FAGOR 8020 T
Programming Manual

FAGOR

SOF 709

FAGOR 8020 T

PROGRAMMING MANUAL

C O N T E N T S

		Page
1.	**INTRODUCTION**	7
1.1.	External programming	8
1.2.	DNC Connection	9
1.3.	FAGORDNC communication program	9
2.	**CREATING A PROGRAM**	10
3.	**PROGRAM FORMAT**	10
3.1.	Parametric programming	11
4.	**PROGRAM NUMBERING**	11
5.	**PROGRAM BLOCKS**	11
5.1.	Block numbering	11
5.2.	Conditional blocks	12
6.	**PREPARATORY FUNCTIONS**	13
6.1.	Table of G functions used in the CNC 8020 T	13
6.2.	Types of movement	15
6.2.1.	G00. Positioning	15
6.2.2.	G01. Linear interpolation	16
6.2.3.	G02/G03. Circular interpolation	17
6.2.3.1.	Circular interpolation in cartesian coordinates by programming the radius of the arc	19
6.3.	G04. Dwell	21
6.4.	Transition between blocks	21
6.4.1.	G05. Round corner	21
6.4.2.	G07. Square corner	22
6.5.	G08. Circular path tangent to previous path	23
6.6.	G09. Circular path programmed by two points (3 point arc definition)	25

6.7.	G14, G15,G16. C axis programming	27
6.8.	Unconditional calls/Jumps	34
6.9.	G31-G32. Storage and retrieval of part program's datum point	35
6.10.	G33. Threadcutting	37
6.11.	G36. Controlled corner rounding	41
6.12.	G37. Tangential entry	43
6.13.	G38. Tangential exit	45
6.14.	G39. Chamfering	47
6.15.	Tool radius compensation	48
6.15.1.	Selection and initiation of tool radius compensation	52
6.15.2.	Operating with tool radius compensation	55
6.15.3.	Tool radius compensation freeze with G00	59
6.15.4.	Cancellation of tool radius compensation	59
6.16.	G50. Loading the tool offset table	62
6.17	G51. Changing the engaged tool dimensions	63
6.18.	G53-G59. Zero offsets	64
6.19.	Units of measurement	67
6.20.	G72. Scaling	67
6.21.	G74. Machine-reference search	68
6.22.	G75. Probing	69
6.22.1.	Probing canned cycles	70
6.23.	G76. Automatic block generation	83
6.24.	G90. Absolute programming. G91. Incremental programming	89
6.25.	G92. Preselection of coordinate values and setting of max. S value at constant surface speed G96	91
6.26.	G93. Preselection of polar origin	92
6.27.	G94. Feedrate F in mm/min (inches/min)	93

6.28.	G95. Feedrate **F** in mm/rev (inches/rev)	93
6.29.	G96. S speed in m/min (inch/min.) at constant surface speed	93
6.30.	G97. S speed in rev./minute	94
7.	**PROGRAMMING OF COORDINATE VALUES**	95
7.1.	Cartesian coordinates	95
7.2.	Polar coordinates	96
7.3.	Two angles	99
7.4.	One angle and one cartesian value	100
8.	**F. FEEDRATE PROGRAMMING**	102
9.	**S. SPINDGLE SPEED AND ORIENTATION**	103
10.	**T. TOOL PROGRAMMING**	104
11.	**M. MISCELLANEOUS FUNCTIONS**	107
11.1.	M00. Program stop	107
11.2.	M01. Conditional stop of program	107
11.3.	M02. End of program	108
11.4.	M30. End of program with return to beginning	108
11.5.	M03. Clockwise start of the spindle	108
11.6.	M04. Counter-clokwise start of the spindle	108
11.7.	M05. Spindle stop	108
11.8.	M19. Spindle orientation	108
11.9.	M41,M42,M43,M44 Spindle range selection	109
11.10.	M45. Speed of live/synchronized tool	109

12.	**STANDARD AND PARAMETRIC SUBROUTINES**	111
12.1.	Identification of a standard subroutine	112
12.2.	Calling on a standard subroutine	112
12.3.	Parametric subroutines	113
12.3.1.	Identification of a parametric subroutine	113
12.4.	Calling on a parametric subroutine	114
12.5.	Levels of nesting	114
13.	**PARAMETRIC PROGRAMMING. OPERATIONS WITH PARAMETERS**	115
14.	**CANNED CYCLES**	132
14.1.	G66. Pattern repeating	133
14.2.	G68. Stock removal on the X axis	137
14.3.	G69. Stock removal along the Z axis	140
14.4.	G81. Canned turning cycle of straight sections	143
14.5.	G82. Canned facing cycle of straight sections	145
14.6.	G83. Deep hole drilling cycle	147
14.7.	G84. Turning cycle with arcs	149
14.8.	G85. Facing cycle with arcs	151
14.9.	G86. Threadcutting cycle (standard)	153
14.10.	G87. Threadcutting cycle (frontal)	155
14.11.	G88. Grooving cycle along X axis	157
14.12.	G89. Grooving cycle along Z axis	159
15.	**ERROR CODES**	161

1. INTRODUCTION

The CNC 8020 T control can be programmed both from its front panel and from external peripherals (tape reader, cassette reader/recorder, computer, etc.).

The memory capacity availabe for part programming is 32K.

In this CNC the part programs can be entered in four different operating modes:

2 - PLAY BACK
3 - TEACH IN
6 - EDITING
7 - PERIPHERALS

In mode 7, the programs are transferred to the CNC from any external peripheral (RS 232 C). In the other modes, the programs are entered directly from the front panel of the CNC.

This means that programming can be carried out both at the machine and at a remote location; e.g., in a programming office.

In the **PLAY BACK** mode, the axes are shifted manually (Jog) and the coordinates reached are then entered as the program coordinates.

In the **TEACH IN** mode, a block is written and executed and then entered as part of the program.

In the **EDITING** mode, the complete program is recorded and then executed.

1.1. External programming

If the programming is to be carried out by means of an external peripheral, ISO code must be used.

% will initiate the program, followed by the program number (five digits), followed by the characters, **R**e**T**urn or **L**ine**F**eed and the **N** of the first block). The characters preceeding % will be disregarded. **RT** or **LF** must be used after each block prior to the **N** of the beginning of the following block.

To end the program the characters **ESC**ape (ESC) or **E**nd **O**f **T**ape (EOT) or a series of 20 nul characters (**ASCII 00**) must be used.

Comments to be displayed on the CRT must be written between brackets ( ). (43 characters max., including the brackets).

NOTE:

To assure the display of the comments:

- The block must contain an M,S,T or movement function.
- Its execution time must be greater than 200 ms.

1.2. DNC connection

Every CNC FAGOR DNC 8020 offers as standard service, the possibility of working with DNC (Distributed numerical control), enabling the communication between the CNC and a computer to carry out the following functions:

- . Directory and deletion commands
- . Program and table transfers between the CNC and a computer
- . Execution of infinite program
- . Machine remote control
- . Ability to supervise the status of the avanced DNC systems

1.3. The FAGORDNC Communication Program

Commercialized in a 5" flexible diskette is an application for the connection of **FAGOR 8020** numerical controls to an **IBM** or **COMPATIBLE** computer, using the DNC incorporated in those controls.

Several CNC can be connected to the DNC through the RS 232 lines of these computers.

The operation mode is interactive, with **MENUS** which guide the user and simplify the use of this program. The computer is used as a part-programs centralized **STORAGE**, avoiding the use of awkward puncher tapes. This simplifies the upgrading of versions, enables to make safety copies, listing and edition of part programs with inclusion of comments...

The DNC connection manual and the **FAGORDNC** communication can be requested at this address. The delivery time: immediate.

2. CREATING A PROGRAM

The machining program must be entered in a format acceptable to the CNC.

It must include all the geometrical and technological data required for the machine-tool to perform the required functions and movements.

A program is built up in the format of a sequence of blocks.

Each programming block consists of:

N : Block number
G : Preparatory functions
C,X,Z: Coordinate values
F : Feedrate
S : Spindle speed
T : Tool number
M : Auxiliary functions

This order has to be maintained within each block, although each block might not necessarily contain all these items.

