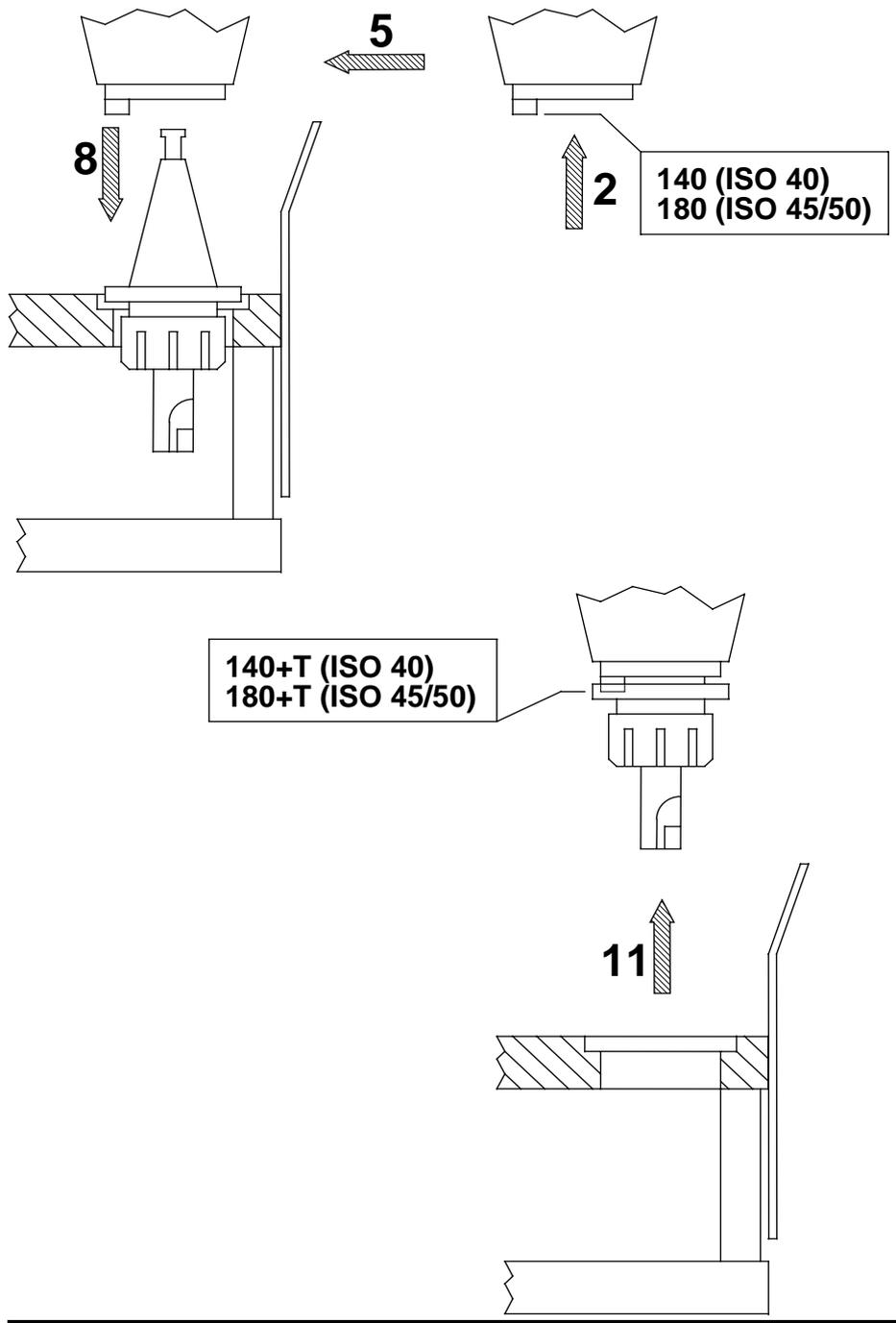
By programming the **G45/...** (Increment high quota in mm ex. G45/50T1), the tool will move higher, of the programmed value, from the normal mode.

The CNC will execute these operations in sequence:

- 1) Check for empty tool
- 2) Z rapid with Change of dimension assuming PA0 or PA9 origin (see Nota1).
- 3) Spindle and coolants stop and quill key positioning (see Notes 3-4)
- 4) Traverse on the XY position with co-ordinate of the requested Tool
- 5) Blow the air for the Cone cleaning
- 6) Traverse to Z0 plane with PA0 (equivalent to the tool holder plane)
- 7) Release the electro valve for tool pick-up
- 8) Z rapid with Change of quote (This operation is automatically done after the tool reset)

Example:

```
N10G45T1  
N20G0X0Y0Z0PA1M3S1000F150  
N30.....
```



G47 TOOL RELEASE CYCLE.

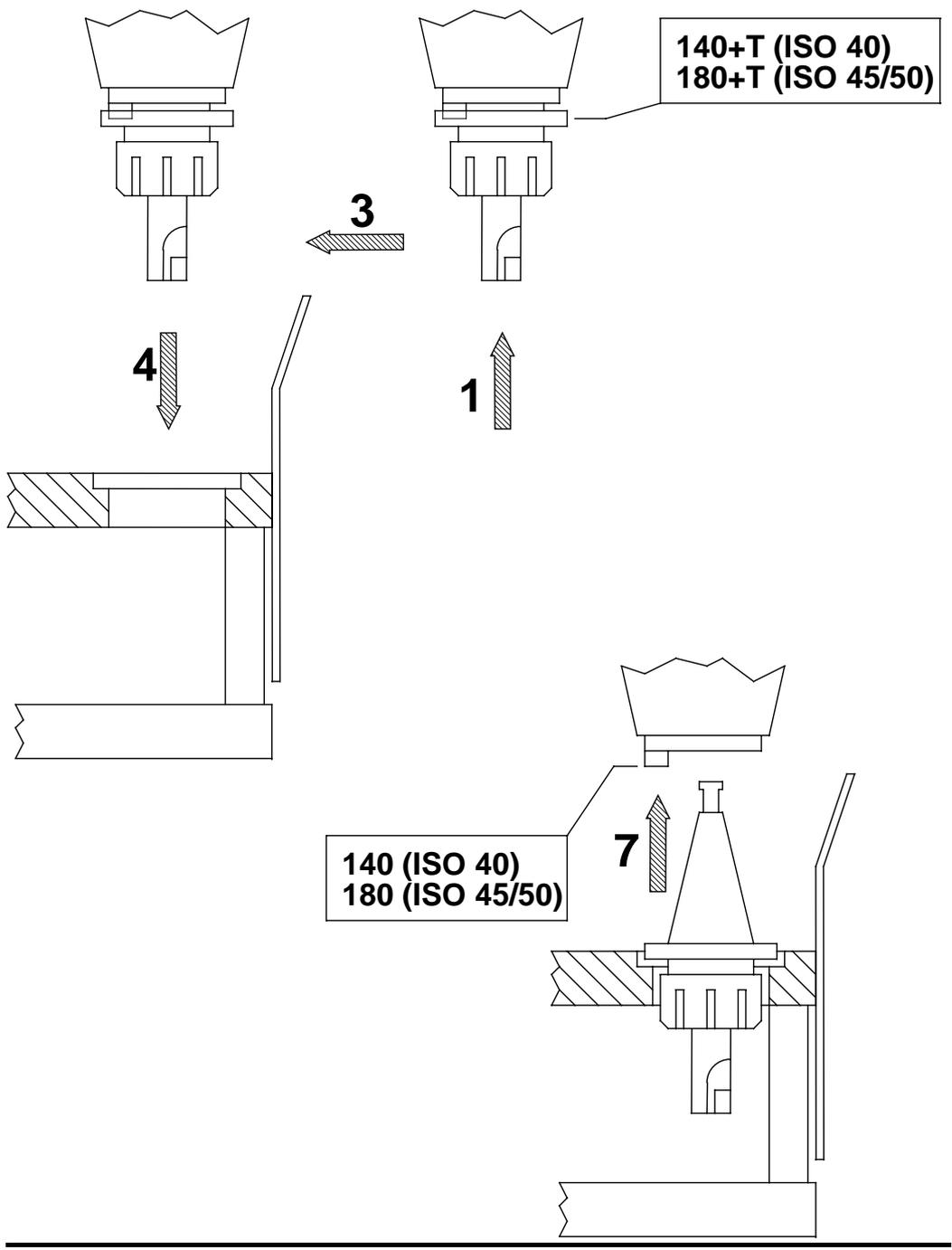
It identifies the Automatic Release Tool Cycle.

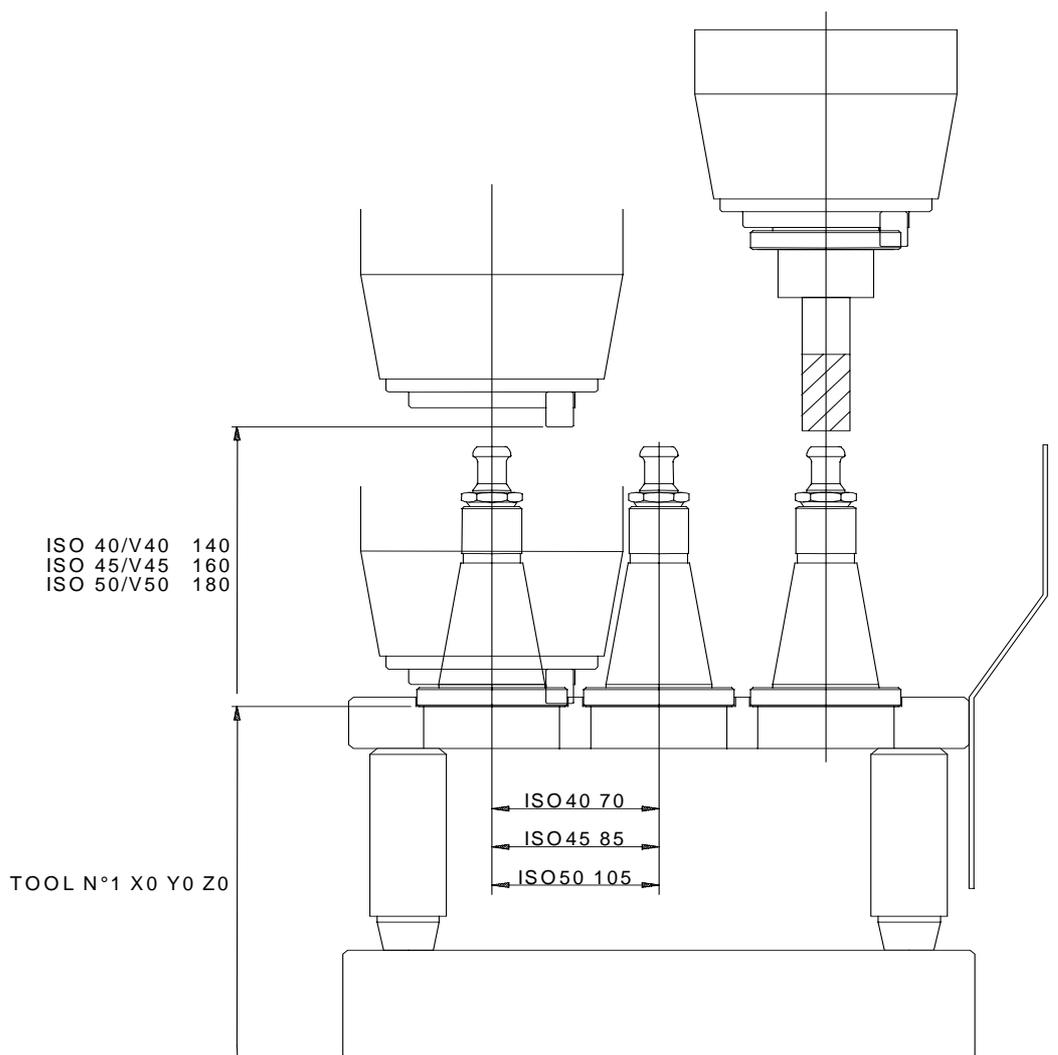
Also in this case the considerations done for Cycle " G45 " are valid.

The CNC will execute these operations in sequence:

- 1) Z rapid with Change dimension assuming PA0 or PA9 origin
- 2) Spindle and coolants stop and key positioning
- 3) Traverse XY on Plane of co-ordinate of the selected tool
- 4) Traverse to Z0 with PA0 (equivalent to the tool Holder Plane)
- 5) Electrovalve on and tool release
- 6) Check "Free Cone"
- 7) Z Rapid to the change quota
- 8) Electrovalve off

Example: %N0
 N10 G45T1
 N20 G0X0Y0M3S1000F150PA1
 N30 Z1
 N40 G1Z-1
 N50 X10Y10
 N60 G0Z1
 N70 G47
 N80 G45T2
 N90 G0X50Y50
 N100.....





NOTES ABOUT G45 and G47:

- 1) The G45 and G47 forced the absolute origin PA0 or PA9 dependently of the programmed tool number. As it is possible to have two tools magazines, with equal positions number, that number is defined in the Configuration file.
The first tools magazine must have the PA0 origin and the second the PA9 origin.
- 2) In the machine with two tools magazines it is obligatory that the one on the left have the PA0 origin and the second one on the right the PA9 origin.
- 3) The G45 and G47 functions cancel eventual interpolation plane variation; the G18 or G19 functions must be reprogrammed after the G45, or G47 functions.
- 4) The G45 and G47 functions cancel the origins translation G746 or G748
The G746 or G748 functions must be reprogrammed after the G45 or G47.
- 5) The G45 and G47 functions move the head in vertical position.
After tool change must be reprogrammed the head positioning if necessary.
- 6) The G45 and G47 functions cancel the radius compensation G41 G44.
- 7) After the G45 or G47 must be reprogrammed the functions that enable the spindle rotation and the coolants.
- 8) During the execution of G45 or G47 the function M10 is forced.
If we break this cycles is necessary reprogrammed the functions M11{.....}.
- 9) Any functions programmed on the G47 line are ignored by the CNC, on the G45 lines are accepted only the tool programming (T).

G49 [.....] PROGRAM LINK.

It must be programmed before the final %.

Example: N235G47
 N236G49[**BLOCK\DDD**]
 N99999%

If a program, containing the G49 [program name], is charged on the CNC memory (1PROG-LOAD) when the G49 function is executed the CNC automatically charge the program indicated inside the parentheses and is executed immediately.

A few seconds passes between the end of the first program and the start of the second.

Call between different programs:

Example:

<u>Program AA</u>		<u>Program BB</u>		<u>Program CC</u>
%N0		%N0		%N0
.....	
N99998 G49 [BLOCK\BB]	⇒	N99998 G49 [BLOCK\CC]	⇒
N99999%		N99999%		N99999%

Diagram description: The diagram shows three program blocks labeled Program AA, Program BB, and Program CC. Each block contains a sequence of lines: a program number (%N0), a series of dots (.....), a G49 call with a block name in parentheses, and a final percentage sign (%). Program AA calls Program BB, Program BB calls Program CC, and Program CC ends with a final percentage sign. Arrows indicate the flow of execution: a right-pointing arrow (⇒) from Program AA to Program BB, a right-pointing arrow (⇒) from Program BB to Program CC, and a right-pointing arrow (⇒) from Program CC to the final percentage sign. Additionally, upward-pointing arrows (↑) are placed between the dots (.....) of Program AA and Program BB, and between the dots (.....) of Program BB and Program CC, indicating the transition between programs.

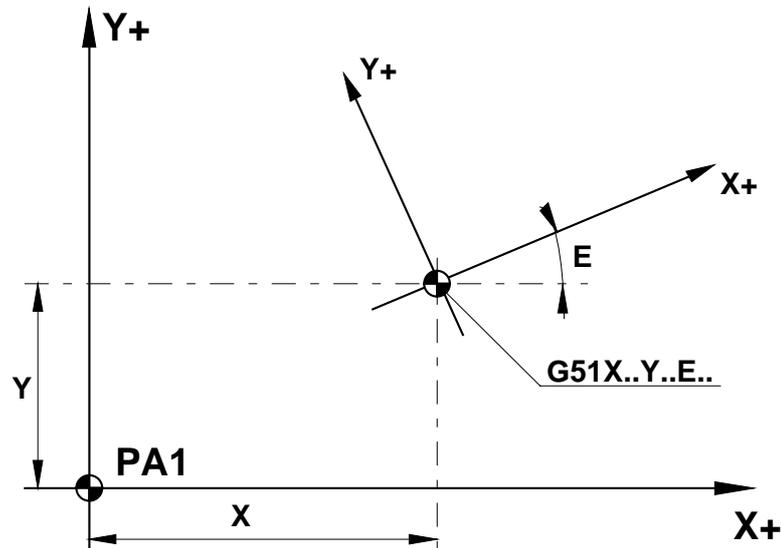
G50-G51 ROTOTRANSFORMATION.

The G50 function erases the G51.

The G51 function indicates a Rototransformation of the absolute reference system according to a new Cartesian system of programmed co-ordinates in the same block.

One time the G51 function is activated, all the followings elements must be programmed according to the new reference system. The translation and rotation values are programmed in the line containing the G51 function:

G51X Y (translation) E (rotation angle)



It is possible to program the rototransformation in routines by using the QX QY QE:

```
Example: N10.....
         N20 G51 X0 Y0 E0
         N30.....
         N40.....
         N50.....
         N60 H1 N20 N50 QX10 QY10 QE30
```

The QX, QY, and QE values are reseted at the exit from the routines.

By using the G51 function it is impossible to rototranslate parts of programs with Q incremental repetitions:

```
Example: %N0
         N10 G51 X Y E
         N20 G81 X Y Z R M S F PA1
         N30 H5 N20 QX QY
         N40 .....
```

Execute five G81 to the X Y quotes of the N20 line, for a correct program it is necessary to refer the routine on the N30 line to the line with the G51.

The G51 function execute first the translations and successively the rotations.

This function is active on the perpendicular plane of the tool either in G17 or G18 or G19 planes. The G51 remains activated since the CNC read the G50 function.

The G50 function is active at the CNC start.

It isn't possible to use the function G51 together with G351, G751 or G752.

G55 HEAD ROTATION

It identifies the Rotation Cycle of the Head, without adjustment of the Work Plane to the Rotation angle.

The G55 cancels the G56 G355 G356 G755 functions and the origin translation.

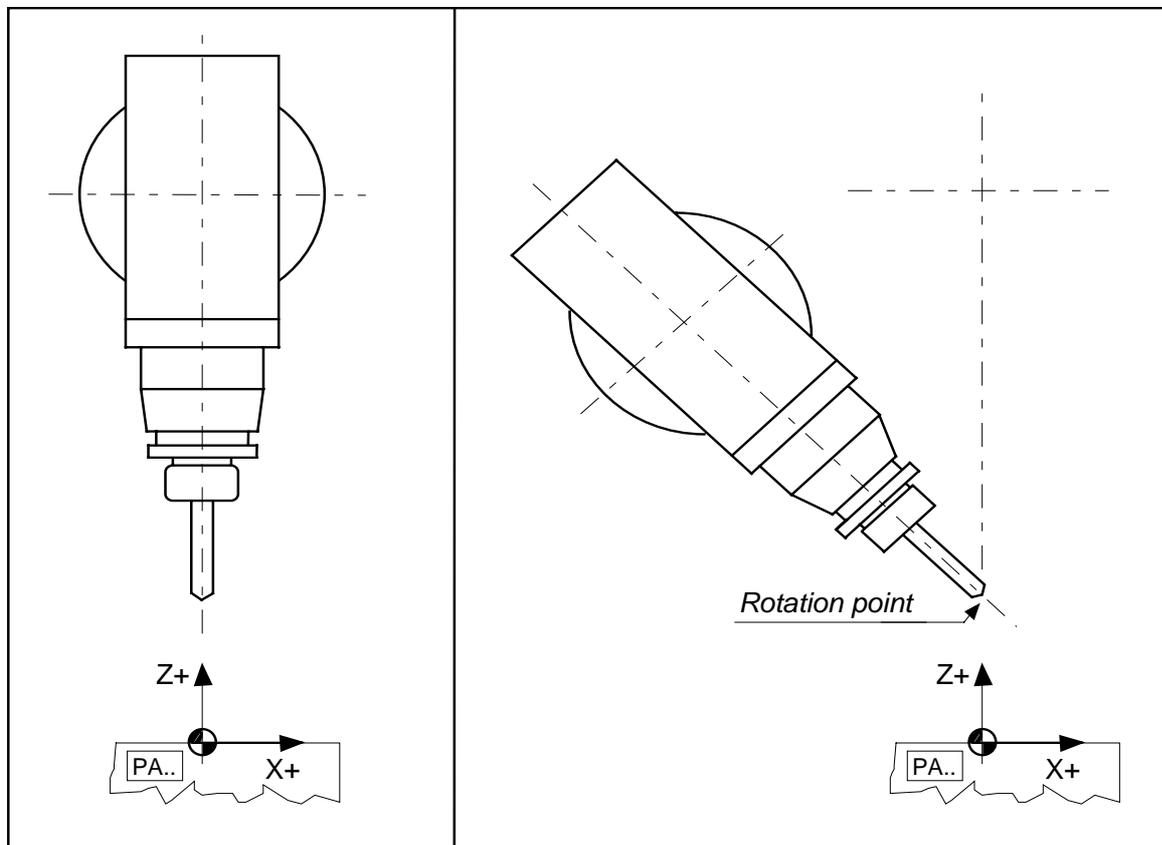
This function is modal, is active at the CN start and it is erased by the G55 G355 G356 G755

On machines series A B S the sign is positive for a clockwise rotation and negative for a counter-clockwise rotation.

Example: %N0

```
.....  
N10G55C-60  
N20G0X50Y100Z50 PA1
```

The head is turned at -60 degrees.



G56 HEAD AND ORIGIN ROTATION

It identifies the rotation cycle of the head, with adjustment of the Work Plane to the Rotation angle.

This function is modal and it is erased by the G55 G355 G356 G755 or by the RESET.

In the G56 block it is possible to program an origin translation in X, Y and Z.

This translation is calculated respect the original reference system before the rotation head angle

The G56 function is allowed only in the X Y interpolation plane (G17).

No possible to use together G746 function.