3. PROGRAM FORMAT

The CNC 8020 T can be programmed metrically (in mm) or in inches.

Metric format (in mm):

P(%)5 N4 G2 X+/-4.3 Z+/-4.3 I+/-4.3 K+/-4.3
R+/-4.3 A+/-4.3 P+/-5.4 F4 S4 T2.2 M2

Format in inches:

P(%)5 N4 G2 X+/-3.4 Z+/-3.4 I+/-3.4 K+/-3.4
R+/-3.4 A+/-4.3 P+/-5.4 F4 S4 T2.2 M2

NOTE: +/- 4.3 Means that a positive or negative figure with up to four digits to the left of the decimal point and three to the right may be programmed (e.g. -256.7895).

4 Means that only a positive integer (no decimals) of up to four digits may be programmed.

2.2 Means that only a positive value of up to two digits to the left and two to the right of the decimal point may be programmed.

A Must always be programmed in degrees.

Maximum value: +/-8028,607
Resolution : 0.001 degrees

3.1. Parametric programming

It is also possible to program in a block whatever function by parameters, except the program number, the block number, G22 and G23 functions, in such a way that, when executing the block, the function takes the current value of the parameter. Combinations of fixed values and parameters can be programmed in the same block, e.g.:

N4 GP36 X37.5 ZP13 FP10 S1500 TP4.P4 MP2.

The CNC has 100 parameters (P00/P99).

4. PROGRAM NUMBERING

Every program must be numbered between 0 and 99998.

This number must be entered at the beginning of the program before the first block.

If the program is entered from an external peripheral, the symbol % is used, followed by the number required and the pressing of **LF** or **RETURN** or both, followed by the **N** of the first block.

5. PROGRAM BLOCKS

5.1. Block numbering

The block number is used to identify each of the blocks that make up a program.

The block number consists of the letter N followed by a figure between 0 and 9999.

This number must be written at the start of each block.

BLock may be given any number between 0 and 9999 except that no block may be given a lower number than the blocks preceding it in the program.

It is advisable to avoid giving blocks consecutive numbers, so that new blocks can be inserted where required.

If the CNC is programmed from its front panel, blocks are automatically numbered in steps of 10. The automatic numbering can be manually altered.

5.2. Conditional blocks

The are two types of conditional blocks:

a) N4 STANDARD CONDITIONAL BLOCK

If next to the block number N4 (0-9999), a decimal point (.) is written, the block is characterized as a conditional block. That means that the CNC will execute it only if the relevant external signal (conditional input) is activated.

During any program execution, the CNC reads four blocks ahead the one being executed; so the external signal is to be activated at least during the execution of the fifth block previous to the conditional block, for its execution to be carried out.

b) N4 SPECIAL CONDITIONAL BLOCK

If next to the block number N4, two decimal points (..) are written, the block is characterized as a special conditional block, in other words, the CNC will execute it only if the relevant external signal (conditional input) is activated.

In this case, it is enough to activate the external signal (conditional input) during the execution of the block previous to the special conditional block, for its execution to be carried out.

The **N4..**, special conditional block, cancels G41 or G42 radius compensation.

6. PREPARATORY FUNCTIONS

The preparatory functions are programmed by means of the letter G followed by one or two digits (G2).

They are always programmed at the start of the block and are used to determine the geometry and operating state of the CNC.

6.1. Table of G functions used in the CNC 8020 T

(Modal)	G00	: Rapid positioning
(Modal)	G01*	: Linear interpolation
(Modal)	G02	: Clockwise circular interpolation
(Modal)	G03	: Counter-clockwise circular interpolation
	G04	: Dwell, duration programmed by means of K
(Modal)	G05	: Round corner
(Modal)	G07*	: Square corner
	G08	: Arc tangent to previous path
	G09	: Arc programmed by two points (3 point arc definition)
(Modal)	G14	: Activate C axis in degrees
(Modal)	G15	: Machining of the cylindrical surface of the part
(Modal)	G16	: Machining of the face of the part
	G20	: Call for standard subroutine
	G21	: Call for parametric subroutine
	G22	: Definition of standard subroutine
	G23	: Definition of parametric subroutine
	G24	: End of subroutine
	G25	: Unconditional jump
	G26	: Conditional jump/call if it equals zero
	G27	: Conditional jump/call if it differs from zero
	G28	: Conditional jump/call if it is smaller
	G29	: Conditional jump/call if it is equal or greater
	G30	: Display error code defined by K
	G31	: Store present program's datum point
	G32	: Retrieve datum point stored by G31
(Modal)	G33	: Threadcutting
	G36	: Controlled corner rounding
	G37	: Tangential approach
	G38	: Tangential exit
	G39	: Chamfering
(Modal)	G40*	: Cancellation of radius compensation
(Modal)	G41	: Left hand radius compensation
(Modal)	G42	: Right hand radius compensation
	G50	: Loading of tool offsets by program
	G51	: Modification of offsets of engaged tool
(Modal)	G53-G59	: Zero offsets

	G66	:	Pattern repeating
	G68	:	Stock removal (X)
	G69	:	Stock removal (Z)
(Modal)	G70	:	Programming in inches
(Modal)	G71	:	Programming in millimeters
	G72	:	Scaling
(Modal)	G74	:	Automatic search for machine reference
	G75 N2	:	Probing
	G76	:	Automatic block generation
	G81	:	Canned turning cycle with straight sections
	G82	:	Canned facing cycle with straight sections
	G83	:	Deep hole drilling
	G84	:	Turning with arcs
	G85	:	Facing with arcs
	G86	:	Longitudinal threadcutting cycle
	G87	:	Face threadcutting cycle
	G88	:	Grooving cycle (X)
	G89	:	Grooving cycle (Z)
(Modal)	G90*	:	Programming of absolute coordinates
(Modal)	G91	:	Programming of incremental coordinates
	G92	:	Preselection of coordinates and setting of max. S value
	G93	:	Preselection of polar origin
(Modal)	G94	:	Feedrate F in mm/min (inch/min.)
(Modal)	G95*	:	Feedrate F in mm/rev (inch/rev.)
(Modal)	G96	:	Speed S in m/min (feet/min.) (Constant surface speed)
(Modal)	G97*	:	Speed S in rev/min.

NOTE:

a) (Modal) means that once the **G** functions have been programmed they remain active until cancelled by another G which is incompatible or by **M02**, **M30**, **EMERGENCY** or **RESET**.

b) The **G** functions marked * are those wich the CNC assumes on being turned on or after executing **M02** or **M30** or after an **EMERGENCY** or **RESET**.

All the **G**'s required may be programmed in any order in the same block, except **G20,G21,G22,G23,G24,G25,G26,G27,G28,G29,G30,G31,G32,G50,G53-G59, G72,G74** and **G92** which have to be alone in a block.

If incompatible G functions are programmed in the same block, the CNC assumes the one programmed last.

6.2. Types of movement

6.2.1. G00. Rapid positioning

The movements programmed following G00 are executed at a rapid feedrate set, during the final adjustment of the machine, by the machine-parameter P25 and P65.

There are two different ways of movement in G00, depending on the value applied to the machine-parameter P101(3).

a) **Path not controlled P101(3)=0**

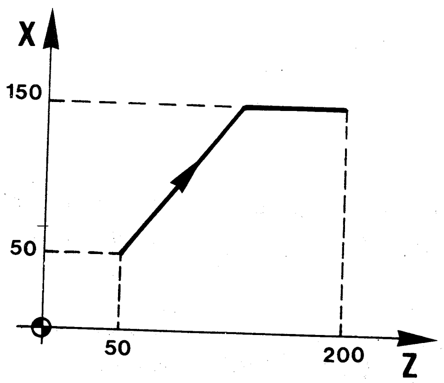
The value of rapid feedrate is independent for each axis, thus the path is not controlled when more than one axis move at the same time.

b) **Vectored G00 P101(3)=1**

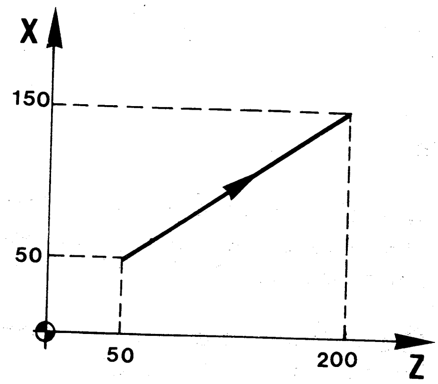
When both axes move simultaneously, the resultant path is a straight between the initial and the final points.

X axis programming in diameters: N4 G00 G90 X300 Z200

a) P101(3)=0



b) P101(3)=1



When programming G00, the last F programmed is not cancelled, e.i., when G01,G02 or G03 is programmed again, the mentioned F is recovered.

In G00 movements the machine-parameter P4 can be used to identify whether the feedrate override knob operates between 0% and 100% or is frozen at 100%

The code G00 freezes the tool radius compensation G41,G42. In other words, when operating with G41 or G42 and G00 being programmed, the radius compensation is without function until G01,G02 or G03 is again programmed.

The code G00 is modal and incompatible with G01,G02,G03 and G33.

The function G00 can be programmed with G or G0.

6.2.2. G01. Linear interpolation

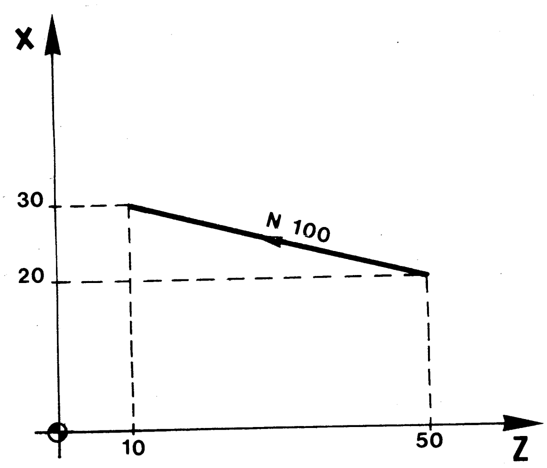
The movements programmed after G01 are performed in a straight line at the feedrate F programmed.

When the two axes move simultaneously, the resulting path is a straight line between the initial point and the final point.

The machine moves along that path at the programmed feedrate F.

The CNC calculates the feedrates of each axis so that the feedrate of the resulting path is the programmed F.

Example: X axis in diameters. Initial point X40 Z50



N100 G90 G01 X60 Z10 F300

The knob on the front panel of the CNC can be used to vary the programmed feedrate F between 0% and 120% or between 0% and 100% depending on P94(3).

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

Function G01 can be programmed as G1.

The CNC assumes the code G00 on being turned on, after executing **M02/M30**, after an **EMERGENCY** and after a **RESET**.

6.2.3. G02/G03. Circular interpolation

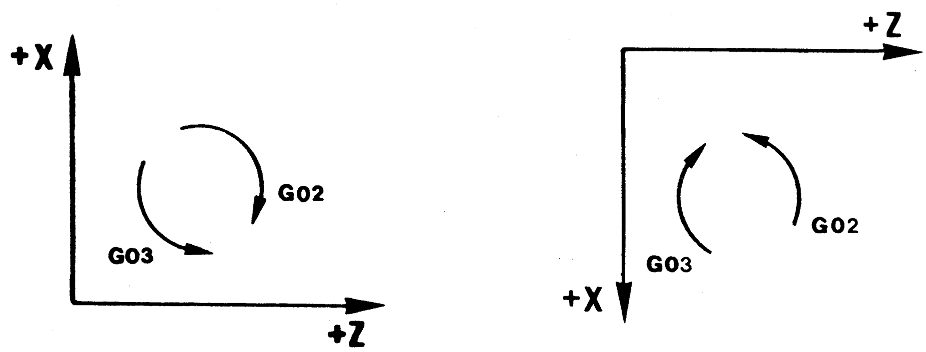
G02: Clockwise circular interpolation.
G03: Counter-clockwise circular interpolation.

The movements programmed following G02/G03 are performed in a circular path at the programmed feedrate F.

The definitions of clockwise (G02) and counter-clockwise (G03) have been fixed according to the system of coordinates depicted below:

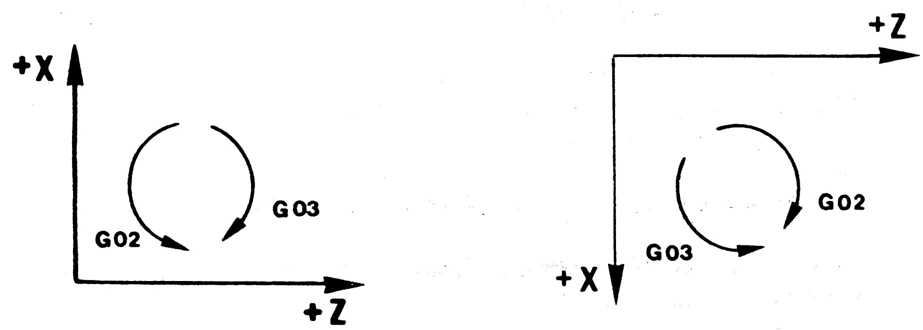
a) Parameter P94(1)= 0

Direction of the machine axes



b) Parameter P94(1) = 1

Direction of the machine axes



The functions G02/G03 are modal and incompatible with each other as well as with G00,G01 and G33.

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

The functions G02/G03 can be programmed as G2/G3.

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

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

```
N4          : Block number
G02 (G03)   : It defines the interpolation
X+/-4.3     : Coordinate value of the arc's final point along the
              X axis.
Z+/-4.3     : Coordinate value of the arc's final point along the
              Z axis.
I+/-4.3     : Distance from the arc's starting point to the center
              along X axis.
K+/-4.3     : Distance from the arc's starting point to the center
              along Z axis.
I,K         : Must be programmed with a sign. They must be
              programmed even when their value is 0.
```

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

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

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

NOTE:

When a circular interpolation is programmed in G02 or G03, the arc's center is taken as the new polar origin.

Even when the **X** axis is programmed in diameters, I is always programmed in radius.

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

The format is as follows:

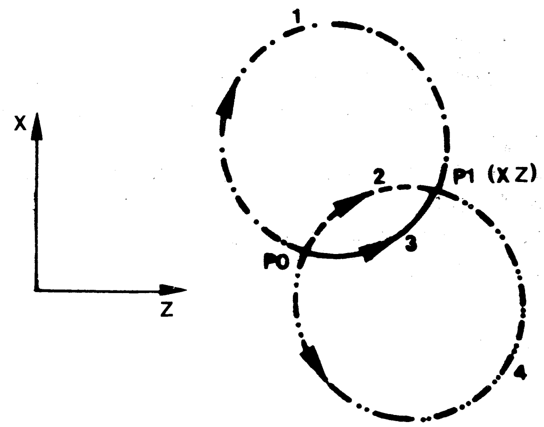
In mm: G02(G03) X+/-4.3 Z+/-4.3 R+/-4.3
In inches: G02(G03) X+/-3.4 Z+/-3.4 R+/-3.4

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

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

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

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



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

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

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

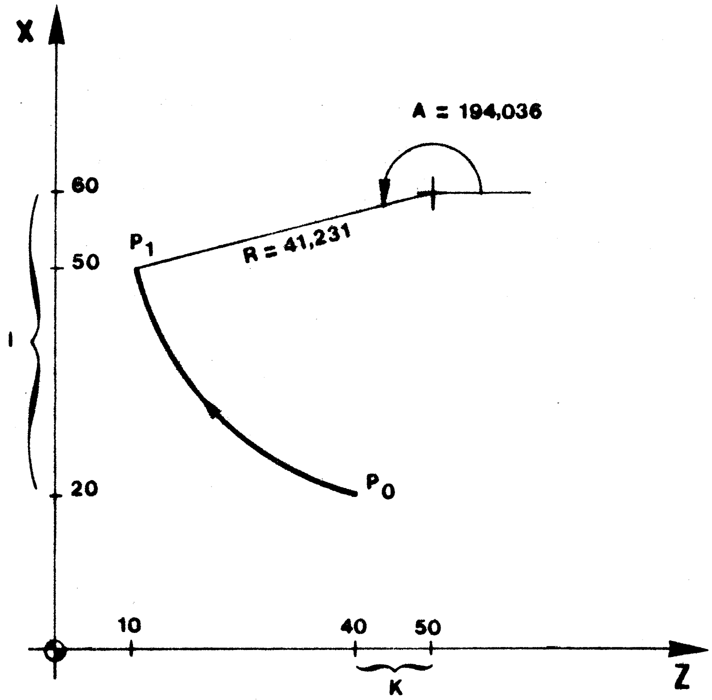
XZ being the final point in cartesian coordinates.