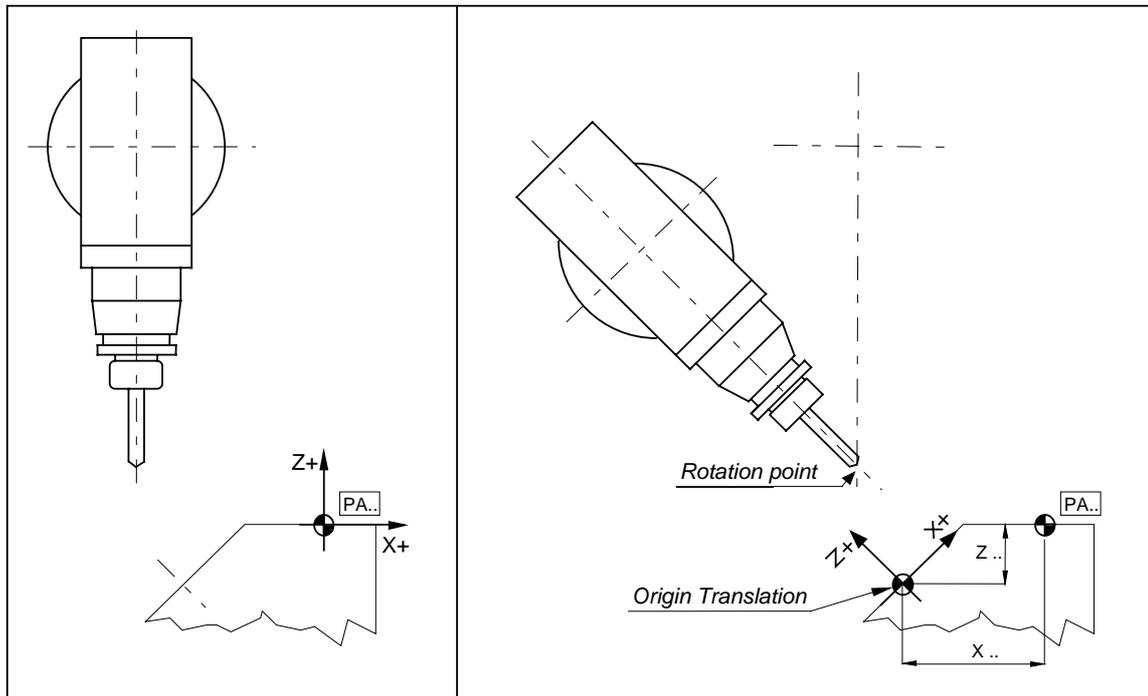
Example: %N0

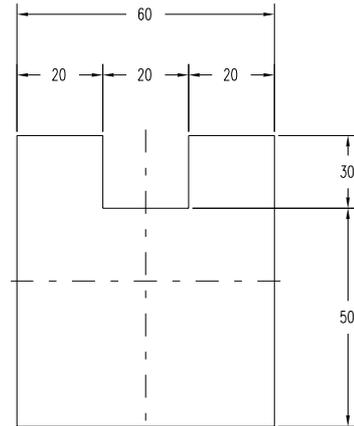
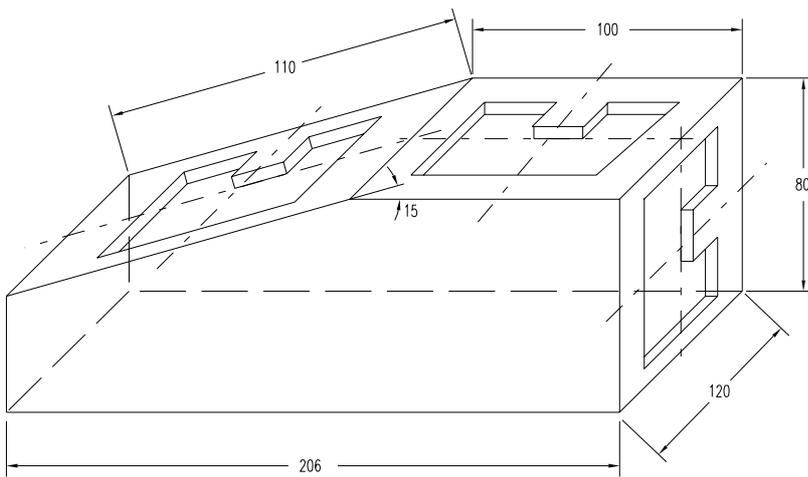
```
.....  
N10 G56 X-85 Y-10 Z-28 C-45  
N11 G0 X0 Y30  
N12 Z5
```

All the lines programmed after the N10 will be executed with an origin translation about -85mm in X, -10mm in Y, -28mm in Z and with the X Y plane rotated respect the Y axis since be orthogonal to the head position (C45°).

With a axis C rotation of 90 degrees the axis X will move when it is programmed a Z axis movement and vice versa.

The CNC visualize, on the screen, the quotes like if the head was always in the 0 angle position.





```

%N0
M10
M11{XYZ}
N1 G45T1
N2 G0X0Y0PA1
N3 G0Z2F300S3501M13
N4 G1Z-2
N5 G41X8Y2D5
N6 G3X0Y10I0J2
N7 G1X-10
N8 Y40
N9 X-30
N10 Y-40
N11 X30
N12 Y40
N13 X10
N14 Y10
N15 X0
N16 G3X-8Y2
N17 G40X0Y0
N18 G0Z10
N19 G0X60
N20 G56X50Y0Z-40C90
N21 H1N2N18
N22 G0X-50
N23 G56X0Y0Z0C0
N24 G0X-100
N25 P1=- (50+(55*COS15))
N25 P2=-55*SIN15
N26 G56XP1ZP2C-15
N27 H1N2N18
N28 G0Z20
N29 G56X0Y0Z0C0
N30 G47
N9999%

```

G58 ROTARY TABLE REVOLUTIONS RESET.

The G58 function is used for zero setting the revolutions counter of the absolute rotary axis, defined with type 37 code in configuration.

Example: %N0
 N10 G0A840 that is $360^\circ \times 2 + 120^\circ$
 N11 G58 Display A 120° or -240°

Without the G58 or M58 {axis name} the rotary axis can rotate from -20000 to 20000 degrees (end of travel limits).

The G58 function must be used for every PA.. origine definition in case of the rotary axis has done many revolutions, this must be done at the beginning of the program or in MDI.

G59 RETURN FROM A MACRO.

The G59 function allows returning on the main program from a <Macro> cycle and must be programmed at the end of this macro.

G60-G61 SCALE FACTOR.

The G60 function erases the G61.

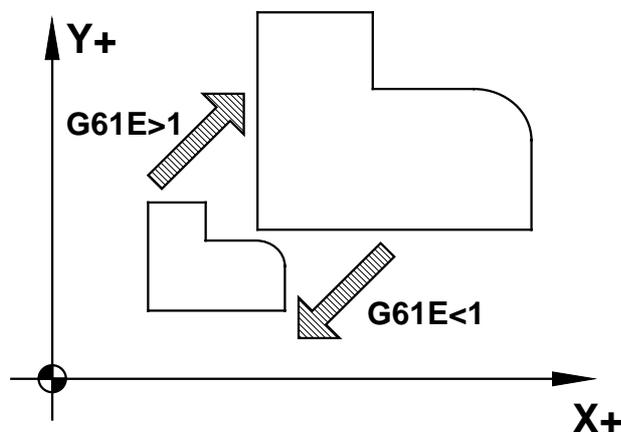
The G61 function identifies the change of scale for the programmed axes, it is possible to reduce or increase the real physic dimensions according to the programmed value.

Example: %N0
 N10 G61 E1.5 increase factor (because >1)
 N20 G0 X10 Y20 PA1 moves to X15 Y30
 N30.....

The scale factor E is active only for X Y Z axes and modify this three axes at the same time.

Besides it possible to define a different factor for each axis.

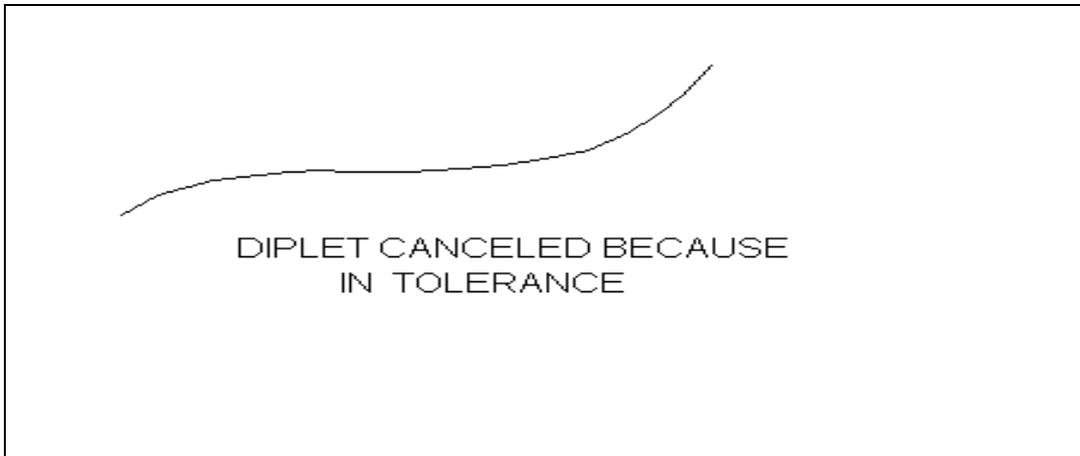
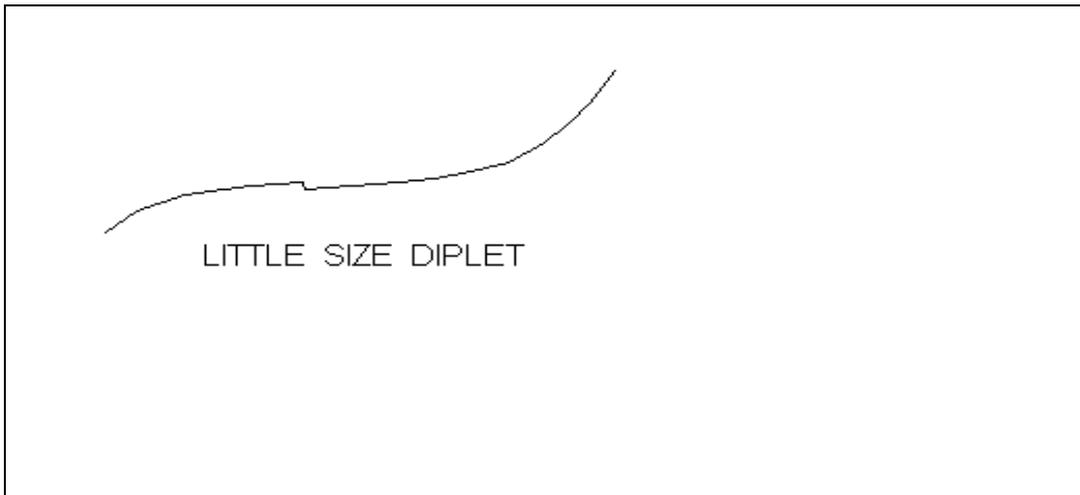
Example: G61X1.5Y2Z3 or G61A-1



G63 HIGH SPEED MILLING OF PROFILES AND SURFACES (OPTIONAL)

The new G63 has been realised to perform the finishing and the time of execution. The result is a smooth and continuous movement of the machine even with high cutting speed. The main innovations are:

- 1) The ISO program trajectory (points "G1" and/or circles "G2-G3") is processed internally and the result is a tool path with high degree of continuity (till the Jerk).
- 2) The geometrical interpolation soft the tangency and curvature discontinuity generated by the CAM.
- 3) Are introduced new algorithms for the management of angular points that the NC deletes if on the programmed tolerance.



It is necessary to weigh up the use of the operator for the tolerance parameter E on the G63 function:

N10 **G63 E0.02**

In the previously G63 release (as in RIPUNTI, program on the NC to convert in Spline "G360") the tolerance parameter can be express in hundreds of millimetre or tenths of millimetre during roughing or half-finishing execution: in those working phases when is presents a machining allowance of tenth or hundreds of millimetre.

In finishing, instead, the E parameter must not be bigger than few microns (from 1 to 3).

In the new G63 release the tolerance parameter has a meaning deeply different and takes the concept of “maximal local deformation where it is possible to manage the desired degree of continuity”.

The tolerance parameter can and must be of some hundreds to guarantee on the finishing surface the disappearance of the “point-point” effect, without to compromise the quality of the finishing.

If the operator programs a tolerance of few microns the G63 reproduces exactly all the particulars of the original geometry.

These are the rules for the use of the E parameter:

FINISHING

Processing Typology	Tolerance Parameter E (mm.)
Blended Surfaces with more degrees of continuity CAM with tolerance of few microns that produces contiguous linear elements that forms angles in order on one degree (or less).	$E < 0.02$
Blended Surfaces in tangency CAM with tolerance in order of one hundred of mm. that produces contiguous linear elements that forms angles of some degrees.	$0.02 < E < 0.03$
CAM not particularly precise that produces contiguous linear elements that forms significant angles and of irregular entity.	$0.03 < E < 0.05$

ROUGHING OR HALF-FINISHING

Machining Allowance	Tolerance Parameter E (mm.)
0.060	0.050
0.075	0.055
0.090	0.060
0.120	0.070
0.150	0.080
0.180	0.090
0.210	0.100
0.300	0.180
0.400	0.220
0.500	0.260
0.600	0.300
0.700	0.340

In a bloc G63 without parameter E is assigned a default value E0.005

With a value E equal or Greater of 0.005 the movement is more soft but we can found rounded edges.

In case of surfaces with edges to respect to program E values lower then 0.005.

G70 G71 INCH-METRIC MODES.

RUNNING IN "WINDOWS NT" :

The visualisation can be modified with the appropriate menu (show), for necessity, in mm. or inches. (But don't modify the execution)

A change, out of respect for the value of the PLC entire constant K18, is NOT memorized.

The condition of the PLC integer constant K18, determines the CNC start mode of visualisation and running: if 0 in mm., if 1 in inches.

By programming the **G70** function it is possible to use programs in inch dimensions.
By programming the **G71** function return to use programs with metric dimensions.

G70 and **G71** are independent of the operating mode set by the value of K18.
The **G70** functions affect linear axes co-ordinates, cycles, D and F values.

To use the **P** Parameter with the value in inch it is necessary to write in the block after the value the item **i**.

Example : P10=3.78i (same as P10=96.012)

G80 DRILLING CYCLE RESET.

The G80 function cancels any drilling cycle (from G81 to G89).

G81 - G81/ DRILLING CYCLE.

It identifies the Drilling or Blading cycle with or without High Return.

Example: %N0
N1 G0 X Y Z A B PA. T..
N2 G81/.. X Y Z A B L R QZ- M.. S.... F.....z.....f.....

The CNC will execute these operations in sequence:

- 1) Traverse to position X Y A B
- 2) Traverse to L Plane
- 2a) Feedrate f to (L+z)
- 3) Feedrate F to programmed Z plane (with stops)
- 4) Stops at the bottom.
- 5) Return at traverse to R plane with spindle on.

Programming G81/ has a stop at the bottom, for a number of revolutions equal to the input value after the slash.

By programming QZ this stops is executed for all the time that the perpendicular axis does the QZ.. incremental move.

If is programmed the QZ value without the / after the G81, the plane perpendicular axis stops and immediately restart.

THE QZ VALUE MUST BE ALWAYS NEGATIVE.

After programming a G81-89 or a G96-97 function, the CNC forgets the previous programmed L value but considers the one defined in the G.. block of the fixed cycle used..

In case that, IN THE FIXED CYCLE DEFINITION BLOCK, the L dimension is omitted, the position is done on R dimension. If both R and L values are omitted the CNC considers the last programmed R-value. If this doesn't exist the traverse and the return to the plane perpendicular axis will be executed to the zero quote.

For execute a series of drilling cycle it is not necessary to rewrite the address G but, in the followed blocks, the new co-ordinates and if changed the others informations about: F, S, R, L, Z.

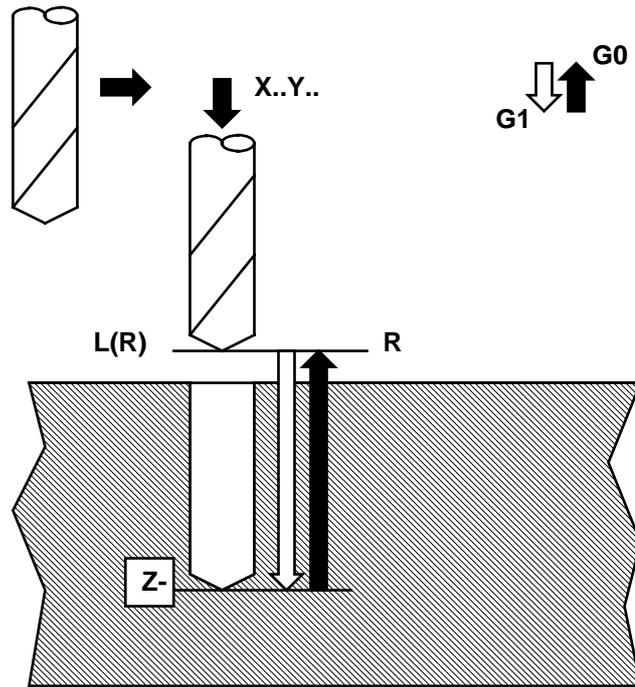
If 'z' or 'f' = 0 the point 2a) is not executed. The passage between the point 2a and the point 3 is executed without stop. The point 3 is always executed by feedrate (F).

G82 DRILLING CYCLE.

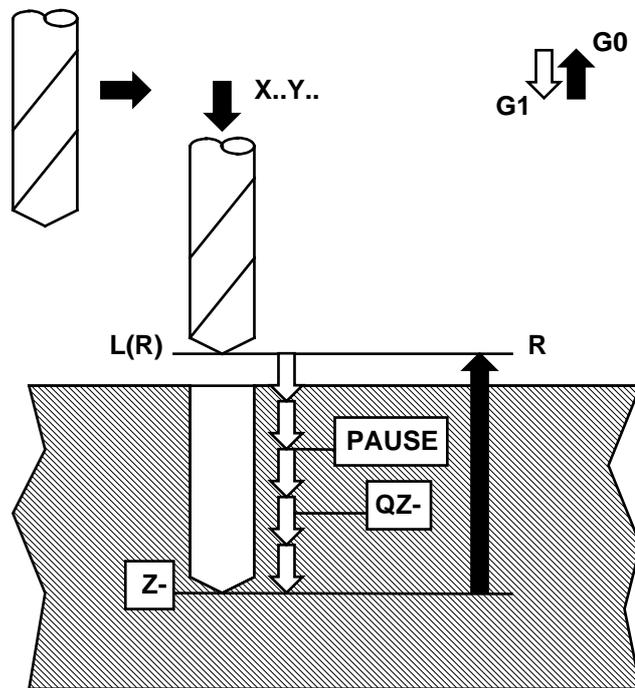
IDENTICAL AT THE G81 FUNCTION

G81 G82

e.g. G81X..Y..Z-..R..



e.g. G81/..X..Y..Z-..R..QZ-..



G83 DEEP DRILLING CYCLE.

It identifies the Deep Hole Drilling Cycle

Example: %N0

```
N1 G0 X Y Z A B PA. T..
```

```
N2 G83 X Y Z A B L R QZ- M.. S.... F..... z..... f..... r...
```

The CNC will execute these operations in sequence:

- 1) Traverse to X Y A B position.
- 2) Traverse to L.
- 2a) Feedrate f to (L+z)
- 3) Feedrate F to L+QZ.
- 4) Traverse return to the L.
- 5) Traverse to L+QZ.
- 6) Feedrate to L+QZ + (QZ-10%).
- 7) Repeat the described operations in point 4, decrementing the QZ till a max. up to 50% and reach the Z final dimension.
- 8) Traverse to the R plane with the Spindle on.

THE QZ VALUE MUST BE ALWAYS NEGATIVE.

After programming a G81-89 or a G96-97 function, the CNC forgets the previous programmed L value but considers the one defined in the block with G.. of the fixed cycle used.

In case that, IN THE FIXED CYCLE DEFINITION BLOCK, the L dimension is omitted, the position is done on R dimension. If both R and L values are omitted the CNC considers the last programmed R-value.

If this doesn't exist the traverse and the return to the plane perpendicular axis will be executed to the zero quote.

For execute a series of drilling cycle it is not necessary to rewrite the address G but, in the followed blocks, the new co-ordinates and if changed the others informations about: F, S, R, L, Z.

It is possible to program the r.. parameter that permit to return on the hole in rapid by stopping r... millimeters higher than the quotas reached previously (if not programmed r = 1/10 of QZ....).

G84 TAPPING CYCLE.

It identifies the Tapping cycle.

Example: N1 G0 X Y Z A B PA. T..
N2 G84 X Y Z A B R M.. S.... F.....QZ.....

The CNC will executed these operation in sequence:

- 1) Traverse to the X Y A B position.
- 2) Traverse to the L plane.
- 3) Feedrate to the Z plane.
- 3a) Executed in substitution of point 3 if programmed the:
Feedrate to L+QZ or to L+QZ + (QZ - 10%)
- 4) Spindle motion reverse and return at feedrate to the R plane.
- 5) Remove spindle reversal.
- 6) Traverse to the R plane.

It continue to repeat the operations defined in the points 3a 4 and 5 decrementing the QZ of the 10% each time until to a maximum of the 50%. Reach this value the QZ stays constant up to the attainment of the final quota.

In G84 cycle the value of the started F has the Drilling pitch and it is express in Micron/Revolutions ($F \cdot 1000 = 1 \text{ mm./rev.}$).

The axis in operation goes on following the programmed pitch with the number of the spindle revolutions, so we also modify the revolutions with the Override Spindle potentiometer, the Drilling does not feel the effects.

The M3/M4 function decides whether right or left spindle rotation.

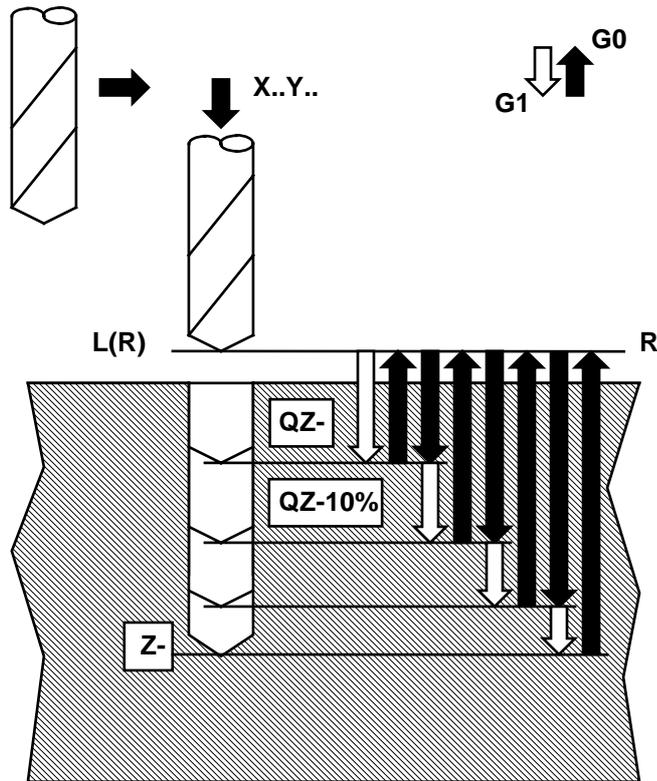
After programming a G81-89 or a G96-97 function, the CNC forgets the previous programmed L value but considers the one defined in the G.. block of the fixed cycle used.

In case that, IN THE FIXED CYCLE DEFINITION BLOCK, the L dimension is omitted, the position is done on R dimension. If both R and L values are omitted the CNC considers the last programmed R-value. If this doesn't exist the traverse and the return to the plane perpendicular axis will be executed to the zero quote.

For execute a series of drilling cycle it is not necessary to rewrite the address G but, in the followed blocks, the new co-ordinates and if changed the others informations about: F, S, R, L, Z.