NOTE:

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

Examples: Let us suppose that the programming is in absolute coordinate values (G90) and the X axis one is in diameters.

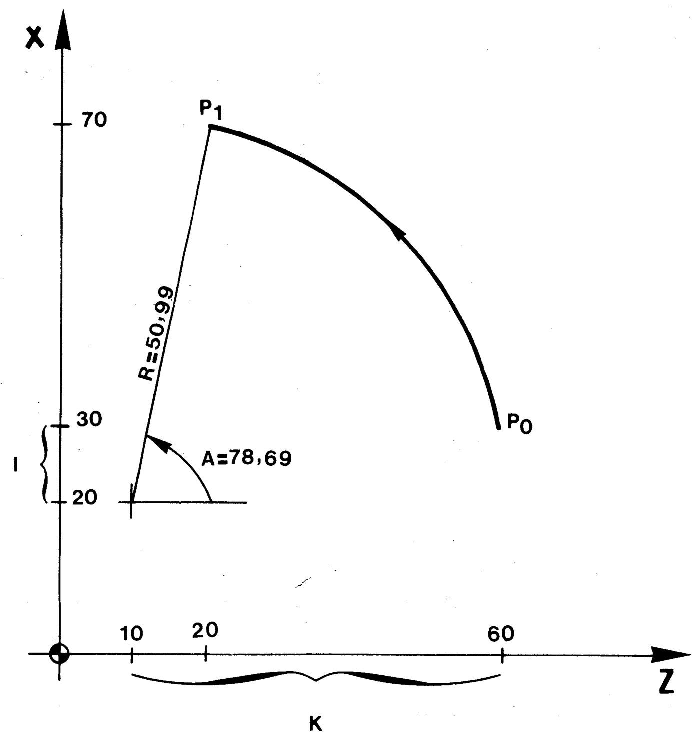
1.



Starting point P0 (X40 Z40)

Cartesian coordinates: N4 G02 X100 Z10 I40 K10
Polar coordinates : N4 G02 A194.036 I40 K10
Radius programming : N4 G02 X100 Z10 R41.231

2.



Starting point P0 (X60 Z60)

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

6.3. G04. Dwell

Function G04 can be used to program a dwell.

The dwell value is programmed by means of the letter K.

```
Example: G04  K0.05  Dwell of 0.05 seconds
         G04  K2.5   Dwell of 2.5 seconds
```

If the value of **K** is programmed with a number, its maximum value will be 99.99. But if programmed by parameter **(K P2)**, its maximum value will be 655.35

The dwell is executed at the start of the block in which it is programmed.

Function G04 can be programmed as G4.

6.4. Transition between blocks

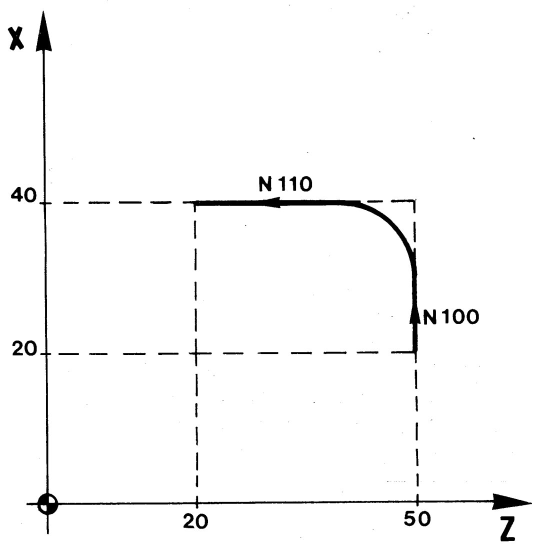
6.4.1. G05. Round corner

When operating on G05, the CNC starts to execute the next block of the program as soon as the deceleration of the axes programmed in the previous block begins.

In other words, the movements programmed in the next block are executed before the machine has reached the exact position programmed in the previous block.

Example:

X in diameters. The starting point is X40 Z50




```
N100 G90 G01 G05 X80
N110             Z20
```

As can be seen in the example, the edges would remain rounded in the case of two mutually perpendicular movements.

The difference between the theoretical and actual profiles is a function of the feedrate value.

The faster the feedrate, the greater the difference between the theoretical and actual profiles.

Function G05 is modal and incompatible with G07.

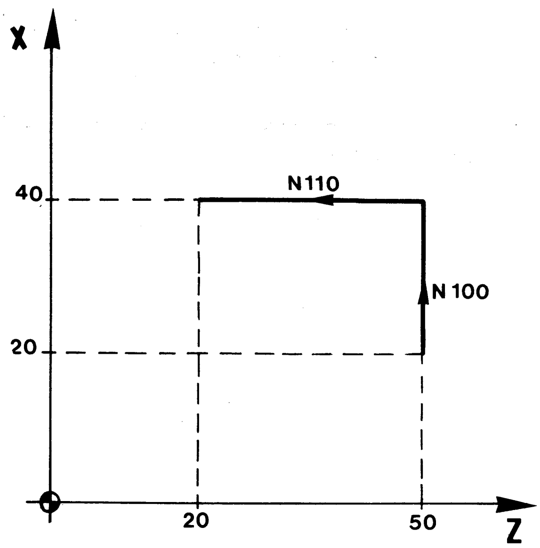
Function G05 can be programmed as G5.

6.4.2. G07. Square corner

When operating on G07, the CNC does not execute the next block of the program until the exact position programmed in the previous block has been reached.

Example:

X in diameters. The starting point is X40 Z50




```
N100 G90 G01 G07 X80
N110             Z20
```

The theoretical and actual profiles coincide.
Function G07 is modal and incompatible with G05.
Function G07 can be programmed as G7.
The CNC assumes function G07 on power **ON** and after M02,M30,**EMERGENCY** or **RESET**.

6.5. G08. Circular path tangent to previous path

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

Format with cartesian coordinates:

N4 G08 X+/-4.3 Z+/-4.3

N4 : Block number

G08 : Code defining circular interpolation tangent to previous path.

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

Format with polar coordinates:

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

N4 : Block number

G08 : Code defining circular interpolation tangent to previous path

R+/-4.3: Radius (referred to polar origin) of the arc's
R+/-3.4 final point.

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

Example:

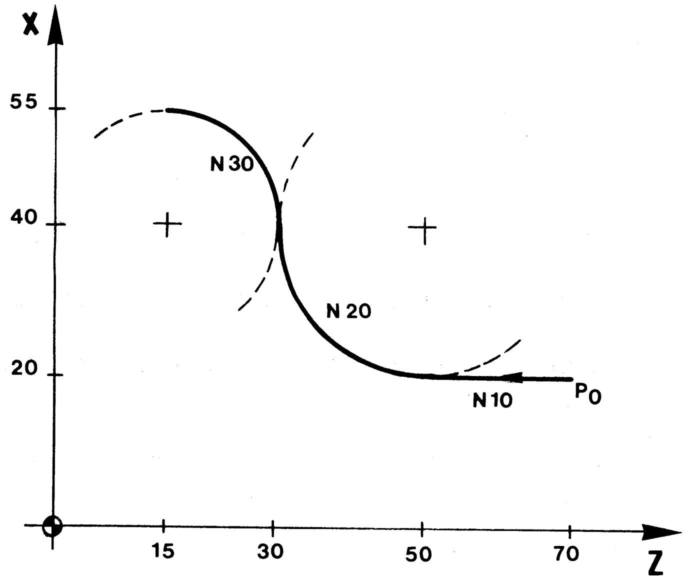
X in diameters

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

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

Its programming may be:

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



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

When G08 is not used, the programming will be:

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

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

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

The function G08 replaces G00,G01,G02 and G03 only in the block in which it is written.

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

6.6. G09. Circular path programmed by two points (3 point arc definition)

Two points (the final plus one intermediate point) are sufficient to program an arc provided that the current position is the starting point. In other words, an intermediate point is programmed instead of the center.

This way of programming an arc is particularly useful when copying a part in **PLAY BACK** mode. After writing G09, **JOG** the axes to the intermediate point and press **ENTER**, then **JOG** again to the final point and press **ENTER** and the block will be recorded.

Cartesian coordinates

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

N4 : Block number.
G09: Code identifying 3 point arc definition.

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

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

I+/-4.3: Coordinate values of the arc's intermediate point

K+/-4.3: Coordinate values of the arc's intermediate point

Polar coordinates

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

N4 : Block number.

G09 : Code identifying 3 point arc definition.

R+/-4.3: Radius (referred to polar origin) of the final point of the arc.

A+/-3.3: Angle of the final point of the arc.

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

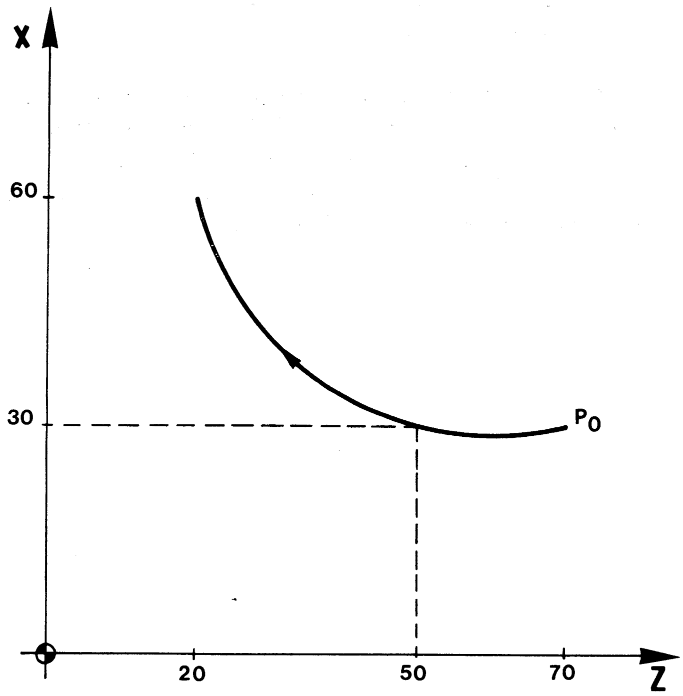
K+/-4.3: Y value of the intermediate point.

(The intermediate point must always be programmed in cartesian coordinates).

Example:

X in diameters. Starting point X60 Z70. Final point X120 Z20

N4 G09 X120 Z20 I60 K50



G09 is not modal. It's not necessary to program the direction of the movement (G02,G03) when programming G09.

Function G09 replaces G00,G01,G02 and G03 only in the block in which it is written.

NOTE:

A complete circle cannot be executed by means of G09, i.e. The arc's initial and final point must be different. Otherwise error 40 will be generated.

6.7. G14,G15,G16. C axis programming

- G14. Activate C axis in degrees.
- G15. Machining of the cylindrical surface of the part (main plane C,Z)
- G16. Machining of the face of the part (Main plane C,Z)

Once the typical turning operations are completed, other operations, like the milling of the cylindrical surface and/or the face of the part can be done with the CNC 8020 T since it can control the machine's main leadscrew carrying out linear interpolations between the C,X,Z axes.

G14. ACTIVATE THE C AXIS IN DEGREES

General considerations:

. Programming G14, the positioning of the C axis can be controlled if machine-parameter P102(2)=1.

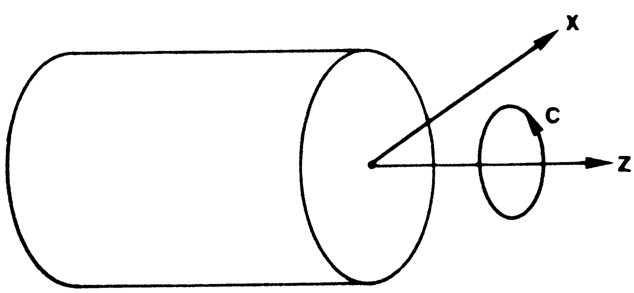
. G14 must be programmed alone in a block.

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

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

. When programming G14, G95 and G96 are cancelled.

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



To program the character **C** from the front panel, the * key must be pressed.

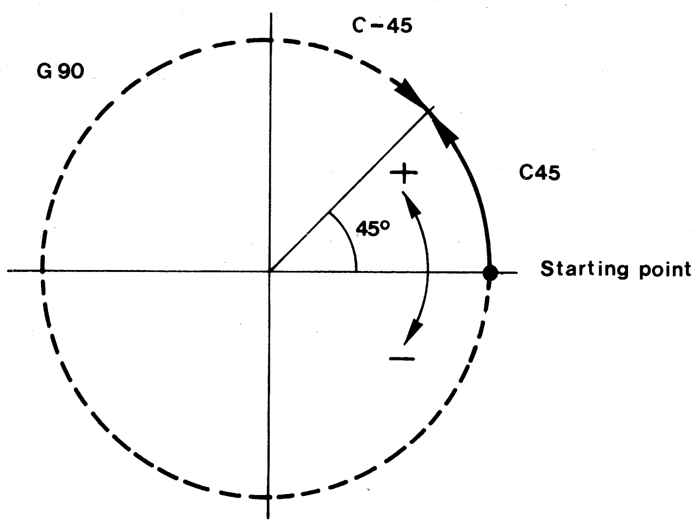
The **C** axis movement must be programmed in degrees and the feedrate **F4** in degrees/minute.
The programming format is the following:

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

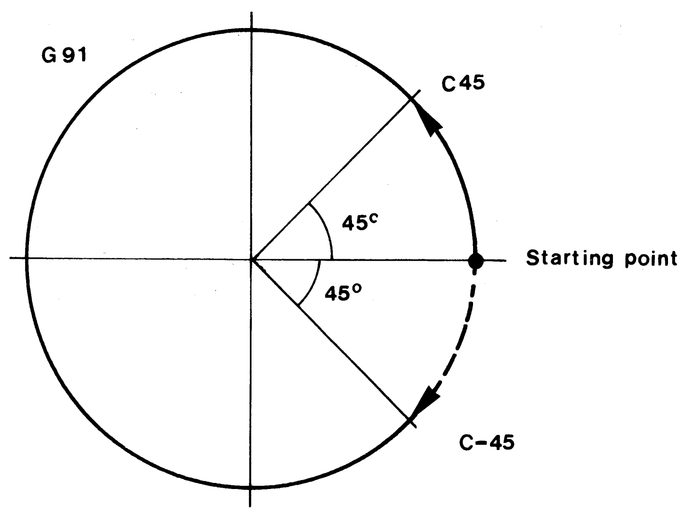
When being G14 activated, the following block is executed:
N4 G91 G01 C720 F500

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

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



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



To program the character **C** from the front panel, the * key must be pressed.