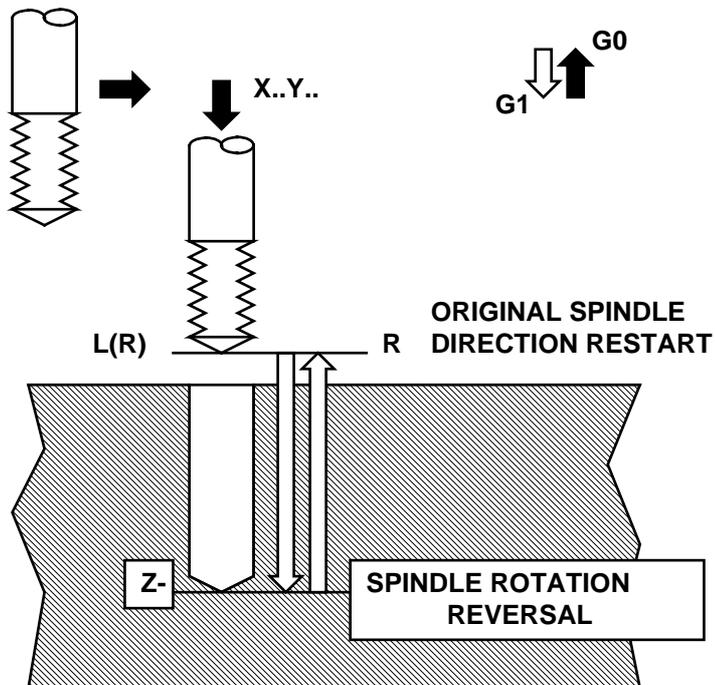
G83

E.g. G83X..Y..Z-..R..QZ-..



G84

E.g. G84X..Y..Z-..R..F..



G85 DEEP HOLE DRILLING CYCLE.

It identifies the Deep Hole Drilling Cycle

Example: %N0
N1 G0 X Y Z A B PA. T..
N2 G85 X Y Z A B L R QZ- M.. S.... F.....z....f....

The CNC will execute these operations in sequence:

- 1) Traverse to X Y A B position
- 2) Traverse to L
- 2a) Feedrate f to (L+z)
- 3) Feedrate F to L+QZ
- 4) Traverse return to the R plane
- 5) Traverse to L+QZ
- 6) Feedrate to L+QZ + (QZ-10%)
- 7) Repeat the described operations in point 4, decrementing the QZ till a max. up to 50% and reach the Z final dimension
- 8) Traverse to the R plane with the Spindle on

THE QZ VALUE MUST BE ALWAYS NEGATIVE.

After programming a G81-89 or a G96-97 function, the CNC forgets the previous programmed L value but considers the one defined in the block with G.. of the fixed cycle used..

In case that, IN THE FIXED CYCLE DEFINITION BLOCK, the L dimension is omitted, the position is done on R dimension. If both R and L values are omitted the CNC considers the last programmed R-value. If this doesn't exist the traverse and the return to the plane perpendicular axis will be executed to the zero quote.

To execute a series of drilling cycle it is not necessary to rewrite the address G but, in the followed blocks, the new co-ordinates and if changed the others informations about: F, S, R, L, and Z.

G86 BORING CYCLE

It identifies the Boring Cycle.

Example: %N0
 N1 G0 X Y Z A B PA. T..
 N2 G86/ X Y Z A B R L M.. S.... F.....

The CNC will execute these operations in sequence:

- 1) Traverse to position X Y A B
- 2) Traverse to L Plane
- 3) Feedrate to programmed Z plane
- 4) Stops at the bottom (by programming G86/)
- 5) Spindle stops and return at traverse to R plane
- 6) Spindle rotation on

Programming G86/ has a stop at the bottom, for a number of revolutions equal to the input value after the slash.

By programming QZ this stops is executed for all the time that the perpendicular axis does the QZ.. incremental move.

If is programmed the QZ value without the / after the G81, the plane perpendicular axis stops and immediately restart.

THE QZ VALUE MUST BE ALWAYS NEGATIVE.

After programming a G81-89 or a G96-97 function, the CNC forgets the previous programmed L value but considers the one defined in the block with G.. of the fixed cycle used.

In case that, IN THE FIXED CYCLE DEFINITION BLOCK, the L dimension is omitted, the position is done on R dimension. If both R and L values are omitted the CNC considers the last programmed R-value. If this doesn't exist the traverse and the return to the plane perpendicular axis will be executed to the zero quote.

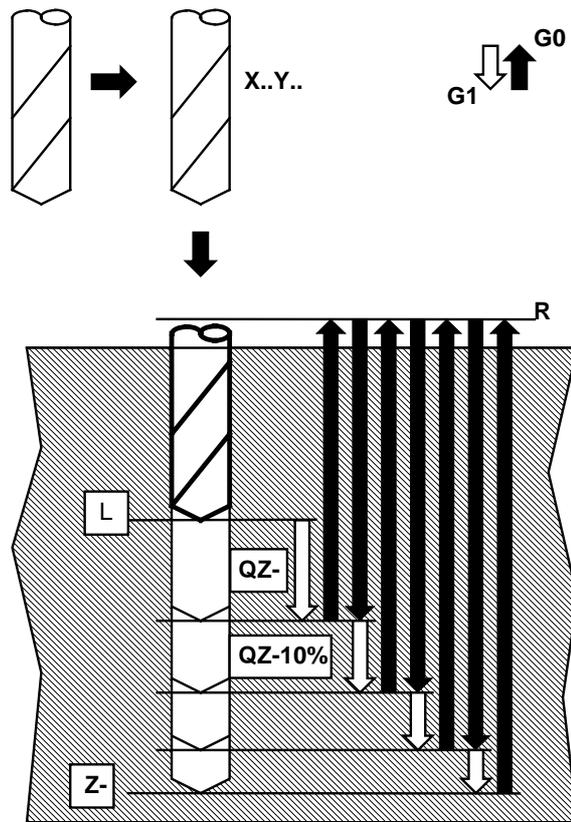
To execute a series of drilling cycle it is not necessary to rewrite the address G but, in the followed blocks, the new co-ordinates and if changed the others informations about: F, S, R, L, and Z.

G87 BORING CYCLE

IDENTICAL AT THE G86 FUNCTION

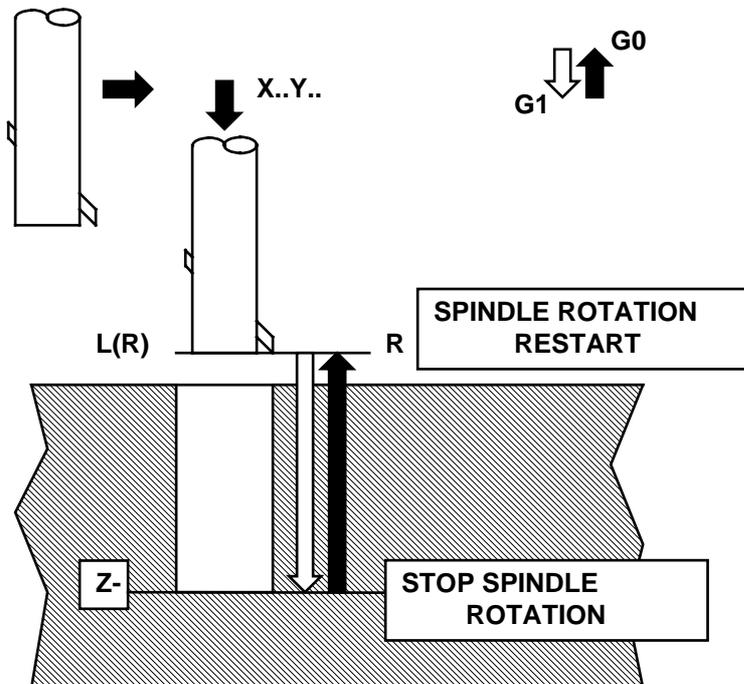
G85

E.g. G85X..Y..Z-..R..L..QZ-..



G86 G87

E.g. G86X..Y..Z-..R..



G88 DRILLING CYCLE FOR SPACED WALLS

It identifies the Drilling Cycle for Spaced Walls

Example : %N0
N1 G0X Y Z A B PA. T..
N2 G88/ X Y Z A B R z f
N3 Z' R'
N4 Z" R"
N5 G0Z
N6

The CNC will execute these operations in sequence:

- 1) Traverse to X Y A B in the plane.
- 2) Traverse to the R plane.
- 2a) Feedrate f to (L+z)
- 3) Feedrate F up to the Z plane.
- 4) Eventual stop if G88/ is programmed N3 block:
- 5) Traverse to the R' plane.
- 5a) Feedrate f to (L+z)
- 6) Feedrate F to the Z' plane N4 block:
- 7) Traverse to the R" plane.
- 7a) Feedrate f to (L+z)
- 8) Feedrate F to the L" plane.

In the N3 and N4 blocks are programmed only the Z and R-values.

By programming G88/ the machine has a stop at the final feedrate quote, for a number of revolutions equal to the input value after the slash, and after executed the successive block.

To exit from the piece it is necessary to program an exit block.

After programming a G81-89 or a G96-97 function, the CNC forgets the previous programmed L value but considers the one defined in the block with G.. of the fixed cycle used..

In case that, IN THE FIXED CYCLE DEFINITION BLOCK, the L dimension is omitted, the position is done on R dimension. If both R and L values are omitted the CNC considers the last programmed R-value. If this doesn't exist the traverse and the return to the plane perpendicular axis will be executed to the zero quote.

For execute a series of drilling cycle it is not necessary to rewrite the address G but, in the followed blocks, the new co-ordinates and if changed the others informations about: F, S, R, L, Z.

G89 REVERSE BORING CYCLE

It identifies the Reverse Boring Cycle

Example: %N0
N1 G0 X Y Z A B PA. T..
N2 G89/ X Y A B Z- R L M.. S.... F.....

The CNC will execute these operations in sequence:

- 1) Traverse to position X Y A B.
- 2) Traverse to the L Plane.
- 3) Feedrate to programmed Z plane.
- 4) Stops if programmed G896/.
- 5) Feedrate to the L plane.
- 6) Traverse to the R plane.

By programming G88/ the machine has a stop at the final feedrate quote, for a number of revolutions equal to the input value after the slash, and after traverse to the R plane.

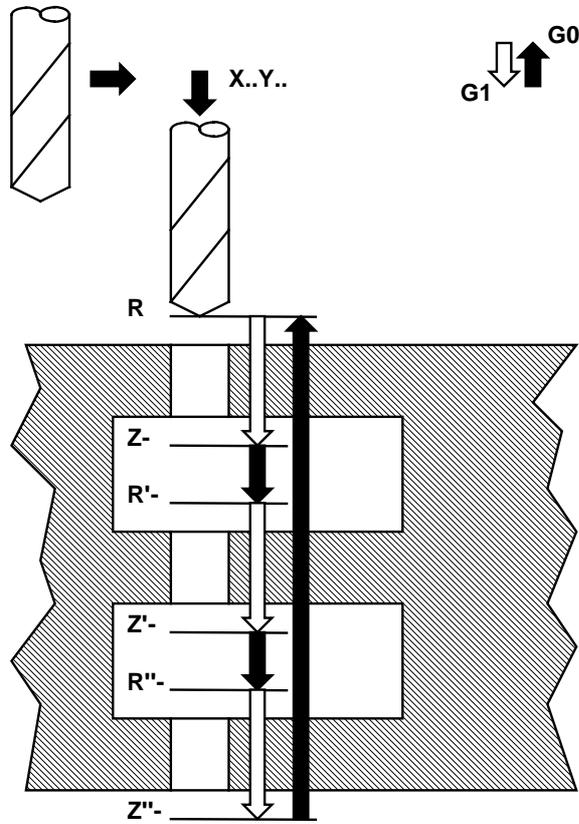
After programming a G81-89 or a G96-97 function, the CN forgets the previous programmed L value but considers the one defined in the block with G.. of the fixed cycle used.

In case that, IN THE FIXED CYCLE DEFINITION BLOCK, the L dimension is omitted, the position is done on R dimension. If both R and L values are omitted the CN considers the last programmed R-value. If this doesn't exist the traverse and the return to the plane perpendicular axis will be executed to the zero quote.

To execute a series of drilling cycle it is not necessary to rewrite the address G but, in the followed blocks, the new co-ordinates and if changed the others informations about: F, S, R, L, and Z.

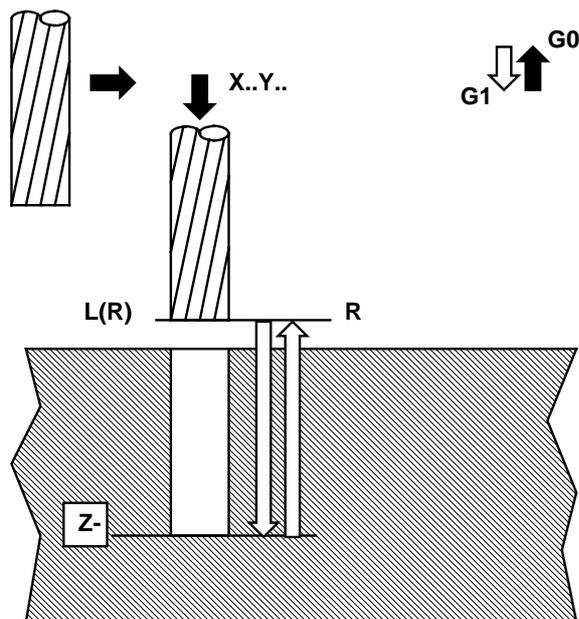
G88

E.g. G88X..Y..Z..R..
Z'..R'..
Z''..R''..
G0Z2



G89

E.g. G89X..Y..Z..R..
L(R)



G94 FEEDRATE IN mm/min

The G94 function identifies the Tool Machine feedrate in mm/min. that is the natural way that the Numerical Control has still from the beginning.

G95 FEEDRATE IN micron/rev

The G95 function identifies the Tool Machine feedrate in micron/revolutions, that can be useful in borings.

G96 BORING CYCLE

This instruction identifies the macroinstruction for boring with detachment from the wall.

Example: N10 GO X Y Z A B PA. T..
N20 G96/ X Y Z A B E R L M.. S.... F.....

The E function is the spindle key angle respect the X+ axis.

It is POSITIVE the angle that the quill key has to execute, in counter clockwise way, starting from the X+ axis; NEGATIVE, the angle executed in clockwise way.

By mounting the tool with the same orientation of the spindle key at the end of the cycle the tool will executes a detachment from the wall and after will traverse to the R plane.

The CN will execute these operations in the following sequence:

- 1) Traverse to X Y A B position
- 2) Traverse to L
- 3) Feedrate to the Z plane
- 4) Stops if programmed G96/
- 5) Spindle stops and spindle key positioning
- 6) Moving the X Y axes about 2 tenths of millimetre in direction defined with the E angle value.
- 7) Traverse to the R plane
- 8) Traverse to the number 1) X Y plane.
- 9) Spindle rotation on.

By programming G96/ the machine has a stop at the final feedrate quote, for a number of revolutions equal to the input value after the slash, and after traverse to the R plane.

After programming a G81-89 or a G96-97 function, the CN forgets the previous programmed L value but considers the one defined in the block with G.. of the fixed cycle used..

In case that, IN THE FIXED CYCLE DEFINITION BLOCK, the L dimension is omitted, the position is done on R dimension. If both R and L values are omitted the CN considers the last programmed R-value. If this doesn't exist the traverse and the return to the plane perpendicular axis will be executed to the zero quote.

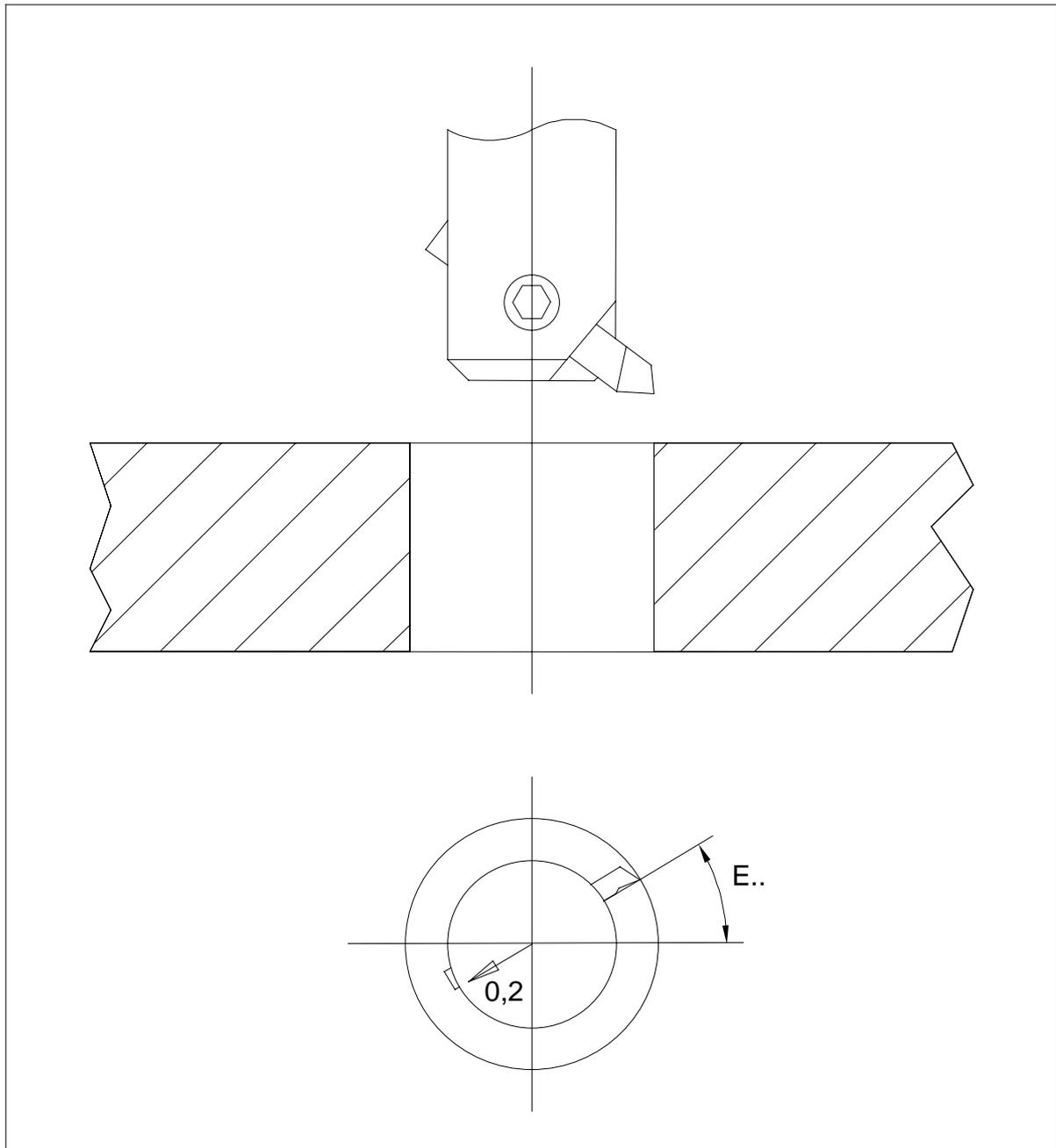
The E angle must be always equal of the spindle key angle and also the tool. For the Machine equipped with automatic tool change or with a Multiplexed rotary head this angle is 225 degrees.

In the others cases this angle must be measured.

If the E angle is omitted the CN take the ZERO value.

To execute a series of fixed cycle it is not necessary to rewrite the address G but, in the followed blocks, the new co-ordinates and if changed the others informations about: F, S, R, L, and Z.

G96



Example of boring cycle:

```
%N0  
N1 G45T1  
N2 G96 X0 Y0 Z-20 R2 E30 F400S2000M13PA1  
N3 X100  
N4 G47  
N9999%
```

G97 MILLING A BORE AND MILLING A THREAD CYCLES

This instruction identifies the macroinstruction for two operations: Milling a Bore and also milling a Thread.

The CN recognizes the one from the other function by verifying the existence of the QX parameter in the G97 block:

- If in the G97 block is programmed a QX different from zero or if the previous QX has not be deleted by programming the QX0, the CN will recognize the Milling a Bore cycle.
- If in the block is programmed a QX= or if don't exist previous QX, the CN will recognize the Milling a Thread cycle.

Milling a Bore cycle

example : N10 G0 X Y Z A B PA. T..
 N20 G97 X Y Z A B R L QZ QX D M.. S F

Where:

QZ = Distance raised from the bottom.

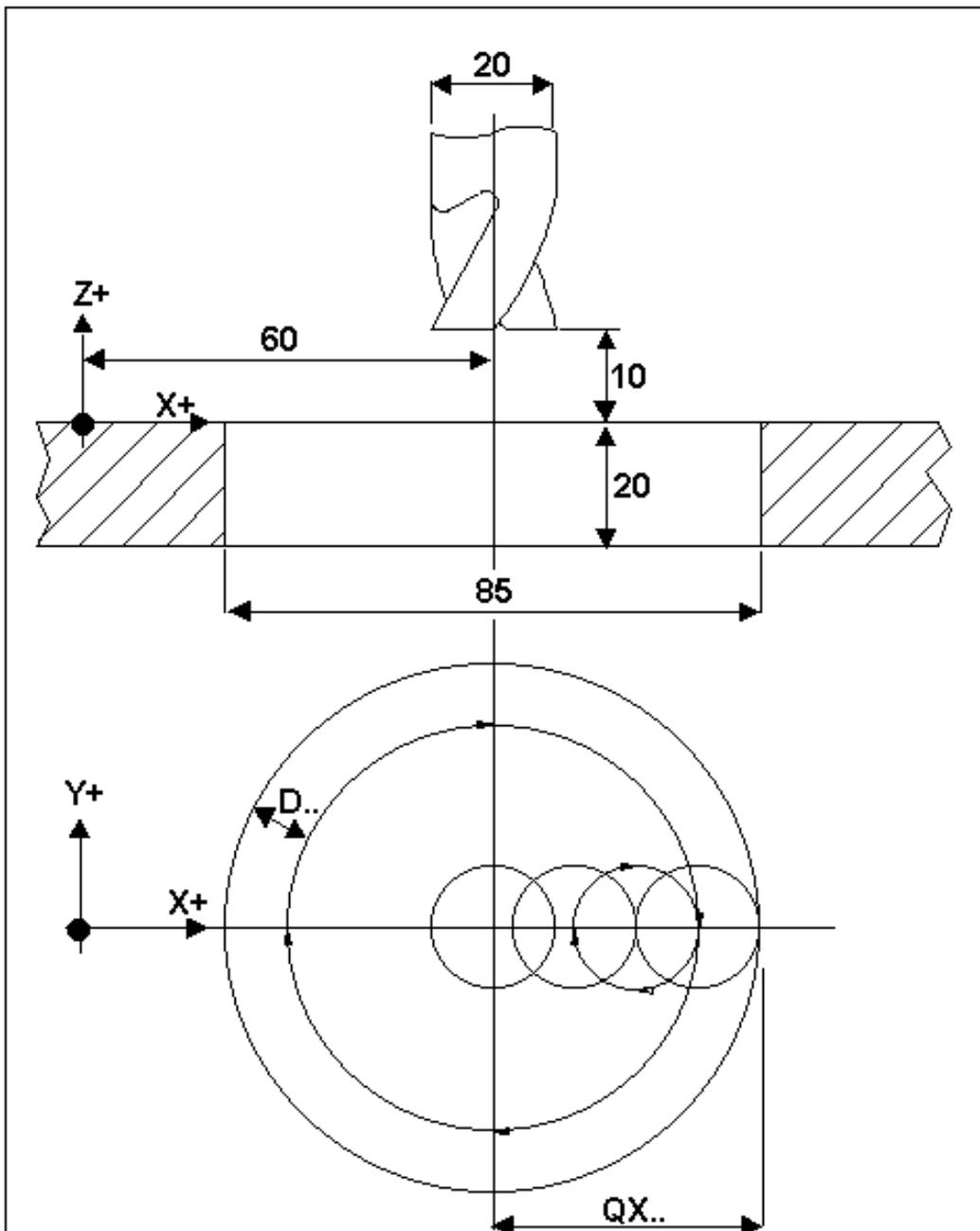
QX = Radius of the outer Bore.