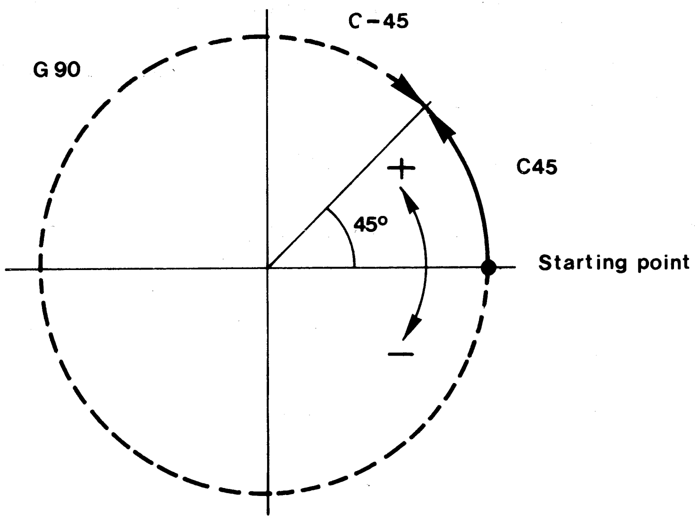
The **C** axis movement must be programmed in degrees and the feedrate **F4** in degrees/minute.
The programming format is the following:

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

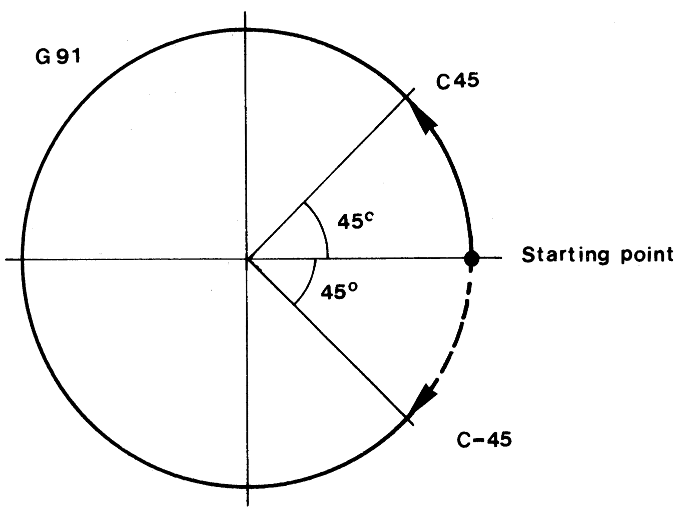
When being G14 activated, the following block is executed:
N4 G91 G01 C720 F500

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

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



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



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

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

General considerations for the programming of both functions:

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

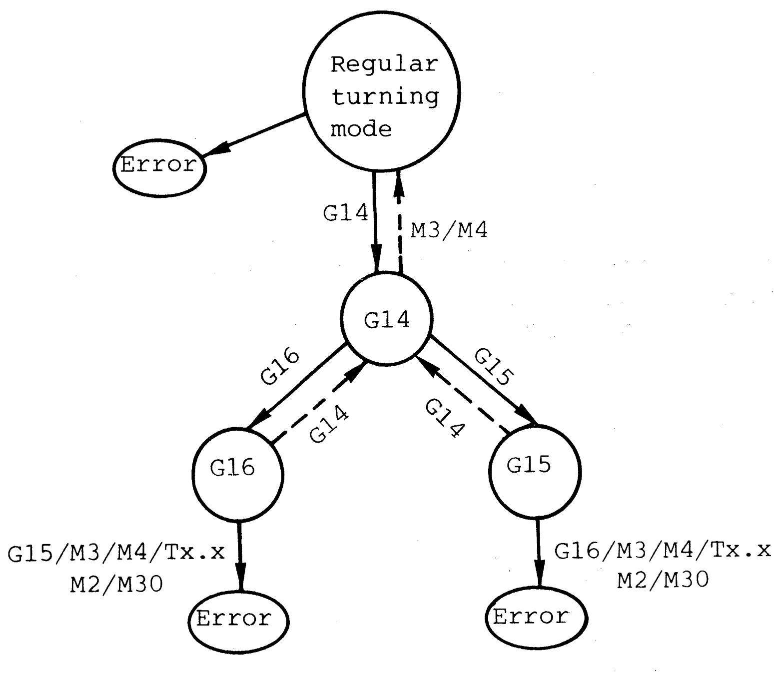
. By programming either G15 or G16, the tool radius compensation G41/G42 are cancelled.

. G15 and G16 cancel functions G95 and G96.

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

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

. To cancel G15 or G16, program G14.

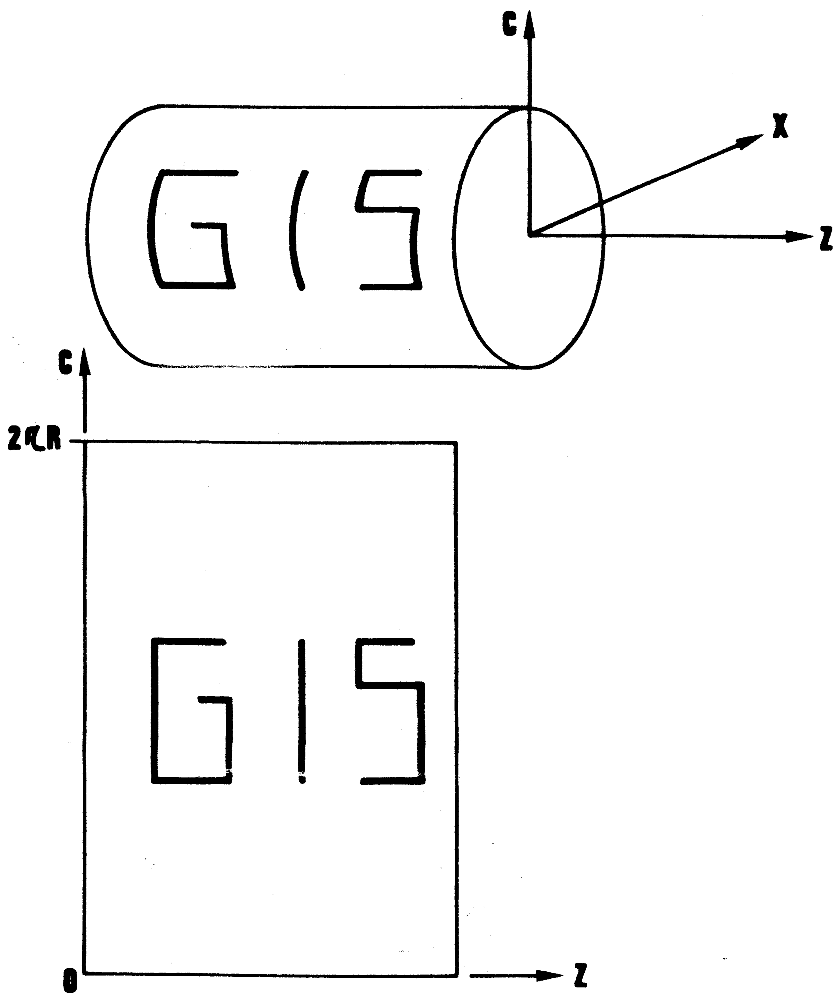


When working with G15 or G16, the C axis must be treated as linear axis and the movements must be programmed in mm or inches. The CNC will convert them into degrees.

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

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

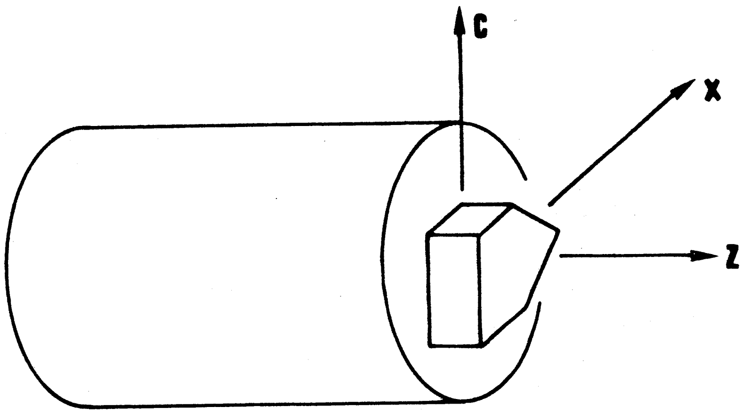
The origin point of the defined plane is the one corresponding to the C axis machine-reference point.

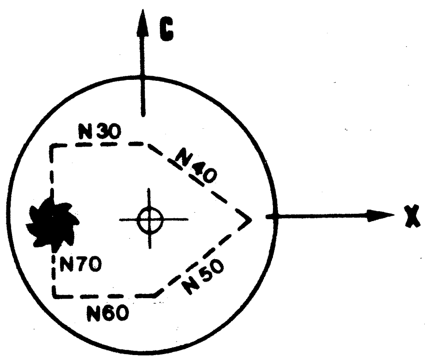


G16. MACHINING OF THE FACE OF THE PART (main plane Z X)

Example: Let's suppose that the coordinates are absolute (G90)

```
N --  --
N30  X0
N40  C0    X14
N50  C-10  X0
N60  X-13
N70  C10
N--  --
```





6.8. G25. Unconditional jump/call

The function G25 can be used to jump to another block of the current program.

There are two possibilities:

a) N4 G25 N4

```
N4  - Block number
G25 - Code for unconditional jump
N4  - Number of the block the jump is aimed at
```

When the CNC reads this block, it jumps to the targeted block and the program continues.

```
Example: N0   G00  X100
         N5             Z50
         N10  G25  N50
         N15       X50
         N20       Z70
         N50  G01  X20
```

When the block 10 is reached, the CNC jumps to block 50 and then the program continues until it is finished.

b) N4 G25 N4.4.2

```
N4      -> Block number
G25     -> Code unconditional jump
N4.4.2 -> Number of repetitions
 | |----> Number of the last block to be executed
 |------> Number of the block to which the jump is targeted
```

When the CNC reads such a block, it jumps to the block identified between the N and the first decimal point. Then, it executes the section of the program between the mentioned block and the one identified between the two decimal points as many times as set by the last digit. This digit may have a value between 0 and 99, unless it is programmed by a parameter, in which case the limits are 0 and 255.

If only N4.4 is written, the CNC will assume N4.4.1.

When the execution of this section is finished the CNC goes to the block next to the one in which G25 N4.4.2 was programmed.

```
Example: N0   G00  X10
         N5        Z20
         N10  G01  X50  M3
         N15  G00  Z0
         N20       X0
         N25  G25       N0.20.8
         N30            M30
```

When block 25 is reached, the CNC will jump to block 0 and will execute 8 times the section N0-N20. On completion of this, it will go to the block 30.

Functions G26,G27,G28,G29 and G30 (conditional jumps/calls) will be described in Chapter 13: PARAMETRIC PROGRAMMING. OPERATIONS WITH PARAMETERS.

6.9. G31-G32. Storage and retrieval of part program's datum point

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

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

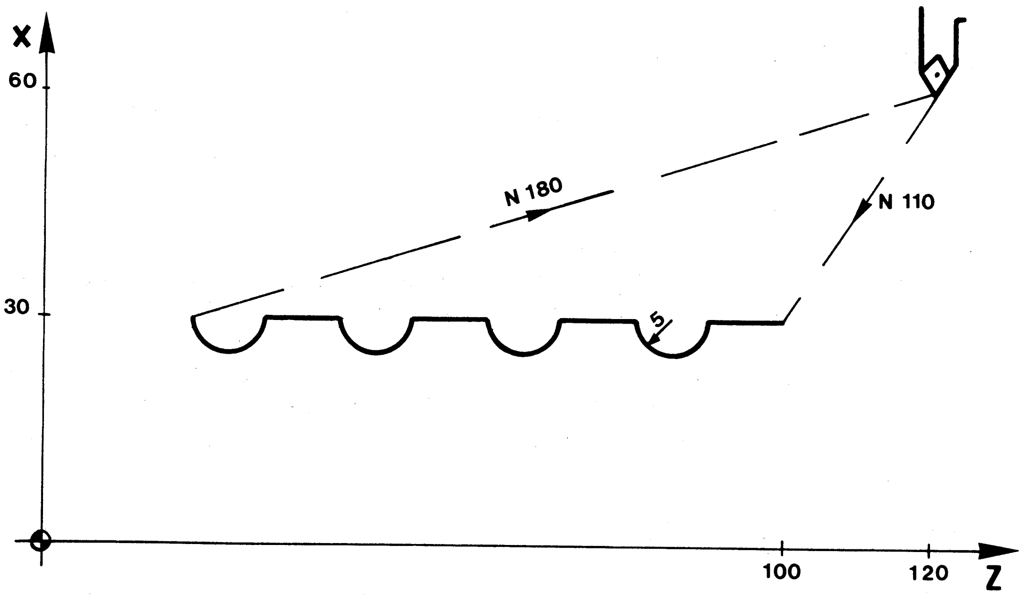
No other function can be programmed in a block in which G31 or G32 is programmed.

The format is :

```
N4  G31
N4  G32
```

N4 : Block number
G31: Keep the current coordinate origin
G32: Recover the coordinate origin kept by G31

Example:



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

```
N110 X60 Z100          Approach to the part
N120 G31               Keep the origin coordinates
N130 G92 X0 Z0         Zero offset
N140 G01 X0 Z-10       Machining
N150 G02 X0 Z20 R5     Machining
N160 G25 N130.150.3    Machining
N170 G32               Recover the initial origin
N180 G00 X120 Z120     Return to the starting point
```

6.10. G33. Threadcutting

Longitudinal frontal and tapered threads can be cut using G33 function. G33 is modal. It is cancelled by G00,G01,G02,G03,M02,M30, **EMERGENCY** or **RESET**.

Longitudinal thread

It can be programmed by means of: N4 G33 Z+/-4.3 K3.4 (mm)
Z+/-3.4 K2.4 (inches)

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

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

Frontal thread (scroll)

It can programmed as follows: N4 G33 X+/-4.3 I3.4 (mm)
X+/-3.4 I2.4 (inches)

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

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

Tapered thread

It can be programmed as follows:

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

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

The X and Z values will be absolute or incremental depending on whether G90 or G91 has been programmed. Only one pitch value (I,K) must be programmed. The CNC will calculate the other one. Thus:

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

or,

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

Can be programmed.

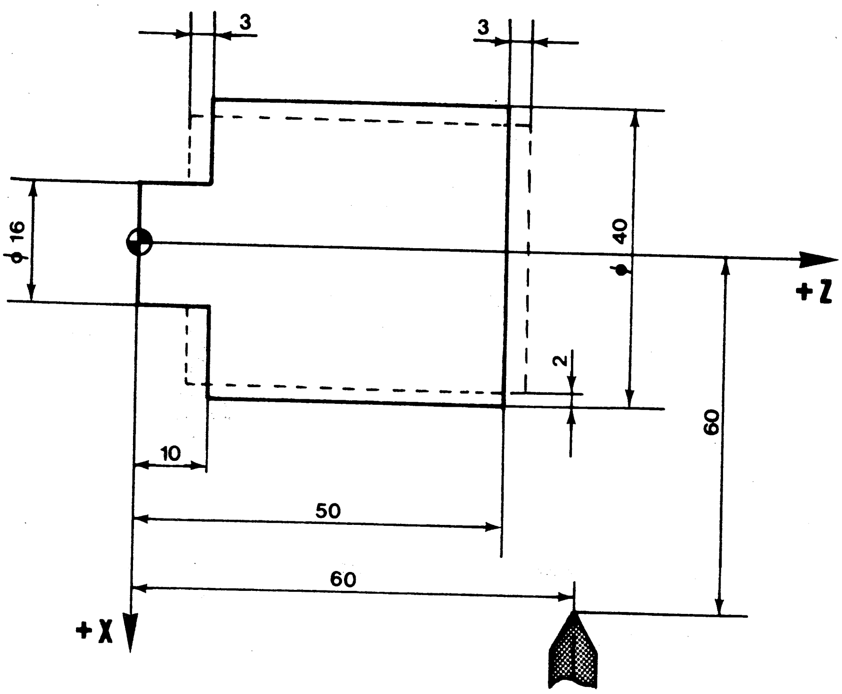
Nevertheless, both pitch values (I and K) can also be entered to force the CNC to cut the tapered thread with a pitch different from the one the CNC would have calculated.