D = Tool Radius.

The CNC will execute these operations in the following sequence...

- 1) Traverse to X Y A B position.
- 2) Traverse to L.
- 3) Feedrate to the Z plane.
- 4) Reverse traverse to Z + QZ position.
- 5) a) In a case when $QX < D$, the machine will 'Cut-Out' and error 13 will appear.
b) In a case when $QX < 2D$, it will join the radius at $(QX-D)/2$
c) In a case when $QX > 2D$, it will execute a linear approach to a point X+
 $(QX-D)$ and then execute a radius with the value $(QX-D)/2$.
- 6) Clockwise circular interpolation with a centre X Y and with radius QX-D
- 7) Detachment from the workpiece with a radius $(QX-D)/2$ and return to the point X Y in linear interpolation.
- 8) Traverse to R plane.

G97 Milling a Bore



Example of milling a bore:

```
%N0  
N1 G45T1  
N2 G97 X60 Y0 Z-23 L2 R10 QX42.5 D10 F400S2000M13PA1  
N3 X-60  
N4 G47  
N9999%
```

Milling a Thread cycle

Example: N10 G0 X Y Z A B PA. T..
 N20 G97 X Y Z A B R L QZ E K W M.. S... F....

Where:

QZ+- =Distance raised from the bottom.

E +- = Radius path of centre tool. The sign indicates the thread direction:

 + = Counter clockwise, - = Clockwise.

K +- = Pitch of thread.

W +- = Pitch of helical approach to the workpiece.

The CNC will execute these operations in the following sequence:

- 1) Traverse to the X Y position
- 2) Traverse to L
- 3) Feedrate to the Z plane
- 4) Reverse traverse to Z + QZ plane.
- 5) Approach to the workpiece with helical interpolation of radius E/2 and Pitch W, to the point X+E. The start position of the thread in Z-axis is then Z+QZ+W/2.
- 6) Helical interpolation, Clockwise (E+) or Counter clockwise (E-) with centre X Y, radius E, and pitch K.
- 7) Detachment from the workpiece with helical interpolation radius E/2 and pitch W to the point X+E.
- 8) Return to the R plane

After programming a G81-89 or a G96-97 function, the CN forgets the previous programmed L value but considers the one defined in the G block of the fixed cycle used.

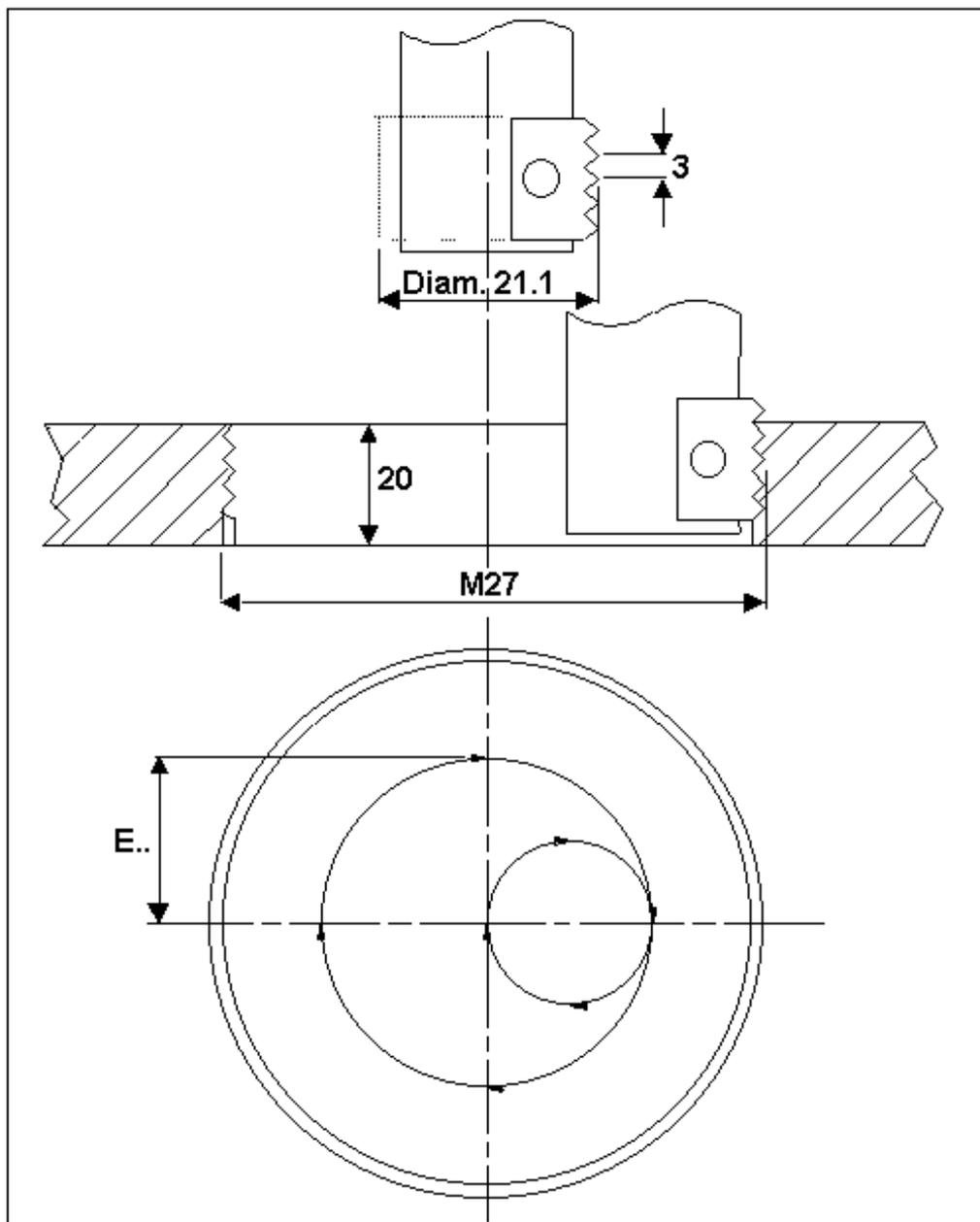
In case that, IN THE FIXED CYCLE DEFINITION BLOCK, the L dimension is omitted, the position is done on R dimension. If both R and L values are omitted the CN considers the last programmed R-value.

If this doesn't exist the traverse and the return to the plane perpendicular axis will be executed to the zero quote.

If activated the G341 function the D value is ignored and the CN consider the radius value imposed in the detailed table.

For execute a series of fixed cycle it is not necessary to rewrite the address G but, in the followed blocks, the new co-ordinates and if changed the others informations about: F, S, R, L, Z.

G97 Milling a Thread



Example of milling a thread:

```
%N0  
N1 G45T1  
N2 G99 P1=13.5-(21.1/2)  
N3 G97X-100 Y0 Z-21 R2 QZ0 EP1 K3 W1.5 F400S2000M13PA1  
N4 X100  
N5 G47  
N9999%
```

G99 AXES STOP

It is used for input M functions or messages.
This function delete the previously G function.
Don't insert this function inside a contouring.

For Manual Tool Change, with the " M6 T "

Example : %N0
 N10 G99M6T1

%N0

N1 G99 (TSPP_E500)
N2 G45T1 (ROUGH-MILL D. 18)
N3 G99 (TSPP_E500, DIAM18)
N4 F266S1125PA1
N5 G0X70Y0
N6 M13
N7 G0Z2
N8 G1Z-2.5
N9 G41X60Y-20D9.5
N10 G1Y-40
N11 G3X70Y-50I70J-40
N12 G1X80
N13 G3X90Y-40I80J-40
N14 G1Y-20
N15 X100
N16 G3X125Y-45I125J-20
N17 G1Y-90
N18 G2X100Y-115I100J-90
N19 G1X50
N20 G2X25Y-90I50J-90
N21 G1Y-45
N22 G3X50Y-20I25J-20
N23 G1X60
N24 H1N8N23QZ-2.5
N25 G0Z2
N26 G0X75Y-82.5
N27 G1Z-2.5
N28 G41X75Y-105D9.5
N29 G1X96.236
N30 G3X104.896Y-100I96.236J-95
N31 G1X112.113Y-87.5
N32 G3Y-77.5I103.453J-82.5
N33 G1X104.896Y-65
N34 G3X96.236Y-60I96.236J-70
N35 G1X53.764
N36 G3X45.104Y-65I53.764J-70
N37 G1X37.887Y-77.5
N38 G3Y-87.5I46.547J-82.5
N39 G1X45.104Y-100
N40 G3X53.764Y-105I53.764J-95
N41 G1X75
N42 H1N27N41QZ-2.5
N43 G47
N44 G45T2(FINISH-MILL D. 18)
N45 G99(TSPP_E500, DIAM18)
N46 F230S1200PA1
N47 H1N5N42QD-.5
N48 G47
N49 G45T3(CENTER-DRILL D. 10)
N50 G99 (TSPP_E500, DIAM10)
N51 F128S787PA1
N52 G0X90Y-82.5
N53 G81Z-6.5R-3M13
N54 X75Y-67.5
N55 X60Y-82.5
N56 X75Y-97.5
N57 G47
N58 G45T4(TWIST-DRILL D.6.75)
N59 G99(TSPP_E500, DIAM6.75)
N60 F185S1414PA1
N61 G0X90Y-82.5
N62 G83Z-21.528R-3QZ-6.75M13
N63 X75Y-67.5
N64 X60Y-82.5
N65 X75Y-97.5
N66 G47
N67 G45T5(SCREW-TAP ISO D. 8)
N68 G99 (TSPP_E500, DIAM8)
N69 F1250S258PA1
N70 G0X90Y-82.5
N71 G84Z-18R-3L2M13
N72 X75Y-67.5
N73 X60Y-82.5
N74 X75Y-97.5
N75 G47
N99999%

Gxxx MACRO (Subroutine)

A Macro (Subroutine) is an ISO part-program that must start WITHOUT the character '%' and must FINISH with the function **G59** (The main program start and finish with the character '%').

The Macro is called with the Gnumber function from the main program or from another subroutine. (The number can go till 999; but the user must use from 101 to 299). When the number is the same as a CNC existing function this replace the old G function. The Gnumber function gives the name to the file that must be charged in the directory C:\MACRO. The Gnumber function must be write on the list of the file GIUTEN.CNC in the directory C:\MACRO.

Part-Program

(FILE AAA)

```

%N0          (FILE Gxxx)
N1
N2 Gxxx----->      N10          (FILE Gyyy)
N3      <--          N20
N4          |          N30 Gyyy ----->      N100
N5          |          N40          <--          N200          (FILE Gzzz)
N6000%       |          N50          |          N300
              |          N60          |          N400 Gzzz ----->      N1000
              <--      N70 G59       |          N500          <--          N2000
                                   <--      N600 G59       |          N3000
                                                         |          N4000
                                                         <--      N5000 G59
    
```

On the macros the programming rules are the same as the rules on the main part-program. The CNC dispose of a memory about 256 Kbytes for the macro. A routine H must have start and end in the same file. The routine levels are accumulated for each macro.

On the main part-program it is possible to write only the parameters from P0 to P99, on the macros only from P100 to P199.

The main program and the macros can read all the parameters.

Program with call of a macro:

Main Program

```
%N0
N1 G45T1
N2 G99 PA1 F3000 S500 M13
    P1=0
    P2=0
    P3=15
    P4=20
    P5=12
    P6=10
    P7=-2.5
    P8=15
    P9=1
N3 G197
N4 G47
N99999%
```

Macro G197

```
N197001 G99(MACRO FOR MILLING AN EXTERNAL THREAD)
( P1= Centre in X
( P2= Centre in Y
( P3= Inside diameter
( P4= Outside diameter
( P5= Safety Z
( P6= Final Z
( P7= Step +/-
( P8= Tool Radius
( P9= Number of helix
N197002 P100=P3/2
N197003 P101=P4/2
N197004 P102=P101-P100+P8+1
N197005 P103=P100+P102
N197006 P104=P100/P102
N197007 P105= - P102
N197008 P107=P7/P104
N197009 P108=P107/4
N197010 P109=P9
N197011 P106=P6-P108-P7
N197012 G51XP1YP2
N197013 G0XP103Y0
N197014 ZP5
N197015 G1ZP106
N197016 G41XP103YP102DP8
N197017 G3XP100Y0IP103J0KP107
N197018 G2I0J0KP7
N197019 P109=P109-1
N197020 P109>0 N197018
N197021 G3XP103YP105IP103J0KP107
N197022 G40Y0
N197023 G0ZP5
N197024 G59
```

FUNCTIONS FOR TOOLS PRESETTING

TOOL LENGTH AND TOOL RADIUS SETTING PROCEDURE

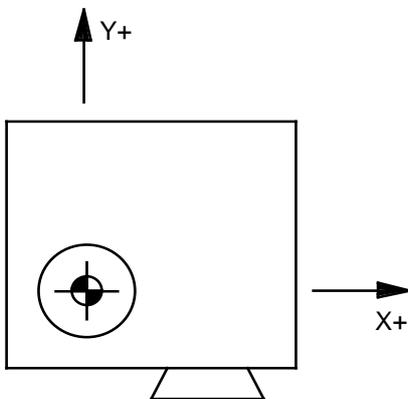
Zero setting device instrument positioning.

This instrument must be mounted in a Tool Machine X, Y and Z accessible axes position.

The head of this instrument is used to set and to verify the tools lengths; the other head in the lateral position is used to set and to verify the tools diameters.

The position of this instrument must be indicated on SHIFT-F9 (see the following picture): the X, Y quotes are referred to the centre of the zero setting head, while the Z quote is positioned near this one (~2 mm over).

To zero setting this instrument (during the installation or if this instrument has been moved) it is necessary to move to the centre in X Y and in Z ~2 mm between the spindle key and the head for the length zero setting and:



Find the length key centre

Press **SHIFT+F9 (OFFSET)**

Press **F4 (AZZER)**

By using $\uparrow\downarrow$ select **QUOTE INPUT X** axis

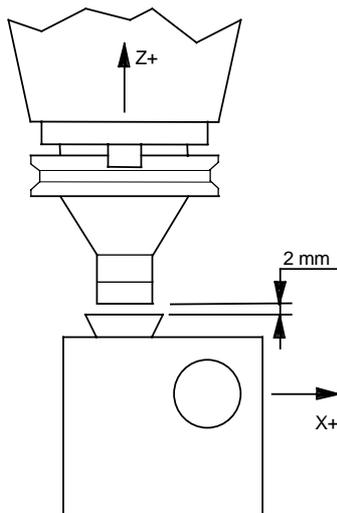
Press **ENTER**

Press **ENTER**

By using \Rightarrow select **QUOTE INPUT Y** axis

Press **ENTER**

Press **ENTER**



Set the Master-tool ~ 2 mm over the length key

By using \Rightarrow select **QUOTE INPUT Z** axis

Press **ENTER**

Press **ENTER**

Press **F12** to exit

Measure cycle and tool verify description:

These cycles are realized in special subprogram that can be recalled in the part-program by using some "G" functions:

- G300** - Tool length measure with table adjournment.
- G301** - Tool length verify with programmed stops (M1) if the verified tool length is out of tolerance.
- G302** - Tool length verify with automatical research of the tool with equal class (if different from 0) when the verified tool length is out of tolerance. If the tool length is on tolerance the CN adjourn the table.
- G303** - Tool radius measure with table adjournment.
- G304** - Tool radius verify with programmed stop (M1) if the verified tool radius is out of tolerance.
- G305** - Tool radius verify with automatical research of the tool with equal class (if different from 0) when verified tool radius is out of tolerance. If the tool radius is on tolerance the CN adjourn the table.
- G306** - Tool length and radius measure with table adjournment.

All cycles have the acquisition of sample quotas to measure and verify the tools.

For the tool lengths the sample quota is done on the spindle key.

For the tool radius the sample quota is done with the sample tool (T 16) that have the radius dimension write in configuration (K 17 constant in micron). Each measure of length and radius is done two times consequently; the tolerance between the two measures is written in configuration (K 15 float constant).

With the K16 float constant the tool-holder cone to zero setting the spindle key is defined and is expressed in millimetres.

The tolerance to verify the tools is written in the calling subprogram part program after "/" character and is expressed in millimetres with the point "." to separate integers from decimals.

It is usable with all verify cycles. When "/" character is not programmed or when its value is zero, the verify cycle become a normal measure cycle.

The qualification of this instrument is done with these functions:

- G377** Setting of the instrument for the tools length.
Move the machine manually with the special tool over the instrument for the tools length and write G377F1000.
- G378** Setting of the instrument for the tools radius.
Move the machine manually with the special tool near the instrument for the tools radius and write G378F1000.

- Tool table disposition for the automatical tool managing:

- CIs** - number from 0 to 255 to indicate the same tools in the holder plate.
When this number is = 0 tools are managed in the traditional way and any automatical tool research is allowed.
When this number is different from 0 this means:
same numbers = same tools type.
This makes the automatical substitution of a non-usable tool to another with the same number.
- @** - Defines the tool work rotation way:
If - the work rotation way is clockwise, if + counter clockwise.
- L** - Tool length: can be edited or the value obtained by the automatical zero setting is kept.
- DL** - Delta of the tool length: can be edited and its value is summed to the l value.
Every time the tool is automatically measured this value became 0.
- r** - The same meaning like l for the tool radius.
- Dr** - The same meaning like Dl for the tool radius.
- ts** - Tool lifetime in seconds
- ts** - Processing number decrease by the M387 function, if the N bit of status is active
- status** - Tool status informations:
 - L** Is written for all the times the tool length measure is done using the automatical cycle.
 - D** Is written for all the times the tool radius measure is done using the automatical cycle.
 - T** Tool Life time. With **ts** and **cl** activate the check of tool lifetime.
 - N** If programmed, alternative with the T, takes the Ts value as a pure number.
 - X** Non usable tool: is written for all the times the tool has not passed the verified cycle (length or radius) and the tool became non-usable, if the cl value is different from zero the automatical search is enabled.

All this values are editable also from the keyboard.

TOOL LIFETIME MANAGEMENT

This check is activated only when in the tools table the CL is different from 0 and it is present the bit " T " on the STATUS.

In the table this values can be written in different ways:

- a) By editing the table in manual and writing the time in minutes.
- b) In the block that call a tool measure function (G300, G303, G306) writing the letter " t " (small ti) with the lifetime in minutes.
- c) The tool dimensions can be transferred from the part-program to the table in a block as the following:

```
N100 G99 M300 M364 {T12C10L100DI0.5R7DR0.05t8500(-LDT)"Tool Description"}
```

The values written in the parenthesis { } except what is inside the " ", are transferred in the tools table. Each tool has the same form as the .UTI files generated after a tools table save:

- T - Tool number: whole number from 1 to 199.
- C - Tool classes: whole number from 0 to 99.
- L - Tool length: in mm with the point that separate the decimals.
- DL - Tool Delta Length: in mm with the point that separate the decimals.
- R - Tool radius: in mm with the point that separate the decimals.
- DR - Tool Delta Radius: in mm with the point that separate the decimals.
- t - Tool work time: in seconds from 0 to 99999.
- (-LDT) Where:if - tool work clockwise, if + counter clockwise.
 - LD flag of the tool status.
 - T if present active the check of the tool lifetime.
- " " - Tool description max. 20 characters.

It is not necessary to write all this parameters but only the ones that you need.

The M300 function write in the bloc is used to adjourn the tools file.

The tool working time is kept in a partial counter that is adjourned all the time that the tool is working and simultaneously is verified that the tool working time is not bigger than the one inserted on the table.

When the tool finishes the established work time it is replaced with a tool with the same class then continue the execution. In the table the substituted tool will have the bit " X " and its work time is annulled.

The execution restart with an automatic repositioning that with the following order:

- 1) Spindle rotation restart.
- 2) Tilting head repositioning.
- 3) X and Y-axes repositioning.
- 4) Z axis repositioning.

When the tool is substituted before to have finish its work time the table is adjourned with the remaining time.

The table is adjourned with the worked time also if the CNC is resetted by pushing <Shift - F11> without to have release the tool in the magazine.

If the CNC is switching off without the reset procedure the remaining tool time is NOT adjourned on the table.

This group of M functions permits the management from a program of the tool substitution that has finished its lifetime.

If this mode is active the tool substitution is executed only from the program

In this way we avoid the milling interruptions with the tool in contact with the piece or during the phases of milling where it is not advisable to substitute the tool.

This function must be programmed at the end of positioning (G0 rapid movements) or in lines of G99.

The tool lifetime is activated on the machine for all the tools on the cone and that have the followings parameters on the tools table:

- a) A value different from 0 defined under the voice 'cl' (class).
- b) The parameter 'T' (time) under the voice 'status'.
- c) A time value (in seconds) different from 0 under the voice 'τs'.
- d) The parameter 'X' under the voice 'status' means that the tool is useless.

If a processing doesn't finish its defined lifetime the table is adjourned with the residual value every time the tools is released on the tools magazine.

The **M1316** function active the programmed substitution mode.

The **M1315** function (default condition) cancels the programmed substitution mode.

The **M1317** function executes immediately the substitution of the tool only if its lifetime is finished.

The function M1317 is ignored if the tool lifetime on the cone is not finished.

The function M1317 is ignored when the tool with the lifetime finished is release from the cone with a function (G47, M06).

E.g.:

```

%N0
N10 G99 M1316
N30
N40 G349[prg1]
N50 G349[prg2]
N60 G349[prg3]
N70 G349[prg4]
N80 G99
N9999%

[prg1] %N0
N10 G45T1
.....
N1000 G47
N9999%

[prg2] %N0
N10 G45T2
N20
.....
N1000 G0 Z100
N9999%

[prg3] %N0
N10 G99 T2
N20 G99 M1317
.....
N1000 G0 Z100
N9999%

[prg4] %N0
N10 G99 T2
N20G99 M1317
.....
N1000 G47
N9999%
```

MEASURE AND VERIFY CYCLES PROGRAMMING

On the programming line can be programmed with the G function the followed functions:

M301{T1,T2,T3} must be programmed for every measure cycles to indicate the tools to measure. The tools are measured with the sequence written between { }. All the cycles must start without the tool on the cone.
When {T1 + T5} is programmed, T1 to T5 tools are measured.
When {+ T7} is programmed, T1 to T7 tools are measured.
The new tools measure is memorized on **C:\CONF_DIR\TOOLS.WRK**

/ Threshold of tolerance for the verify cycles, in mm.
The measure obtained is compared with the one written on the table and replace that one if the difference between those two don't go beyond the value written after the /. If the / character is not programmed the verify cycle become a measure cycle and adjourn the value on the table.
On the manual cycles (G503 and G506) the /1 value establish that the callipers must be insert manually.

Z Z-axis quotas for the manual tool change.

E is the spindle key orientation.

t is the tool lifetime in seconds, works only on the measure cycles.

D The "D1" value programmed disables the tool automatic replacing that's gone out from the tolerance in the G302 and G305 cycles.

Ex: N100 G302/0.02 D1
N110 G47

If the tool isn't in tolerance is vain to replace-it by a new one that has got to be returned in the magazine immediately.

Q1 With the "Q1" programmed value is possible to execute the measure cycle or tool verify without the sample tool measure considering good the last taken. The tool measure obtained will result less precise than that one executed measuring every time with the sample tool.

QE Is the tool orientation angle before the approach to the instrument.

L1 If programmed don't adjourn the tool dimension on the table after a verify cycle that is resulted on tolerance.

QZ To define an increment on Z for the tool radius measure.

R To define an increment on the radius for the tool length measure.

G300 TOOL LENGTH MEASUREMENT

The tool length automatical zero setting is realized with G 300 program and can be recalled by others programs.

Example: N10 **G300 M301 {T1,T2,T6,T8} E.. t..QE..**

- Measure the T1, T2, T6, T8, tools length and store it on disk
C:\CONF_DIR\TOOLS.WRK;
Tools are measured following the sequence indicated between the {} parenthesis.
No tools have to be in the cone to begun a tool length measuring cycle.
The spindle zero setting is done without tool before measuring the tool length.
Parameters P200, P201 and P202 are used:

P200 is used for tool number to zero set.
P201 recall the tool code in the zero setting sequence.
P202 recall the result of the confrontation between two measuring cycles
(if=1 it means OK).
M308 function decrease of 1 P200 and write in P201 the successive tool number.
When {T1 + T5} is programmed, T1 to T5 tools are measured.
When {+ T7} is programmed, T1 to T7 tools are measured.

Before starting a zero set tool cycle, it is necessary to insert in the tools table a higher value of the effective length to avoid to damage the instrument.

G301 TOOL LENGTH VERIFY WITH STOP IF OUT OF TOLERANCE

The tool length verify and zero setting cycles is realized with G 300 program and can be recalled by others programs.

Example: N10 **G301/0.01 E..QE..**

- Execute a verification cycle of the actual length of the mounted tool; the actual length is compared with the one written on the table and substituted to this one if the difference between them don't exceed the value written after the / character.

If the difference between them exceed the imposed value, the P201 parameter take the 0 value.

If the / character is not programmed the actual length is re adjourned in the table.

Parameters P201, P202 and P203 are used.

If the G301 is programmed in the middle of the program it is important to re-program the M function (M3 or M..) and the spindle speed S ... in the next block of the program to restart the spindle.

Example : G301/0.02
 G0X...Y... **S1500 M13**

G302 TOOL LENGTH VERIFY WITH SUBSTITUTION IF OUT OF TOLERANCE.

The tool length verify and zero setting cycles is realized with G 302 program and can be recalled by others programs.

Example: N10 **G302/0.01 E.. D1 QE..**

- Execute a verification cycle of the actual length of the mounted tool; the actual length is compared with the one written on the table and substituted to this one if the difference between them don't exceed the value written after the / character. If the difference between them exceed the imposed value, the P201 parameter take the 0 value, now its searched in the table a similar tool to substitute the one out of tolerance remaking the verifying.

If the / character is not programmed the actual length is re adjourned in the table.

Parameters P189, P201, P202, P203 and P208 are used.

If the G302 is programmed in the middle of the program it is important to re-program the M function (M3 or M..) and the spindle speed S ... in the next block of the program to restart the spindle.

Example : G302/0.02
 G0X...Y... **S1500 M13**

G303 TOOL RADIUS MEASUREMENT

The automatical tool radius zero setting cycle is realized with G 303 program and can be recalled by others programs.

Example: N10 **G303 M301 {T1,T2,T6,T8} QZ+.. t.. E.. Q1 QE..**

- Measure T1, T2, T6, T8 Tools radius and store them on disk in C:\CONF_DIR\TOOLS.WRK

Tools are measured following the sequence indicated between the {} parenthesis.

To begin a radius tool measuring cycle it is necessary to use a tool with known radius.

In the G303 subroutine it correspond to the T16 position. The parameters P200, P201, P202 are used:

P200 is used for tool number to measure.

P201 recall the tool code in the measuring sequence.

P202 recall the result of the confrontation between two measuring cycles (if=1 means OK).

M308 function decrease of 1 P200 and write in P201 the successive tool number

When {T1 + T5} is programmed, T1 to T5 tools are measured.

When {+ T7} is programmed, T1 to T7 tools are measured.

Before starting a zero set tool cycle, it is necessary to insert in the tools table a BIGGER value of the effective RADIUS to avoid to damage the instrument.

G304 TOOL RADIUS VERIFY CYCLE WITH STOP IF OUT OF TOLERANCE

The tool radius verify and zero setting cycles is realized with G 304 program and can be recalled by others programs.

Example: N10 **G304/0.01 E.. QZ+.. Q1 QE..**

- Execute a verification cycle of the actual radius of the mounted tool; the actual radius is compared with the one written on the table and substituted to this one if the difference between them don't exceed the value written after the / character.

If the difference between them exceed the imposed value, the P201 parameter take the 0 value.

If the / character is not programmed the actual radius is re adjourned in the table.

Parameters P201, P202 and P203 are used.

If the G304 is programmed in the middle of the program it is important to re-program the M function (M3 or M..) and the spindle speed S ... in the next block of the program to restart the spindle.

Example : G304/0.02
 G0X...Y... **S1500 M13**

G305 TOOL RADIUS VERIFY CYCLE WITH AUTOMATIC TOOL SEARCH

Tool radius verify cycle with automatical tool search

The tool radius verify and zero setting cycles is realized with G 305 program and can be recalled by others programs.

Example: N10 **G305/0.01 E.. QZ+.. D1 Q1 QE..**

- Execute a verification cycle of the actual radius of the mounted tool; the actual radius is compared with the one written on the table and substituted to this one if the difference between them don't exceed the value written after the / character. If the difference between them exceed the imposed value, the P201 parameter take the 0 value, now its searched in the table a similar tool to substitute the one out of tolerance. If the / character is not programmed the actual radius is re adjourned in the table. Parameters P201, P202 and P203 are used.

If the G305 is programmed in the middle of the program it is important to re-program the M function (M3 or M..) and the spindle speed S ... in the next block of the program to restart the spindle.

Example : G305/0.02
 G0X...Y... **S1500 M13**

NOTA: when in the calling block of a subroutine is programmed / character, the written value is present P204 parameter to be used in the subroutine.

If it is not programmed, 0 value is in P204 parameter. Every time a subroutine is programmed the parameter P204 value is modified.

G306 TOOL LENGTH AND RADIUS MEASUREMENT

The tool length and radius automatic zero setting is realized with G 306 program and can be recalled by others programs.

Example: N10 **G306 M301 {T1,T2,T6,T8} E.. QZ+-. t.. Q1 QE..**

- Measure the T1, T2, T6, T8, tools length and radius and store it on disk
C:\CONF_DIR\TOOLS.WRK.

Tools are measured following the sequence indicated between the {} parenthesis.

No tools have to be in the cone to begin a tool length measuring cycle.

The spindle zero setting is done without tool before measuring the tool length.

Parameters P200, P201 and P202 P208 are used:

When {T1 + T5} is programmed, T1 to T5 tools are measured.

When {+ T7} is programmed, T1 to T7 tools are measured.

Before starting a zero set tool cycle, it is necessary to insert in the tools table a higher value of the effective length and an approximate radius to avoid to damage the instrument.

G316-G319-G350-G354 CNC AXES DEFINITION.

The CNC axes (14 + spindle) are indicated by the known address.
All the axes can be of CNC type or IND (independent) except the spindle that is always in CNC.

NOTE: the characteristic of CNC axes is that they can be moved in AUT all together to arrive in the same instant in the programmed position.
In Man are moved one by one.
They cannot be moved by the PLC command.
The IND axes are moved one by one. Their movement is managed by PLC as well in MAN than in AUT.
In AUT they can be moved also contemporary with the movement of CNC axes.
From part-program it is possible to indicate the position to get by the IND axes with an auxiliary function or by indicating the quota.
The independent axes quota is to be referred always to the origin 0.
CNC axes are divided in principal tern axes an auxiliaries axis.
The principal tern axes are divided in plane axes and normal axis to the plane.

At every reset the CNC present the following status:

- The first and the second CNC type configured axes are the two plane axes.
- The third CNC configured axis is the normal axis to the plane.
- The others CNC configured axis are the auxiliaries axis.
- The IND axes are the independent configured axes.
- The principal tern axes (3 first CNC configured axes) are also the three FEELER axes.

It is possible to modify the status of CNC axes in AUT from part-program.

There is a series of G functions used to modify the status of CNC axes:

- G316 {A,U}** Redefines the kind for the indicate axes address. All the indicated axes between parentheses before the comma modify their kind from CNC to IND.
All the indicated axes between parentheses after the comma modify their kind from IND to CNC. In the transformation from CNC to IND it is not possible to indicate the axes names of the principal tern.
- G317 {XWZABY}** redistribute CNC axes. It must to be indicating all and only the CNC axes.
The first two axes between parentheses became the new axes of the plane, the third become the normal axis to the plane and the remaining axes become auxiliary's axes.
- G318 {XYZ}** Definition of the PROBE tern axes. They have to be all CNC axes.

- G319 {X=a,Y=b}** Axis synonymes assignation.
At every configured axes it is possible to assign a synonym that can be only the initials of another recognized axis.
With the indicated example: all the times that on part-program is programmed a quotas proceeded by X or Y initial, this is considered as programmed with a or b initials. To annul the introduced synonyms it is necessary to write the G319 function without {} parenthesis.
- G350 {c}** Rotary table selection.
When the machine is equipped with more than one rotary table using this function it is possible to select from a program the one desired.
That choose enable the new rotation centres for the G748 programming function.
The initials written between { } select only one of the configured table.
At the start is enabled the first table configured.
- G354 {Y,v}** Parallel axis selection.
When the machine is equipped with two parallel axes to the main axis it is possible with this function the selection from a program one of those two.
Between the bracket are specified first the main axis initials and after the parallel axis initials.
The initials must be only one of the parallel axes configured.
At the start is enabled the first table configured.

G320 G321 Q INCREASE SUM ABILITATION.

The G321 function disabled the q increase sum on the programmed quota when the machine axes have to be moved in a fixed position that have not to change following the Q value programmed in the routines.

G320 function cancel G321 function by reactivating the Q values sum.
The G320 function is active at the start.

G322 G323 HELICOIDAL WORK IN GRINDING ABILITATION.

G323 function enables CNC to execute a circular helicoidal interpolation to effectuate the drill grinding.

This function allows the inversion of the helix axis movement sign without changing the movement of the circle axes.

The G322 function enables the helicoidal interpolation with constant step.
The G322 function is active at the start.

G327 TOOLS TABLE AND ORIGINS MODIFY AND SAVE FROM A PROGRAM

Function used to re-adjourn and/or save the tools tables and origins from a program.

Functions description:

G327 M360 [file name]

Are adjourned all the CNC origins quotas with the values written on the file.
It is possible to use only the files previously saved with the M361 function.

G327 M361 [file name]

Are saved all the CNC origins values on disk with the file name.
The format of this file is readable only with the M360 function.

G327 M362 [file name]

Are adjourned all the CNC tools quotas with the values written on the file.
It is possible to use only the files previously saved with the M363 function.

G327 M363 [file name]

Are saved all the CNC tools values on disk with the file name.
The format of this file is readable only with the M362 function.

G327 M370 [file name]

Are adjourned all the CNC origins quotas with the values written on the file.
The files to use must be in text. It is possible to use only the files previously saved with the M371 functions or the files *.ORI saved in C:\CNC_SAVE from the CNC tools origins table or edit the file with an editor.

G327 M371 [file name]

Are saved all the CNC origins values on disk on the file. The format of this file is text and is editable. We suggest saving this file on the directory C:\CNC_SAVE that is the same used from the CNC to save the origin.

G327 M372 [file name]

Are adjourned all the CNC tools quotas with the values written on the file.
The files to use must be in text. It is possible to use only the files previously saved with the M373 functions or the files *.UTI saved on C:\CNC_SAVE from the CNC tools table or edit the files with a editor.

G327 M373 [file name]

Are saved all CNC tools values on disk on the file. The format of this file is text and is editable. We suggest saving this file on the directory C:\CNC_SAVE that is the same used from the CNC to save the origin.

G327 FILE DEFINITION WITH NAME = TOOL POSITION

In a Part-Program it is possible to insert after the T a word that identify the tool that replace the tool number.

This word must be programmed between “ “ and is composed by a maximum of 32 characters.

Example of programming block for automatic pick-up tool cycle:

```
G45 T "Three cutters tool D20"
```

It is also necessary to specify in a File the tool number and the relative position on the magazine with phrases like:

```
T" Tool name " = Position on the magazine
```

That file must be called from the Part-Program by the G327 M395 [.....] function.

Example:

```
G327M395[C:\DIRECTORY\FILENAME]
```

A program with this function is showed by this example:

File with tools positions definitions: C:\DATA\POSUT

```
T"TOOL D10"=2  
T"Drill D3.5"=4  
T"Boring bar D50" =1  
T"CENTRE DRILL"=3  
T"ISO TAPP M6"=12
```

Example of a program that use the \DATA\POSUT file:

```
%N0  
N10G327M395[C:\DATA\POSUT]  
N20G45T"CENTRE DRILL"  
N30G0PA2X100Y100  
.....  
N50G47  
N60G45T"Drill D3.5"  
N70G81PA2X50Y50Z-30R10F1000  
.....  
N80G47  
N90G45T"TOOL D10"  
N100G0PA2X50Y50  
.....  
N120G47  
N9999%
```

G328 G329 DL AND DR AUTOMATIC CORRECTION

It is possible to change the DL and DR values on the tools table from the part-program by using those special functions:

G328 Assign to the DL on the table the value of the parameter P100.
The L value doesn't change.
The tools to modify must be writing on the M301{T2} function.
The G328 subroutine uses the M328 function.

G329 Assign to the DR on the table the value of the parameter P101.
The R-value doesn't change.
The tools to modify must be writing on the M301{T2} function.
The G329 subroutine uses the M329 function.

Example of the G328 programmed on a part-program:

```
N10 G328 M301 {T3} (adjourn DL of the tool N3)
```

Subroutine G328 in C:\MACRO

This subroutine put the P0 value in the DL.

```
N3280000  
N3280001 P100 = P0  
N3280002 G99 M328 (write the P100 value on the DL of T3)  
N3280003 G59
```

On the part-program:

```
N10 G329 M301 {T5} (adjourn DR of the tool N 5)
```

-Subroutine G329 in C:\MACRO

```
N3290000  
N3290001 G99 P101 = P1  
N3290002 G99 M329 (write the P101 value on the DR of T5)  
N3290003 G59
```

The functions M338 and M339 programmed on a block write in the parameters P100 and P101 the DL and DR value of the working tool.

G340-G341 RADIUS TOOL CORRECTION SELECTION.

Two more functions are available to select the way of correction of the tool radius correction during the profile work: G340 and G341.

CNC has the tool radius correction possibility by using the D function; G340 function is considered active.

The user can program at any time G341 function to activate the radius correction memorized in the tool table. This table is accessible by pushing F8 key.

G340 function cancels G341 function.

Remind that tool radius dimension is activated only during the G41-: -G44 approach cycles and in the G97 cycle.

G342-G343 ACTIVATION OR DE-ACTIVATION DL ON THE TOOL LENGTH

The G343 function cancels the DL correction on the tools length.

The G342 re-establish the DL correction on the length and is active from the start.

G344-G345 ACTIVATION OR DE-ACTIVATION DR ON THE TOOL RADIUS

The G345 function cancels the DR correction on the tools radius.

The G344 re-establish the DR correction on the radius and is active from the start.

G349 [] SUBROUTINE

If a program, containing the G349 [path/program name], is charged on the CN memory (1PROG-LOAD) when the G349 function is executed the CNC automatically charge the subroutine indicated inside the parentheses and is executed immediately.

A subroutine must be in ISO and have in the second block a G.. Function.

Example:	PROG1		PROG33
	%N0		
	N1 ...		
	N2G349[BLOCK\PROG33]	⇒	%N0
	N3	⇐ ⇐ ⇐	N20 G..
	N4	↑	N30
	N5	↑	N40
	N6000%	↑	N50
		⇐ ⇐	N99999%

The G359 function written in the subroutine is programmed to return to the main program.

Example:	PROG2		PROG33
	%N0		
	N1 ...		
	N2G349[BLOCK\PROG33]	⇒	%N0
	N3	⇐ ⇐ ⇐	N20 G..
	N4	↑	N30
	N5	⇐ ⇐	N40 G359
	N6000%		N50
			N99999%

A routine must start and finish in the same file.

It is not possible to nest the subroutines.

G351 ROTO-TRANSLATION

The G50 function cancels the G351.

The G351 function indicates a Roto-translation of the absolute reference system according to a new Cartesian system of programmed co-ordinates in the same block.

One time the G351 function is activated, all the followings elements must be programmed according to the new reference system.

The translation and rotation values are programmed in the line containing the G351 function:

G351X Y Z (translation) E (rotation angle)

It is possible to program the roto-translation in routines by using the QX QY QZ QE:

```
Example:  N10.....
          N20 G351 X0 Y0 Z0 E0
          N30.....
          N40.....
          N50.....
          N60 H1 N20 N50 QX10 QY10 QE30
          N70.....
```

The QX, QY, QZ and QE values are cancelled at the exit from the routines.

By using the G351 function it is impossible to roto-translate parts of programs with Q incremental repetitions:

```
example: %N0
          N10 G351 X Y E
          N20 G81 X Y Z R M S F PA1
          N30 H5 N20 QX QY           Execute five G81 to the X Y quotes of the N20
          N40 .....                line, for a correct program it is necessary to refer the
                                   routine on the N30 line to the line with the G351.
```

The G351 function executes first the translations and successively the rotations.

This function is active on the perpendicular plane of the tool either in G17 or G18 or G19 planes.

It isn't possible to use the function G351 together with G51, G751 or G752.

G355 HEAD ROTATION WITHOUT ADJUSTMENT OF THE WORK PLANE

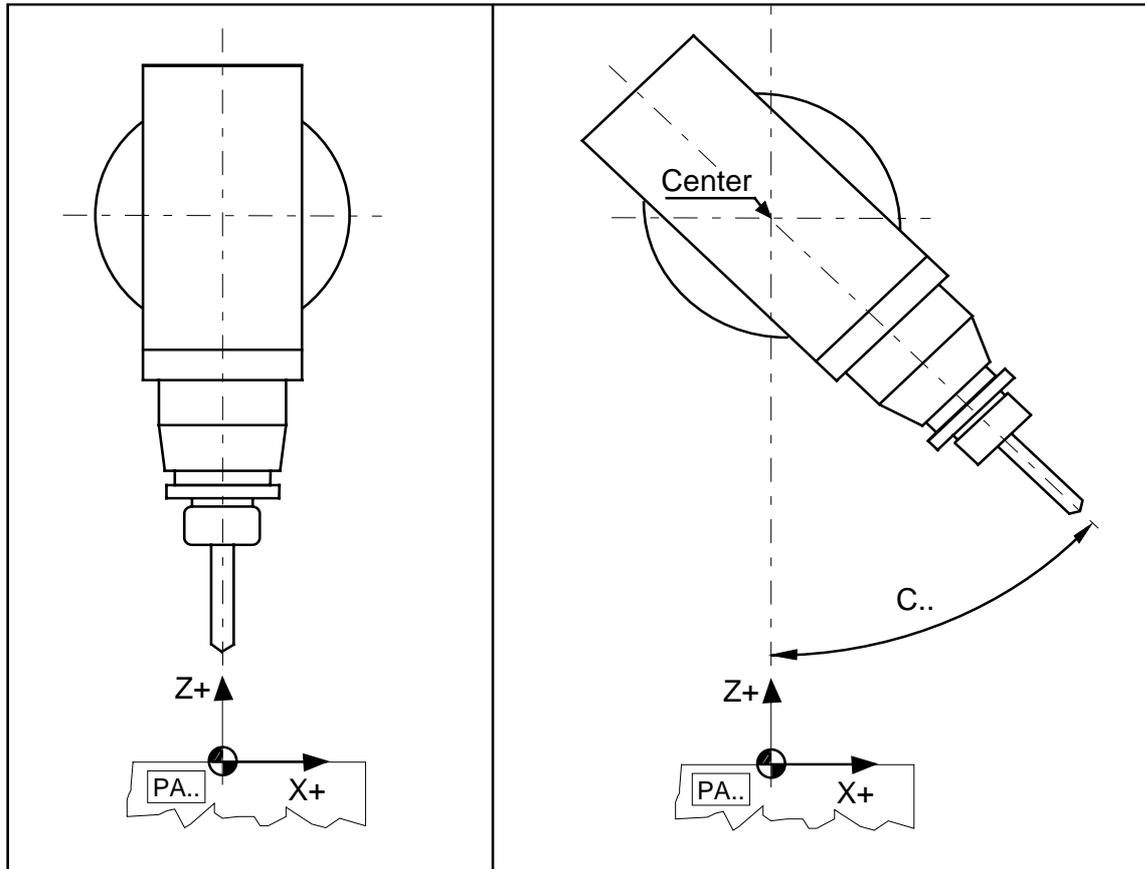
It identifies the Rotation Cycle of the Head, without adjustment of the Work Plane to the Rotation angle. It is modal and is active at the CN start.

This G355 is cancelled by programming G55, G56, G356, and G755 and by pushing the RESET.

The G355 cancels the previous rotation mode.

It cancels an eventual translation of the origin. The head rotation is done from the fulcrum of the head.

Example: G355C-60



G356 SPECIAL HEAD ROTATION

It is modal and is active at the CN start.

This G355 is cancelled by programming G55, G56, G356, G755 and by pushing the RESET.

In the G356 block it is possible to program an origin translation in X, Y and Z.

The reference system is rotated of the programmed angle.

No possible to use together G746 function.

Example: %N0
N10 G356 X20 Y-10 Z30 C30
N20 G0 X0 Y30
N30 C5
N40 Y40
N50 C-5
N60 Y30
N70 Z2

All the lines programmed after the N20 will be executed with an origin translation about 20mm in X, -10mm in Y, 30mm in Z and with the X Y plane rotated respect the Y axis of 30 degrees since be orthogonal to the head position.

The C5 degrees programmed on the block N30 turn the head to 35 degrees. The passing from C30 to C35 is done in G55 mode and remain activated the reference system rotated of 30 degrees as programmed on the block N10.

G358 TABLE AXIS MOVEMENT AS A SPINDLE

This function permit to a rotary axis (table) to move as a spindle by rotating with an adjustable number of revolutions.

To the table axis will be applied a command able to rotate it in continuous by ignoring the end of courses and the origins.

When the rotary axis is in this mode on the quota visualised on the main menu must appear the rotation speed expressed in rpm that replace the position quota.

This kind of movement must remains activated also when the axes moves in traditional way.

This kind of movement can be used in automatic and also in manual.

In automatic is activated writing this block on the part-program:

G358 {axis name movement sign} **S** rpm

The rotation speed can be regulated of $\pm 30\%$ by using the potentiometer. This kind of movement is cancelled by the reset or by programming on the part-program the function:

G357

Example: %N0

 N5 G358{ A+ } S100

 N17 G357

In manual a special soft key select this kind of table movement.

When this selection is on the soft key is lighted. By pushing again this key, when the table is stopped, this function is cancelled, or by the reset.

The axis movement is activated, if predisposed, by another soft key that if pushed start the rotation on the direction established by two keys.

When the table has the rotation on this soft key is lighted. The rotation speed is regulated from 0 to 100% by using the spindle rotation potentiometer in manual. This kind of movement is erased by pushing again the soft key or by the block button or by the reset.

When the rotary table moves in this mode it is possible to rotate also the spindle and to move the others axes, this to consent the determination of the piece origins.

G359 RETURN FROM SUBROUTINE TO MAIN PROGRAM

This function allows to return the execution to the calling program after be exit with the G349 function.

G360 INTERPOLATION WITH N.U.R.B.S.

The CNC can move using the **N.U.R.B.S. CURVES OF THIRD DEGREE**.

It can transform a file of points in Cartesian co-ordinates, generated by a digitising system or by a Cam, to a file that defines the same curves on the space, by verifying and by elaborating again the tangency of the elements.

All this procedures can be executed on board of CNC, the transformation of a ISO file in curves is possible by using the selection "Pre elaboration of CAD files", on the main menu.

A curve on space is programmed with a sequence of three blocks with G360:

```
N1000 G360 X Y Z A C  
N1001 X Y Z A C  
N1002 X Y Z A C R L
```

It is not necessary to program the G360 function in the second and third block. The **N** function visualised is the last of the three blocks.

With the **X Y Z A B C** parameters are defined the coefficients for the curve programming.

For CNC release 12 and subsequent it is possible to create NURBS defined by 6 axes: **X Y Z A B C** where XYZ are the main axes, A is the table, B is the dividing table and C is the continuous head that includes the tool.

The **R** function is the smaller radius of curvature. (It is programmed with the decimal point to separate the mm and must be programmed in the third block).

The **L** function is the curve length on space. (It is programmed with the decimal point to separate the mm and must be programmed in the third block).

G360 must be used only in G31 Mode.

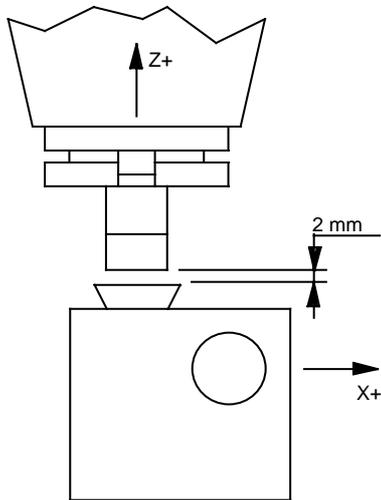
A curve on space must be programmed always with three blocks.

A series of curves on space must be a block number multiple of three.

It is not possible to convert a program in N.U.R.B.S. if contains the radius compensation in space G31 D.. and P..Q..R.. (vectors).

G377 SETTING OF THE INSTRUMENT FOR THE TOOLS LENGTH PRESETTING

This function must be done using the Master-tool.
Move the machine manually with the Master-tool over the instrument for the tools length.



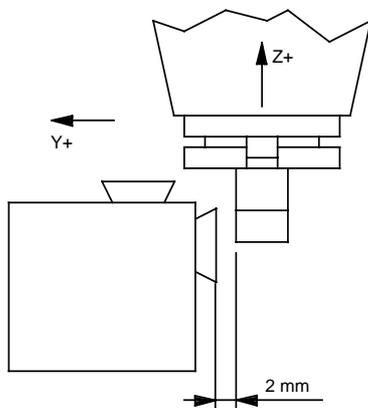
Edit the program:

```
%N0  
G377F500  
N9999%
```

Press **START CN**.

G378 SETTING OF THE INSTRUMENT FOR THE TOOLS RADIUS PRESETTING

Move the machine manually with the Master-tool near the instrument for the tools radius.



Edit the program:

```
%N0  
G378F500  
N9999%
```

Press **START CN**

N.B. THE RAPID AND THE FEEDRATE POTENTIOMETERS MUST BE SET OVER THE 50% AT LEAST.

G384 RIGID TAPPING CYCLE (start with spindle stopped)

It identifies the rigid tapping cycle.

Example: N1 G0 X Y Z A B PA. T..
N2 G384 X Y Z A B R M.. S.... F.....QZ.....

The CNC will executed these operation in sequence:

- 1) Traverse to the X Y A B position.
- 2) Traverse to the L plane.
- 3) Feedrate to the Z plane.
- 3a) Executed in substitution of point 3 if programmed the QZ-...:
Feedrate to L+QZ or to L+QZ + (QZ - 10%)
- 4) Spindle motion reverse and return at feedrate to the R plane.
- 5) Remove spindle reversal.
- 6) Traverse to the R plane.

It continues to repeat the operations defined in the point's 3a, 4 and 5 decrementing the QZ of the 10% each time until to a maximum of the 50%. Reach this value the QZ stays constant up to the attainment of the final quota.

In G384 cycle the value of the started F has the drilling pitch and it is express in Micron/Revolutions ($F 1000 = 1 \text{ mm./rev.}$).

The axis in operation goes on following the programmed pitch with the number of the spindle revolutions, so also if we modify the revolutions with the Override Spindle potentiometer, the drilling does not feel the effects.

It will be possible to re-execute the tapping.

For allow the turn up of the fillet it needs that the tool has the same angular position as the spindle key.

For execute a series of drilling cycle it is not necessary to rewrite the G384 but, in the followed blocks, the new co-ordinates and if changed the others informations about: F, S, R, L, Z.

G390-G391 ABSOLUTES QUOTAS PROGRAMMING

CNC predisposition to the programming of measuring system absolutes quotas.

The G390 allows to work using quotas refereed to the absolute zero of the machine (without to consider the origins PA..)

This function cannot be used for the movement of the tilting head.

The re-establishment of the programming in relative quotas (default) is possible by programming G391 or by a changing the origin or tool.

G399 Z axis rapid move to 5 mm from the positive end of course

This function permits to move the Z axis to 5 mm from the positive end of course; it must be programmed after the G390 function (absolute quotes), the block after the G399 function must have re-programmed the piece origin definition.

Example: N13
N14 G0Z10
N15 G390
N16 G399
N17 G0X100Y15 **PA1**
N18

G428/ PICK-UP PIECE CYCLE

This function is the pick-up piece cycle used on the machine with pieces magazine (Optional).

The G428/nn pick-up the piece that is positioned on the magazine to the position number nn and put this piece in the dividing table.

The grip for the pick-up/release piece is automatically taken and must be positioned in the last position of the magazine (ex. n° 17).

The grip length corrector must be the last tool length corrector (ex: T17).

The pieces magazine origin must be the **PA8**.

The dividing head origin for the pieces change must be the **PA7**.

Example: N1 G428/2

G429/ RELEASE PIECE CYCLE

This function is the pick-up piece cycle used on the machine with pieces magazine (Optional)

The G428/nn pick-up the piece that is positioned on the dividing table and release the piece in the magazine in the position number nn.

The grip for the pick-up/release piece is automatically taken and must be positioned in the last position of the magazine (ex. n° 17).

The grip length corrector must be the last tool length corrector (ex: T17).

The pieces magazine origin must be the **PA8**.

The dividing head origin for the pieces change must be the **PA7**.

Example: N1 G429/2

G500 TOOLS LENGTH MEASUREMENT WITH MANUAL TOOL-CHANGE

The tool length measurement cycle is made through the sub-program G500 and can be recalled by others program.

E.g. *N10 G500 M301 Z200 {T1,T2,T6,T8} QE..*

During the manual tool-change the Z axis quota means the distance between the spindle nose and the PA0 origin.

By programming G500/1 it is possible to insert the special calibres manually.

G503 TOOLS RADIUS MEASUREMENT WITH MANUAL TOOL-CHANGE

The tools radius measurement cycle is carried-out by the G503 sub-program and can be recalled by the others program.

E.g. *N10 G503 M301 Z200 {T1,T2,T6,T8} QE..*

During the manual tool-change the Z axis quota means the distance between the spindle nose and the PA0 origin.

In this cycle was been foreseen the **automatic** sample tool laying-down in the tools magazine.

By programming G503/1 it is possible to insert the special calibres manually.

G506 TOOLS LENGTH AND RADIUS MEAS. WITH MANUAL TOOL CHANGE

The tools length and radius measurement cycle is made through the G506 sub-program and can be re-called by others program.

E.g. *N10 G506 M301 Z200 {T1,T2,T6,T8} QE..*

During the manual tool-change the Z-axis quota means the distance between the spindle nose and the PA0 origin.

In this cycle was been foreseen the **automatic** sample tool laying-down in the tools magazine.

By programming G506/1 it is possible to insert the special calibres manually.

G550 CONFIGURATION AUTOMATIC RE-ESTABLISHMENT

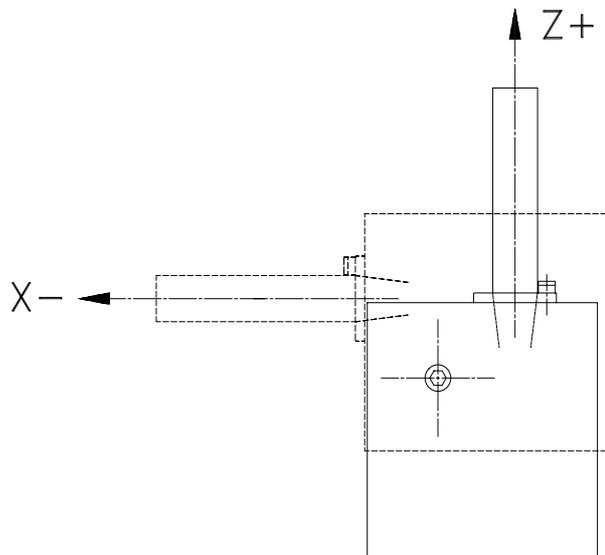
The **G550** function erases the G551 previously programmed and restores the normal condition of the configuration.

Example: G550

G551 CONFIGURATION AUTOMATIC CHANGE

This function allows the automatic configuration change.

Used to work with the normal configuration where are usually set the 6 axes machines (3 linear + 3 rotating) in a "different conditions" from that operating-one at the machine turning-on. For example, let's suppose to have a machine equipped with a dividing unit that turns around the **Y-** axis and hit table that rotates around the **Z+** axis. We want mill a fan blade or digitize through cycles that employ the table-rotating axis: in both these cases is necessary enable the table to rotate around an axis parallel to the **X+** or **X-** axis.



To execute this the operator must:

- 1) Rotate the **B** axis from 0 to 90 degrees.
- 2) Zero-set a work piece origin with the dividing unit still in the B=0 position.
- 3) Write on one of the working file first command instructions as follows:

```
G551{ 90 }
```

WARNING: The angle value written between the parenthesis { } can be 90 or -90. Every other value written in the file will cause an error message.

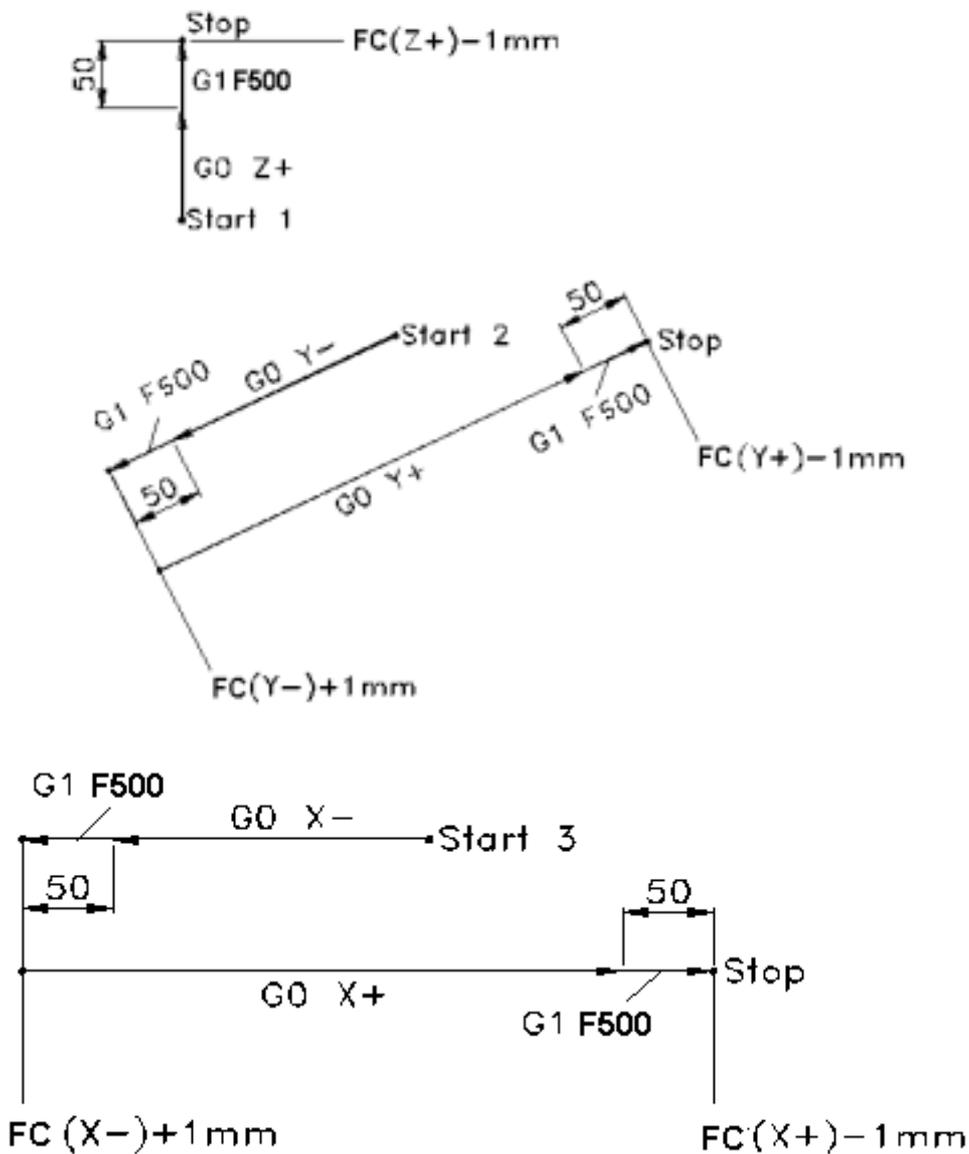
G690 BEARINGS CAGE REALIGNMENT.

On machines with bearings cages , after 70 hours of CNC working shows a yellow message: Execute G690 cycle for bearings cage realignments.

During the following ten hours it is necessary to run the cycle G690.

After these 10 hours (80 hours of CNC working) the message will became red and CNC only accept to start the cycle G690.

- 1) Prepare the machine without the tool on the spindle and free to move in all the axes traverses and the head on position Z high.
- 2) Program the G690 function and after the Start the machine executes in sequence the moves shown on figure; first in Z after in Y and finally in X.
- 3) At the end of the G690 cycle the message disappears automatically.

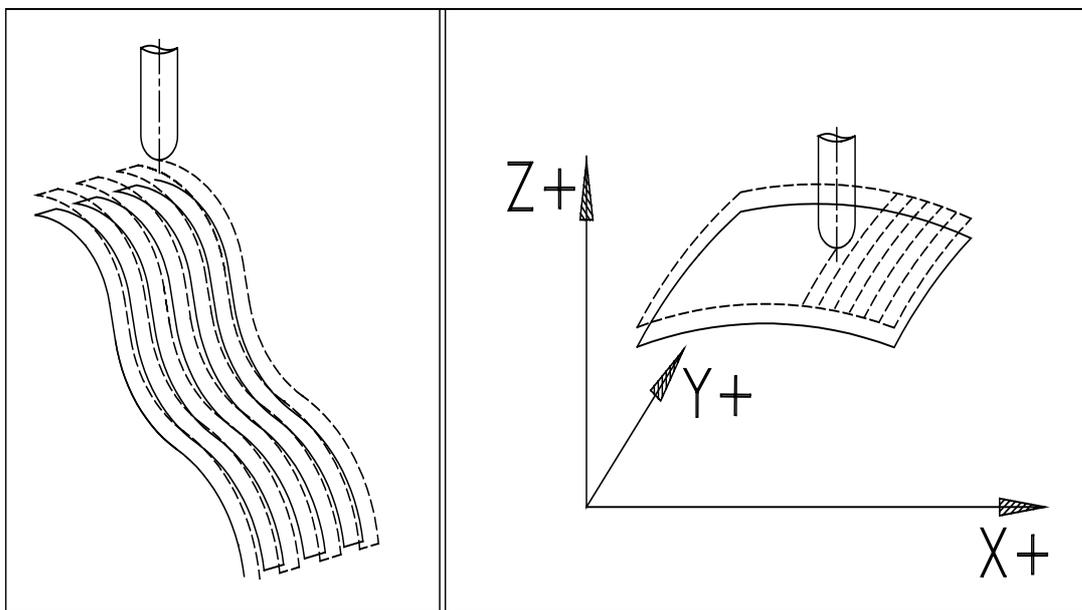


G731 D.. TOOL RADIUS COMPENS. IN SPACE ZERO ON THE TOOL POINT

Is programmed on the **G731 D..** block where D is the tool radius compensation.
Like the G31 D function the G731D..enables the radius correction but also it makes a translation in Z- direction equal to the D value.
This permit to zero-setting the spherical tool on the point.

The 3D program must be composed by the 3 main axes (X, Y and Z) but also by the 3 correction factors (P, Q and R) e.g.: **N10 X.. Y.. Z. P.. Q.. R..**

These correction factors values P, Q and R can change from -1 to 1.
No possible to use together G63 function.
No tool collision checks are done on the points of the surface.



G740 CANCELS G746 and G748

The G740 cancels a G746 or G748.

Example: G740

G746 AXES SYSTEM ROTATION ASSOCIATED TO THE TABLE ROTATION

This function makes the translation and rotation of the axes reference system associated to the movement of tables and diving heads rotating axes.

The axes system PA.. rotate around the centre of the table, and then it will be not parallel to the axes of the machine.

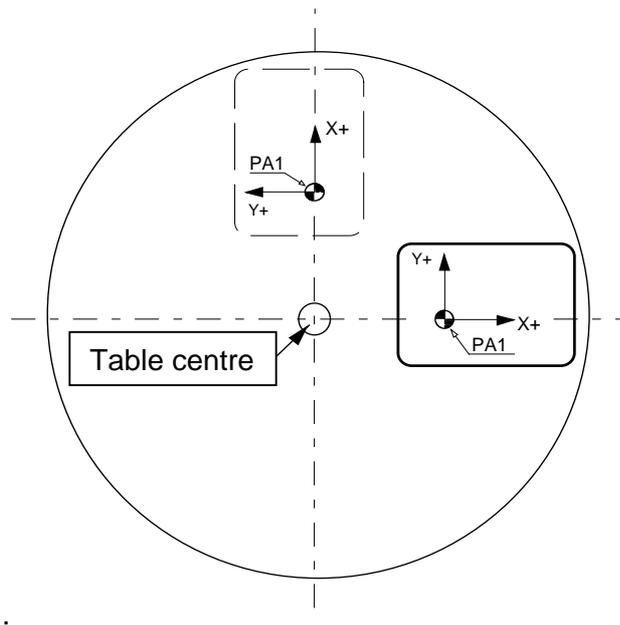
No possible to use together G56 or G356 functions.

This function is cancelled by the G740 function.

Syntax: **G746 { A }** or **G746 { B }** or **G746 { AB }**

Before to use the G746 function it is necessary to make the table pre-set
(See on User Manual Shift-F2 Key).

Note: when are programmed both the rotations (G746 {AB}) is executed first the table rotation and after the tilting table rotation.



G748 AXES SYSTEM TRANSLATION ASSOCIATED TO THE TABLE ROTATION

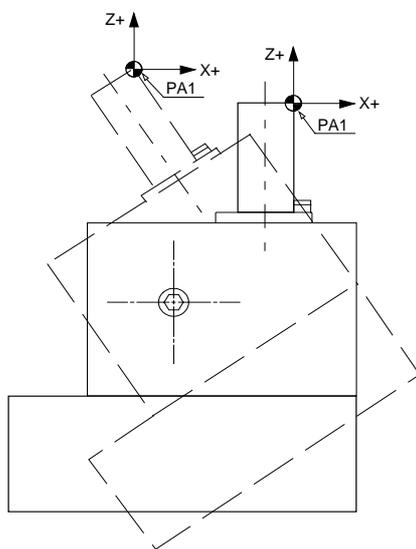
This function makes the translation of the axes reference system associated to the movement of tables and diving heads rotating axes.

The axes system PA.. rotate around the centre of the table, **but remain parallel to the axes of the machine.**

This function is cancelled by the G740 function.

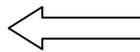
Syntax: **G748 { A }** or **G748 { B }** or **G748 { AB }**

Before to use the G748 function it is necessary to make the table pre-set (See on User Manual Shift-F2 Key).

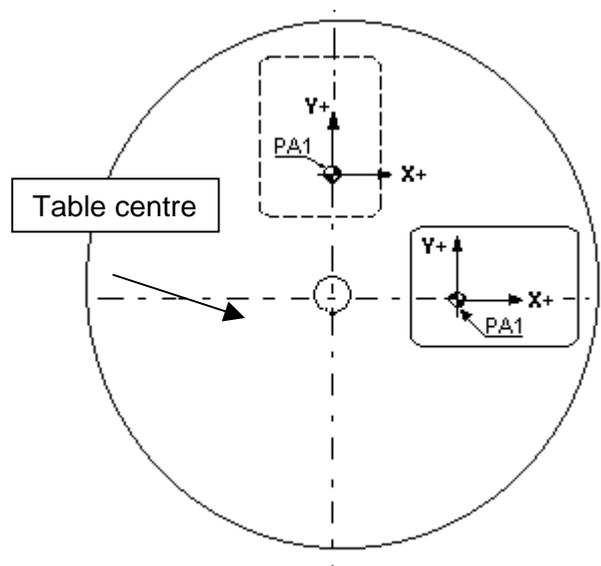
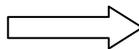


Example:

```
%N0
N1 G45T1
N2 G0A0B45PA1
N3 G748{B}
N4 G0X..Y..
N5 Z..
N6 .....
N7 .....
N34 G740
N35 .....
```



Example: %N0
 N1 G45T1
 N2 G0A90PA1
 N3 G748{A}
 N4 G0X..Y..
 N5 Z..
 N6
 N7
 N34 G740
 N35



3D ROTATION-TRANSLATION

The functions we are going to define are a generalisation of the well-known plane roto-translation function **G51**. With the **G51** function it is possible to operate on the piece co-ordinates, rotations and/or translations on the active work plane (programmed by **G17**, **G18** and **G19**). The generalisation consists on the overcoming of the work plane concept by having the possibilities to program simultaneous moves report to the 3 planes co-ordinates.

G751 TRANSLATION

This function allows a translation of the piece co-ordinates on the direction defined by the user. On the line where is programmed the **G751** function can and must appear the quota of one axis only.

Example 1: %N0

 G751X30
 G0X100Y100Z200PA1F2000
 G1Z0
 X20Y10
 X30Y15

Produces this tool path: G0X130Y100Z200
 G1Z0
 X50Y10
 X60Y15

Can be programmed other translations by writing on the next lines (immediately or not) the same function.

Example 2: %N0

 G751X30
 G751Y50
 G751Z100
 G0X100Y100Z200PA1F2000
 G1Z0
 X20Y10
 X30Y15

Produces this tool path: G0X130Y150Z300
 G1Z100
 X50Y60
 X60Y65

The **G751** function is in incremental that means the new **G751** function will sum the previous **G751** functions:

Example 3: %N0
 ...
 G751X30
 G751X30
 G0X100Y100Z200PA1F2000
 G1Z0
 X20Y10
 X30Y15

Produces the tool path: G0X160Y100Z200
 G1Z0
 X80Y10
 X90Y15

But also: %N0

 G751X60
 G0X100Y100Z200PA1F2000
 G1Z0
 X20Y10
 X30Y15

Produces the same tool path.

To reset the **G751** function uses the **G750** function.

It isn't possible to use the function G751 together with G51 or G351.

G752 ROTATION

This function allows a rotation of the piece co-ordinates around the axis defined by the user. On the line where is programmed the **G752** function can and must appear the quota of one axis only.

Example 1:

```
%N0
.....
G752Z90
G0X100Y100Z200PA1F2000
G1Z0
X20Y10
X30Y15
.....
.....
```

Produces the tool path:

```
G0X-100Y100Z200
G1Z0
X-10Y20
X-15Y30
```

Can be programmed other rotations by writing on the next lines (immediately or not) the same function.

Example 2: In this example is applied first a rotation of 90° from Z+ and after a rotation of 90° from Y+: the first transformation produces:

$\begin{aligned} X+ &\rightarrow Y+ \\ Y+ &\rightarrow X- \\ Z+ &\rightarrow Z+ \end{aligned}$
--

The second

$\begin{aligned} X+ &\rightarrow Z- \\ Y+ &\rightarrow Y+ \\ Z+ &\rightarrow X+ \end{aligned}$
--

That means

$\begin{aligned} X- &\rightarrow Z+ \\ Y+ &\rightarrow Y+ \\ Z+ &\rightarrow X+ \end{aligned}$
--

The composition of the 2 produce

$\begin{aligned} X+ &\rightarrow Y+ \\ Y+ &\rightarrow Z+ \\ Z+ &\rightarrow X+ \end{aligned}$
--

Therefore: %N0

 G752Z90
 G752Y90
 G0X100Y200Z300PA1F2000
 G1Z0
 X20Y10
 X30Y15

Produces the tool path: G0X300Y100Z200
 G1X0
 Y20Z20
 Y30Z15

The **G752** function is in incremental that means the new **G752** function will sum the previous **G752** functions:

Example 3: %N0

 G752Z30
 G752Z30
 G0X100Y200Z300PA1F2000
 G1Z0
 X20Y10
 X30Y15

Produces the tool path: G0X-123.2051Y186.6025Z300
 G1Z0
 X-7.3205Y27.3205
 X2.0096Y33.4808

But also: %N0

 G752Z60
 G0X100Y200Z300PA1F2000
 G1Z0
 X20Y10
 X30Y15

Produces the same tool path.

To reset the **G752** function uses the **G750** function.

It isn't possible to use the function G752 together with G51, or G351.

G755 HEAD ROTATION WITH SPHERICALS TOOLS

The G755 function allows rotating the head with a spherical tool of a programmed C angle (or A or B); the reference system remains the same.

Before to start the rotation the G755 function subtract to the tool length a value equal to the radius "r" write on the tools table plus the L value and make a translation in Z+ direction of the same value.

Considering a length equal to this value does all the successive head rotations executed in G0.

The G755 function is modal; and remains active until is cancelled by a G55, G56, G355 and G356 and by pushing the RESET.

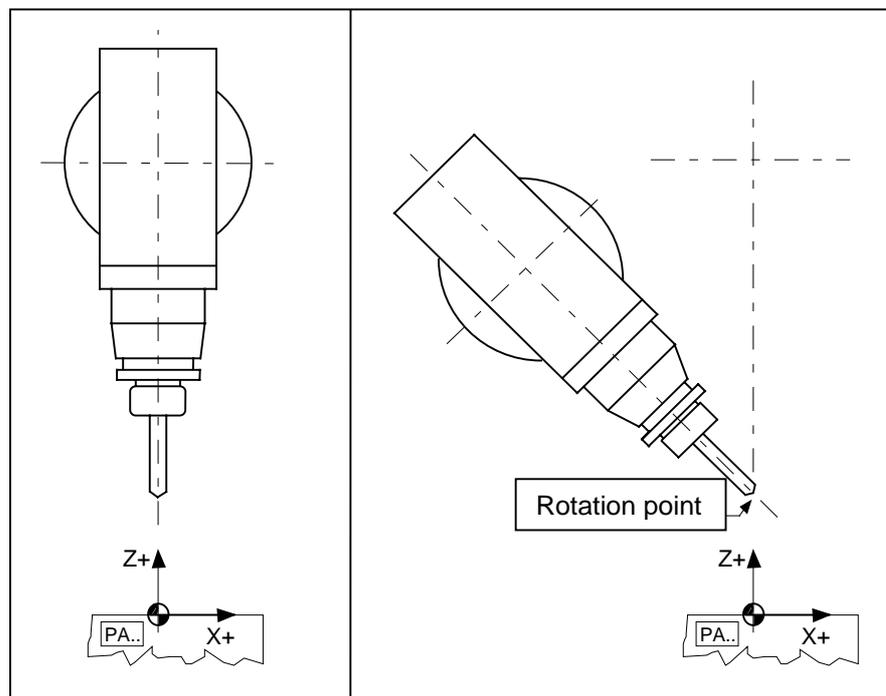
As the G755 function modify the tool length we recommend to use it when the head angle is equal a 0. It is also better to rotate the head to 0 before to program other rotation functions like G55 or G56 or G355 or G356.

The rotation sense is positive if the rotation is clockwise for an operator that looks the head of the machine (machines Type A, B, S) frontally.

Example: %N0
 N10G755C-60
 N20G0X50Y100Z50 PA1

The head is turned at -60 degrees.

The rotation is executed on the spherical tool centre by moving consequently also the axes interested by the rotation.



H ROUTINE

With an instruction bloc, called routine "H", it is possible to repeat up to 999 times a group of blocks, also changing each time some values inside the blocks by the incremental words:

QX..., QY..., QZ..., QA..., QB..., QC..., QD..., QE...

The QX, QY and QZ increments automatically adjusts also the I, J and K (values of centres circles).

Example: N100 H3 N10 N60 QX100

This bloc will execute 3 times the operations included between the bloc number 10 and 60, with an increment of 100 mm on X and I.

If we use a routine H in a program with roto-translation (G51-G351); this must include the bloc of the G51 or G351 function (see ex. B).

It is also possible to include routine inside routine up to 10 nested levels. The increments are related to each level.

EXAMPLE A: (Incr. routine in Z)

```
%N0
.....
N5 G1Z-2
N6 G41X..Y..D..
.....
.....
.....
N50 G40X..Y..
N51 H4 N5 N50 QZ-2
.....
N99999%
```

EXAMPLE B: (Incr. routine with angle E)

```
%N0
.....
N3 G51 X0 Y0 E0
N6 G0 X100 Y-10 PA1
N7 Z5
.....
.....
N50 G0Z5
N51 H4 N3 N50 QE-20
N52 G50
.....
N99999%
```

EXAMPLE C: (Drilling on a matrix)

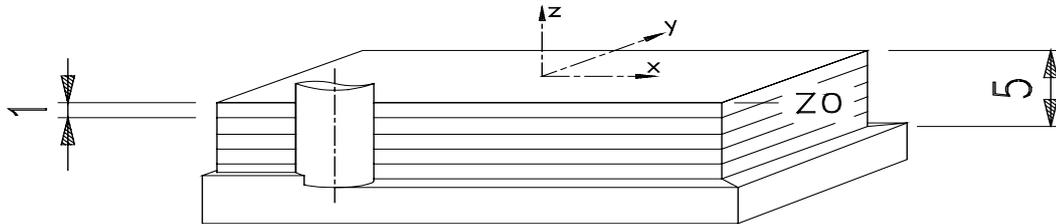
```
%N0
N1 G45T1
N2 G81X5Y5Z-5R3PA1F...
N3 H9 N2 QX10
N4 H4 N2 N3 QY8
N5 G47
N99999%
```

The maximal program dimension where it is possible to include a routine is 1000 Kbytes.

It is also possible to include in a routine "H" a subroutine call using the function G349 [\BLOCK\file name].

EXAMPLES OF REPETITIONS:

PROFILE WITH VARIOUS PASSES IN Z



```

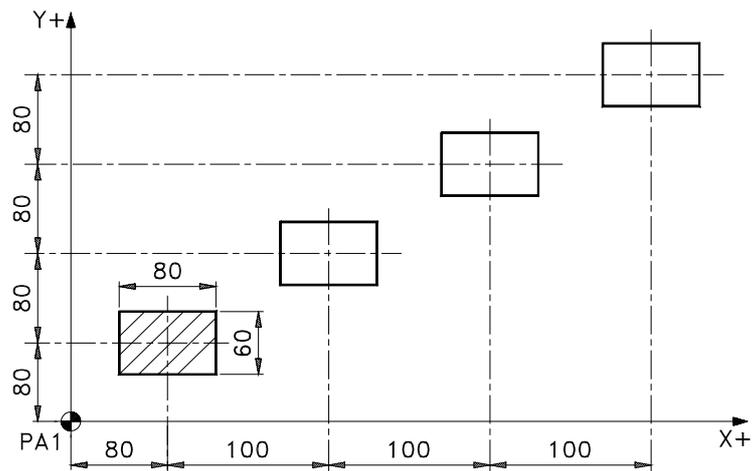
%N0
N1 G45 T1
N2 G0X-25Y20PA1F300S700
N3 Z5M13
N4 G1Z-1
N5 G41Y12D5
N6 G1X15
N7 Y-12
N8 X-15
N9 Y20
N10 G40X-25
N11 H4 N4 N10 QZ-1
N12 G0Z5
N14 G47
N9999%
    
```

LINEAR REPETITION OF A PROFILE

```

%N0
N1 G45 T1
N2 G0X20Y20PA1
N3 Z5F200S600M13
N4 G1Z-2
N5 G41X40D8
N6 G1Y110
N7 X120
N8 Y50
N9 X20
N10G40Y20
N12G0Z5

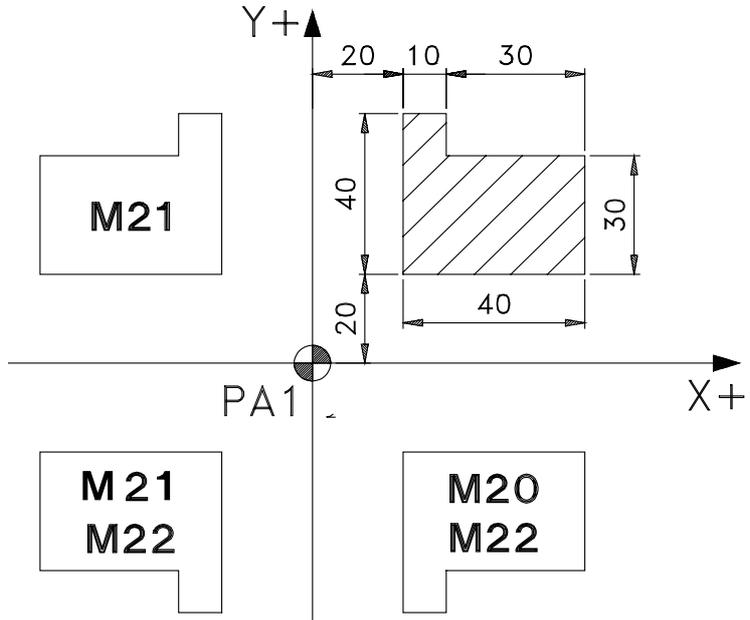
N13H3N2N12QX100QY8
0
N14G47
N9999%
    
```



MIRROR OF A PROFILE

```

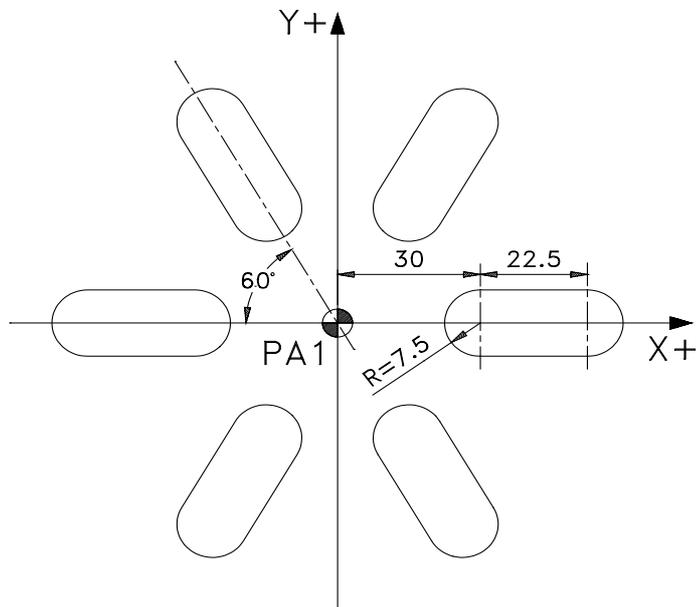
%N0
N1 G45 T1
N2 G0X5Y5PA1F300S700
N3 Z5M13
N4 G1 Z-3
N5 G41X20D8
N6 G1Y60
N7 X30
N8 Y50
N9 X60
N10 Y20
N11 X5
N12 G40Y5
N13 G0Z5
N14 G99 M21
N15 H1 N2 N13
N16 G99 M22
N17 H1 N15
N18 G99 M20
N19 G99 M22
N20 H1 N15
N21 G99 M20
N22 G47
N9999%
    
```



ANGULAR REPETITION OF A PROFILE

```

%N0
N1 G45 T1
N2 G51X0Y0E0
N3 G0X10Y0PA1F300S700
N4 Z5M13
N5 G1Z-2
N6 G41X22.5D5
N7 G2X30Y7.5I30J0
N8 G1X52.5
N9 G2X52.5Y-7.5I52.5J0
N10 G1X30
N11 G2X22.5Y0I30J0
N12 G40X10
N13 G0Z5
N14 H5 N2 N13 QE60
N15 G50
N16 G47
N9999%
    
```



M FUNCTIONS

M1	Programmed stop.
M3	Clockwise spindle rotation.
M4	Anticlockwise spindle rotation.
M5	Stop spindle.
M6	Manual changing tool (Associate to G99).
M7	Enable coolant 1 (outside water).
M8	Enable coolant 2 (water in the spindle).
M9	Disable coolant.
M10	Enable non-programmed axes locking.
M11	Disable non-programmed axes locking.
M13	M3+M7
M14	M4+M7
M15	M3+M8
M16	M4+M8
M17	M3+M50
M18	M4+M50
M19	Spindle orientation.
M20	Cancels inversion sign on axes (active at the start).
M21	Mirror X axis (Quotas X and I, change G2 in G3 and G41 in G42).
M22	Mirror Y axis (Quotas Y and J, change G2 in G3 and G41 in G42).
M23	Mirror Z axis (Quotas Z and K, change G2 in G3 and G41 in G42).
M31	Allow inversion of the sign (only for quotas X and I).
M32	Allow inversion of the sign (only for quotas Y and J).
M33	Allow inversion of the sign (only for quotas Z and K).
M37	Allow inversion of the sign (only for quotas XY and IJ QX QY).
M38	Allow inversion of the sign (only for quotas XZ and IK QX QZ).
M39	Allow inversion of the sign (only for quotas YZ and JK QY QZ).
M40	Allow low gamma (if it is on tool machine)
M42	Allow high gamma (if it is on tool machine)
M50	Enable coolant 3 (outside air)
M51	Enable coolant 4 (outside air-oil)
M52	Enable coolant 5 (air-oil in the spindle)
M53	Enable coolant 4 (outside oil)
M58	Rotary table revolutions reset
M60	Exit strobe 200 msec for pneumatic table control
M80	General power off
M110	Axis A Cone cleaning
M111	Axis A STOP Cone cleaning
M113	M3+M51
M114	M4+M51
M115	M3+M52
M116	M4+M52
M117	M3+M53

M118	M4+M53
M124	Tools magazine OUT (optional)
M125	Tools magazine IN (optional)
M135	Magazine rotation to the position of tool T.. (M135 T1)
M158	Grip opening (optional)
M159	Grip closing (optional)
M177	Rotable magazine door open
M180	Rotable magazine door close
M184	Tools/pieces magazine OUT (optional)
M185	Tools/pieces magazine IN (optional)
M200	Work time
M704	Tail-stock forward with normal air pressure (optional)
M704/1	Tail-stock forward with reduced air pressure (optional)
M705	Tailstock backward (optional)
M708	Tail-stock locking (optional)
M709	Tailstock releasing (optional)
M728	Disable coolant selector
M729	Enable coolant selector
M730	Machine pneumatic door open (optional)

M10-M11 FUNCTIONS

M10 and M11 are modal functions. They establish a certain type of tool machine functioning until one annuls the other.

M10 function is the natural one because is reactivated at every new CN configuration installation. This function establishes that the not programmed axes inside a movement block or those that don't work during a fixed cycle are blocked. It is possible to disable the axes locking programming M11 with the axes inside the { } parentheses.

If the M11 function is programmed without axes names cause the unlocking of all the axes except the indexing axes (ex. C index axis).

To lock specifies axes it is sufficient to program M10 function with the axes inside the {} parentheses. When M10 functions is programmed without indication of axes to lock, all the configured axes are locked.

For normal 2-3 axes programs and with rotary axes not simultaneous:

Example: N30 G99M10
 N31M11{XYZ}

For 4 or 5 axes programs with rotary axes simultaneous:

Example: N30 G99M10
 N31M11{XYZAB}

M40-M42 SPINDLE GEAR

Through the function S the spindle r.p.m. is programmed. Through the number set, if the machine is equipped with some rotation gears, is automatically selected and used the right one.

During the G84 or G384 tapping cycle programmed is forced the M42 for all r.p.m. If is necessary to carry out the tapping in a Low Gear the function M40 has got to be programmed in the previous program block. The M40 programmed will remain operative until is run an automatic or manual tool change or when another function M will be programmed again.

M58{ rotary axis } ROTARY TABLE REVOLUTIONS RESET

This function is used to reset automatically the revolutions of rotary table. To be used with the axis in movement but almost 50° after the 360° and not on the last working block.

Example:

```
G1X-31.2055Y-3.3941Z20.398A409.8421
X-31.2048Y-3.1446Z20.4693A410.6596
X-31.2041Y-2.8865Z20.5373A51.4979 M58{A}{follow in the same direction)
X-31.2035Y-2.6272Z20.602A52.3361
X-31.2028Y-2.3541Z20.6643A53.2127
.....
```

M184 PIECES MAGAZINE OPENING

This function is programmable only when: magazine is in closed position, dividing unit is upright, (B AXIS), and magazine access side door is closed.

If the machine is switched on with the magazine in a mid travel position, is possible to move it in manual carrying out:

```
F11 SOFT_KEYS
F1 AXES JOG MAN
```

So, pull the magazine forward by F5 PARTS STORE/FWD or backward by F9 PARTS STORE/BKG (when the button is left the magazine stops).

When one of the two extreme positions is reached (inside or outside) the magazine will be managed no more by the buttons F5 or F9, but only by M184 and M185 functions.

M185 PIECES MAGAZINE CLOSING

This function is programmable only when: magazine is in open position, dividing unit is upright, (B AXIS), and magazine access side door is closed.

If the machine is switched on with the magazine in a mid travel position follow the same procedure for the M184 function described above.

M200 WORK TIME

It is possible to write in a part-program the M200 function to have the work time.
After a movement or the spindle activation the timer starts, the time is memorised in the P245 parameter (Shift+F6 PAR) as minutes. seconds.

Program example with M200 function:

```
%N0
G99M200
N1G0PA1X0Y0
G0T1Z100
..... (Piece program)
N2G0Z100
G99 (P1 TIME IN SECONDS)
N3 P1=P1+(INT(P245)*60)+(P245-INT(P245))*100
G99 (P2 TIME IN MINUTES.)
N4 P2=P2+INT(P245)+(P245-INT(P245))*(100/60)
N99999%
```

M320-M327 "STOP IN MICRO"

These "M" functions allow stopping the machine movement programmed in the block, when an external event intervenes.

The external event is fully asynchronous as regards the part-program execution.

Are recognized 4 simultaneous external events called as MICRO 1-:-4.

On these Micros transition status is possible to program:

- M 320** Stops the machine movement during the MICRO 1 closing
- M 321** Stops the machine movement during the MICRO 1 opening
- M 322** Stops the machine movement during the MICRO 2 closing
- M 323** Stops the machine movement during the MICRO 2 opening
- M 324** Stops the machine movement during the MICRO 3 closing
- M 325** Stops the machine movement during the MICRO 3 opening
- M 326** Stops the machine movement during the MICRO 4 closing
- M 327** Stops the machine movement during the MICRO 4 opening

During the Micros intervention the axes XYZ quotas are acquired. They're available in the parameters P186, P187, P188 expressed in micron hundredths.

Moreover are acquired every axes quotas that will be used through the M330 M331 functions.

M... MEMORY - DISK FILES MANAGEMENT

The CNC allows the files present in memory management. For this scope are available some M functions:

On the CNC memory is possible to manage only one file at time using the following M functions:

- M 351** Function that allows opening a file on the CNC memory erasing the previous one.
- M 330** Function that stores on the file open in memory every axes configured quotas belonging to the current origin acquired with the apposite function M (M320 :- M327).
- M 331** Function that stores on the open file in memory the absolute quotas regarding every axes configured acquired through the suitable functions M (M320 :- M327).
- M 332** Function that stores on the file open in memory the actual date with hour, minutes and seconds.
- M 350 [file name]** Function that transfers on H/D the file present on CNC memory creating a copy called [file name]. In [file name] has got to be shown also the path. If already exists a file with the same name hit contents will be lost. When the file name isn't present is valid the last one programmed in [.....].

The memory file management is possible only with one file at time; if you would manage more files simultaneously you shall do it directly on the H/D using the following functions M:

- M 357 [file name]** Function that creates on a H/D a file [file name] with 0 bytes length.
Into [file name] has got to be shown also the path.
If already exists a file with the same name hit contents is lost.
When isn't present the file name is valid the last one programmed into [.....].
- M 353 [file name]** Function that stores in the H/D at the file tail [file name] every configured axes quotas acquired with the suitable functions M (M320 :- M321).
- M 354 [file name]** Function that stores in the H/D at the file tail [file name] the actual date with the hour, minutes and seconds.
- M 355 [file name]** Function that stores in the H/D at the file tail [file name] the last string prepared by the Machine Logic.
- M 356 [file name]** Function that stores in the H/D at the file tail [file name] the last numerical variable prepared by the Machine Logic.
- M 392 [file name]** Function that stores on the file [...] on disk the string (...)
- M 393 [file name]** Function that stores on the file [...] on disk the parameter p..
- M 394 [file name]** Function that stores on the file [...] on disk the string (...) and parameter p..

So it is possible to manage on disk different files on every part-program block.

M364 M300 {.....} TOOLS MODIFY FROM A PROGRAM

This function is used to insert the new tool dimensions from a program.

Example of a program with classes, length and radius values modification for the tool T12:

```
%N0  
.....  
.....  
G99 M364 M300 { T12 C10 L123.4 R7.6}  
.....  
.....  
N999999%
```

M382- M385 (.....) MESSAGES FROM A PROGRAM

It is possible the visualisation on the screen of the message write on the program between the parenthesis (.....). This message that cannot exceed 60 characters of length is visualized in red on the space normally used for the errors messages.

The 'M' functions used for this are:

M384 - Enable the visualisation on the screen of the message from a part-program.

M385 - Disable the visualisation of the message from a part-program.

M382 - Enable the visualisation on the screen of the message also for the next blocks.

M383 - Disable the visualisation of the message for the next blocks Is the default condition.

Example of a program:

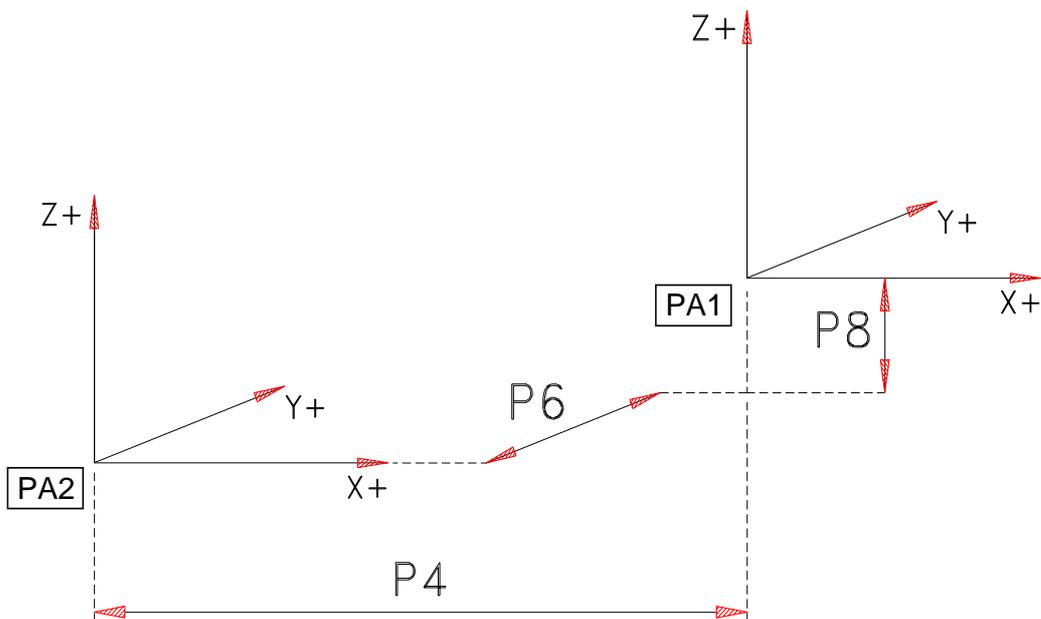
```
%N0  
.....  
G99 M382  
M384 ( INPUT DL VALUE )  
P0=?  
M383  
.....  
N999999%
```

M365 M300 {...} ORIGINS ZEROSET FROM A PROGRAM

Function used to zero setting an origin from a program.

Program example, with PA2 origin absolute quotas reading and new origin PA1 writing:

```
%N0
G99 (PA2 origin absolute quotas reading
P1=PA2X
P2=PA2Y
P3=PA2Z
( P4=distance in X regarding the new origin from PA2
P4=50.25
P5=P1+P4
( P6=distance in Y regarding the new origin from PA2
P6=42
P7=P2+P6
( P8=distance in Z regarding the new origin from PA2
P8=23
P9=P3+P8
( function to write the absolute quotas in the PA1 origin
M365 M300 { PA1 XP5 YP7 ZP9}
.....
N99999%
```



M704-M704/1-M705 M708 M709 TAIL-STOCK MANAGEMENT

It is possible with M704 and M705 to move the tailstock forward or backward.

With the M704 function the tailstock goes forward to reach the piece, automatically lock and switch on the light to signals that the piece is locked.

Vice versa by programming the M705 function the tailstock return backward, release the piece and switch off the light.

If the machine is equipped by the B axis, a switch that signals provides the tailstock to the NC if it is possible to rotate the B axis. In this case, only when the tailstock is backward (switch pressed) will be possible to rotate the B axis.

There are two others functions (M708, M709); with the first (M708) it is possible to disable the locking of the tailstock that normally happen by a manual command or automatic command (M704).

In this case, the tailstock stays against the piece only with the pressure exercised by the air. The M708 function is modal and stays memorized also when we switch off the machine. To return to the normal condition it is necessary to program the M709 function.

It is possible to reduce the tail stock pressure for the push by programming the M704/1 function, in that way the pressure reduces depending to the manual regulation of the adapter presents on the anterior part of the machine.

It is possible to do in two different ways:

- a) By programming the M704 function (tail stock forward with maximal pressure) and successively M704/1 (tail stock forward with reduced pressure)
- b) By programming the M704/1 function (tail stock forward with reduced pressure) and eventually successively M704 (tail stock forward with maximal pressure).

During the pressure commutation, if the tailstock is locked M709 (active), the locking is disabled and automatically re-activated at the end of the procedures.

M728 M729 COOLANT SELECTOR MANAGEMENT

In manual it is possible by using a selector with 5 positions to select the wanted function: M7, M8, M50, M51, M52, M53

From the NC the function is programmed with the appropriate "M " and there are two possibilities:

Rotate the selector over one of the " M " functions, in this way the exit is activated, rotating the selector to " 0 " the exit is disabled.

By programming the M728 function the selector is excluded and it is only necessary to program the wanted " M " function to activated the desired exit.

By programming the M729 function the machine returns to the normal conditions.

This function is modal, therefore must be re-programmed every time is executed the reset of the "navetta" (INS_CBF, INS_UTI). With this function it is possible to use all the functions also if the manual selector is not present.

P.. PARAMETER

As we have already written in the description of the VSP tri-literal commands, it is possible to define a maximum of 100 parameters variables, identified with the P letter followed by an index number from 0 to 99.

It is possible to give a parameter to very functions of the ribbon, so there values will not only be constant, but also variable as the given parameter changes:

Ex: XP10, QYP3, FP8, GP2, PAP80

Where the parameters can have the following contents:

P10=-150.05 P3=2*P4 P8=300 P2=1 P80=4

The value assigned to a selected parameter, is done through the equality:

Pnn = Value (if the Parameter is defined in the VSP table) or
Nnn Pnn = Value or Expression (if the parameter is defined in a program block)

With the expression:

Nnn G99 Pnn = ? The machine stops and wait for user data input (Pnn).

It is possible to continue the program execution by pushing START CN.

With "expression" composed by consistent groups of parameters, constants, mathematical operations and brackets, the whole is also changeable during the program execution.

In the expression which define the parameter values, can also be programmed also the X,Y,Z,A,B,C axes functions, with the limitation to define only a parameter for every ribbon block, without any other machine function excluding the N block number.

Examples: N10 G0 X20 P1=100 ⇒ WRONG
 N10 P1=100 P2=P1*5 ⇒ WRONG

 N10 P1=100 ⇒ CORRECT
 N20 P2=P1*5 ⇒ CORRECT
 N30 XP2 ⇒ CORRECT

This definition block of the parameter is executed together with the following working block pushing on ENTER only one time; this allow to program the definition block of the parameters also inside a profile without stopping the continuity of the work.

It is not possible to do the Block Search on a block of parameter definition.

All the function values, given with the parameters, can be rototranslated and reduced in scale as in the normal programming, while a function, of X, Y, Z, A, B, C type, if it is programmed in one expression, it will keep its last programmed value, which is not altered either with the rotation or with the scale factor.

In the definition of a parameter whose values are known, the following operator sequence presented in priority order:

SIN Sine of a angle in degree and tenth of degree

COS	Cosine of a angle in degree and tenth of degree
TAN	Tangent of a angle in degree and tenth of degree
ASN	Arcsine in degree and tenth of degree between +- $\tilde{\alpha}/2$
ACS	Arc-cosine in degree and tenth of degree between +- $\tilde{\alpha}/2$
ATN	Arc tangent in degree and tenth of degree between +- $\tilde{\alpha}/2$
LOG	Logarithm in base 'e'
LGT	Logarithm in base '10'
EXP	Exponential function 'e^n'
EXG	Exponential function '10^n'
SQR	Square root
^	Raising to power
*	Multiplication
/	Division
NEG	Change of sign
ABS	Absolute value
INT	Whole value cut off par defect
NEI	Whole value more approximate
+	Addition
-	Subtraction
=	Equal

In the valuation of an expression brackets can be used, for a maximum of 9 levels, in order to know the various groups of operations that have the same priority. Inside every level of brackets you can list in priority order, up to 4 operations.

An expression is a computing over an operand series as constant, as parameters, or axes codes:

$$\text{E.g.: } P1 = \text{SQR}(80.5 * (\text{SIN}(-30))) + (P3 - Y / P8 \wedge \text{ATN}2)$$

The written commands can be written either in small letters or in capital letters, while the P, X, Y, etc. commands must be written in capital letters.

It is possible to insert the 'space' character between the commands; a bracket must divide two consecutive commands:

$$P1 = 80.5 * \text{SIN}(-30) \quad \Rightarrow \text{Wrong}$$

$$P1 = 80.5 * (\text{SIN}(-30)) \quad \Rightarrow \text{Correct}$$

The expression is calculated from the left to the right, keeping in mind the priority of the commands and the brackets level.

The representation field of the parameters values goes from $+ -4.19 \times 10^{-307}$ to $+ -1.67 \times 10^{308}$, also if they are displayed with four whole codes and three decimals.

Between the offered possibilities with the parameter programming there are the conditioned jumps, through the ones it is possible to stop the normal sequence of execution of the program block.

The conditioned jumps can be done only inside the same program and it is possible to recall every block, but not the same one.

The operators of relation for the conditioned jumps are the following:

<	- Less
>	- Greater
<=	- Less or equal
>=	- Greater or equal
=	- Equal
!=	- Different

Example:

N100 P2 >= P5 N50 or N100 P2 >= P5 NP10

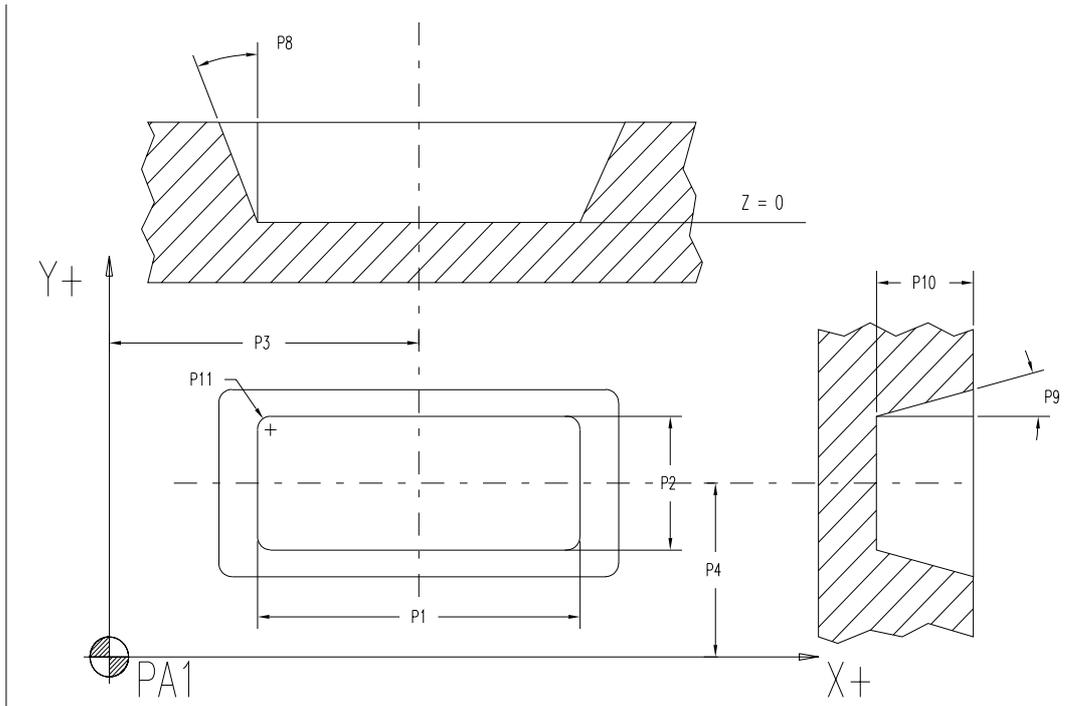
In case the condition is right, and that is the contained value in the P2 parameter is greater or equal to the contained value in the P5 parameter, then, in the first case, we will jump to execute the N50 block, while in the second case, we will jump to execute the block in which number the P10 parameter values are contained.

In case the condition is wrong, the execution will continue with the following block to N100.

The first command must ALWAYS be a parameter, while the second can be a parameter or a constant; if it is none of the two it will take the zero value.

If the second parameter does not exist, then the condition is not verified and during the work execution the machine will stop.

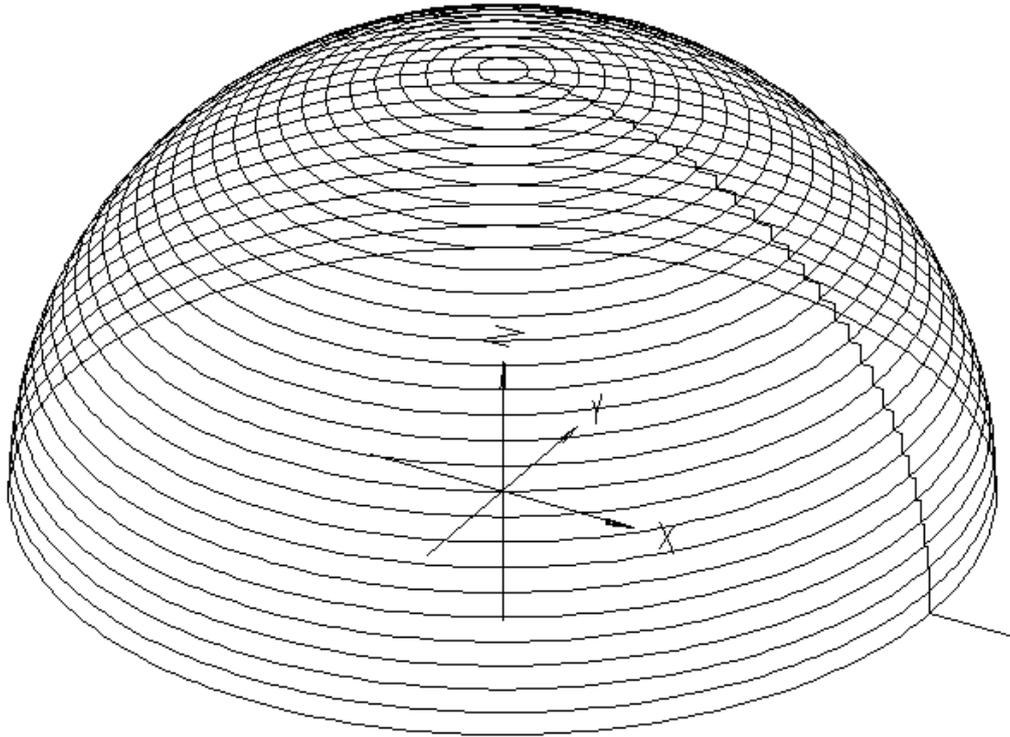
As for the assignment blocks of the parameters values, also this will be executed with the following block of working, pushing the START only one time.



```

%N0
N1 P1=50          ==> POCKET BASE ( X )
N2 P2=30          ==> POCKET HIGH ( Y )
N3 P3=0           ==> X POCKET CENTRE
N4 P4=0           ==> Y POCKET CENTRE
N5 P5=0           ==> ROTATION ANGLE
N6 P6=10          ==> NUMBER OF PASSES
N7 P7=0           ==> START Z
N8 P8=15          ==> TAPER
N9 P9=10          ==> TAPER
N10 P10=20        ==> END Z
N11 P11=8         ==> RADIUS
N12 G45T1
N13 G51XP3YP4EP5PA1
N14 G0X0Y0S1300F650
N15 G0Z2M13
N16 P20=P10/P6    ==> Z INCREMENT
N17 G1ZP7
N18 P25=(P1/2)+(P7*(TAN(P8)))
N19 P26=(P2/2)+(P7*(TAN(P9)))
N20 P27=-P25
N21 P28=-P26
N22 G41R1D..
N23 G13XP25Y0E90R0
N24 G21RP11
N25 G13X0YP26E180R0
N26 G21RP11
N27 G13XP27Y0E270R0
N28 G21RP11
N29 G13X0YP28E0R0
N30 G21RP11
N31 G13XP25Y0E90R0
N32 G40X0Y0R1
N33 P7=P7+P20
N34 P7<P10N17
N35 G0Z50
N36 G47
N9999%

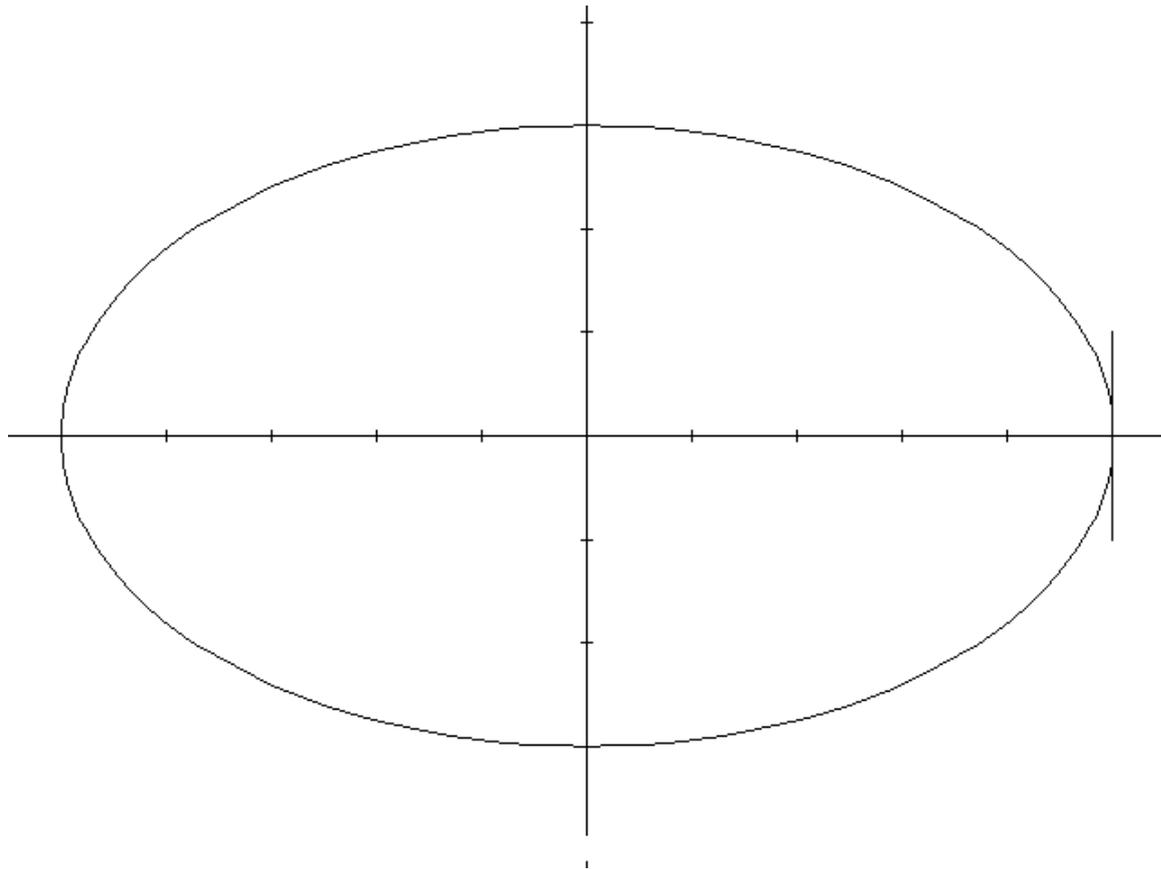
```



```

%N0
N1  P1=50          ==> SPHERE RADIUS
N2  P2=5           ==> ANGULAR INCREMENT
N3  P3=0           ==> STARTING ANGLE
N4  G45T1
N5  G0X60Y0PA1
N6  G99F100S1200M3
N7  G0Z2
N8  P10=P1*SINP3
N9  P11=P1*COSP3
N10 G1ZP10
N11 G1XP11Y0
N12 G3XP11Y0I0J0
N13 P3=P3+P2
N14 P3<90N8
N15 G47
N99999%

```



```

%N0
N1 P1=50           ==> LONGER SEMI-AXES
N2 P2=30           ==> SHORTER SEMI-AXES
N3 P3=5            ==> ANGULAR INCREMENT
N4 P9=0            ==> STARTING ANGLE
N5 G45T1
N6 G0X60Y-10PA1
N7 G99F100S1200M3
N8 G0Z2
N9 G1Z-1
N10 G42XP1Y-10D4
N11 P10=P2*SINP9
N12 P11=P1*COSP9
N13 G1XP11YP10
N14 P9=P9+P3
N15 P9<=360 N11
N16 G1XP1Y10
N17 G40X60Y10
N18 G0Z2
N19 G47
N99999%

```

S S/ FUNCTIONS

With the S function is programmed the spindle revolutions.

Basing on the programmed number if the machine is equipped with more rotation ranges, the right one is automatically selected and inserted (M40).

During the work cycles, the CN keep under constant control the current absorption percentage of the spindle motor.

This percentage is visible on the screen. If the absorption percentage is over the 98% for an equal time to the one imposed in spindle axis configuration, the CN reduce the work speed of 30% on the one programmed. If the absorption doesn't reduce of 98% the CN stop the machine and an error appear.

To keep under control an absorption value less than 98% it is sufficient to program: S (revolutions numb)/(absorption percentage).

Example: S2000/15 = the spindle speed is 2000 rpm, the reduce speed mechanism will be activated if the absorption percentage exceed the 15% for an equal time to the one imposed in axis spindle configuration.