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

EXAMPLES:

a) Longitudinal thread

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



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

Absolute coordinates

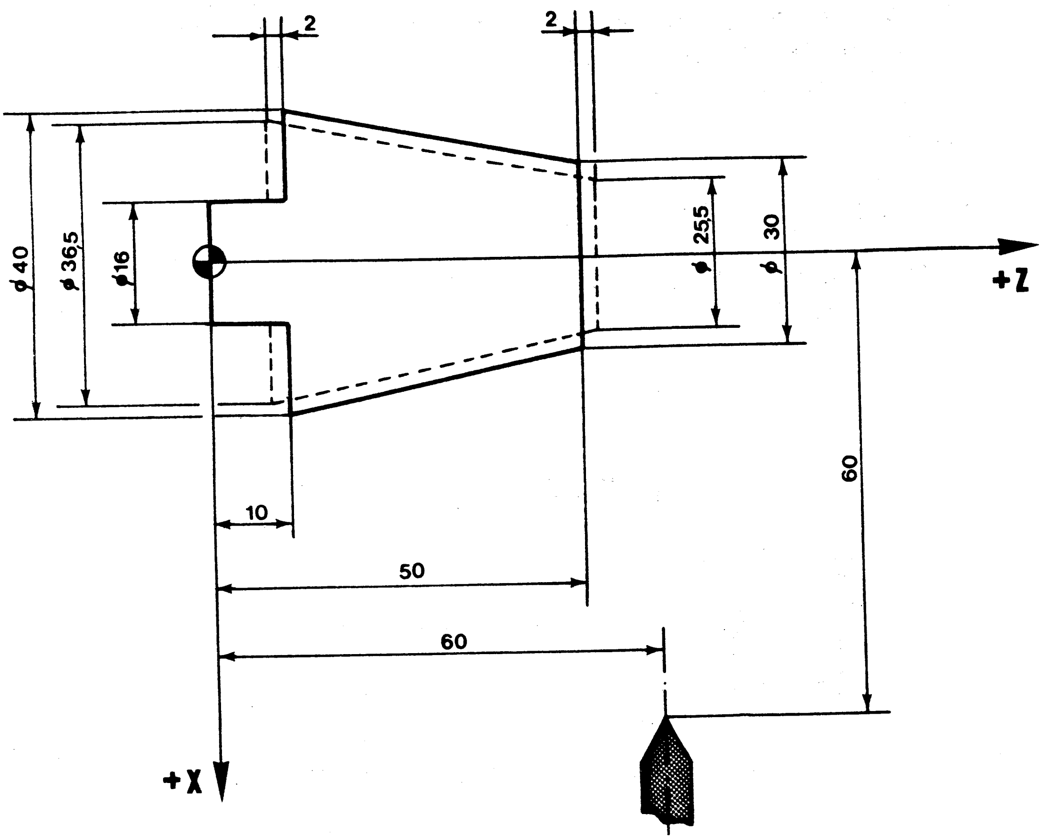
```
N0  G00  G90  X18  Z53
N5  G33            Z7   K5
N10 G00       X60
N15                Z60
```

Incremental coordinates

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

b) Tapered thread

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



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

Absolute coordinates

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

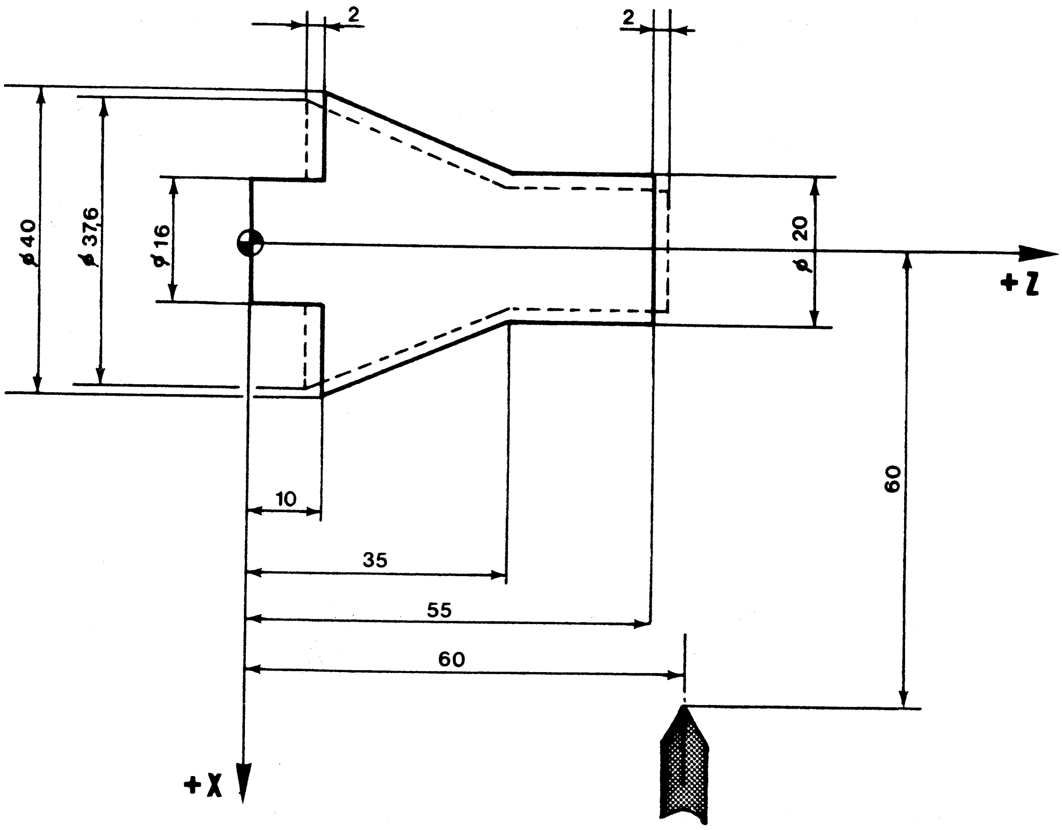
Incremental coordinates

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

c) Thread coupling

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

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



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

Absolute coordinates

```
N0   G00  G90  X8     Z57
N5   G33  G05         Z35   K5
N10            X18,8  Z8    K5
N15  G00       X60
N20                   Z60
```

6.11. G36. Controlled corner rounding

This function rounds the corners with a programmed radius, without the need to calculate the coordinates of the center and the initial and final points of the arc.

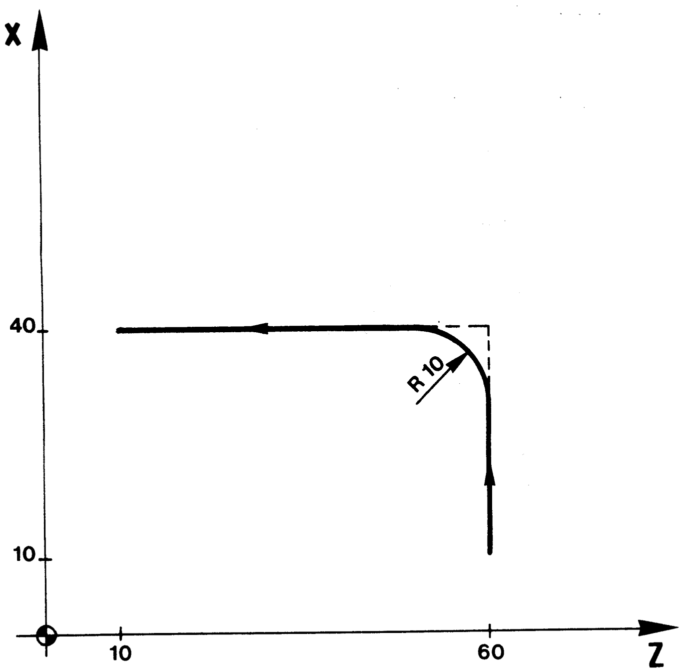
G36 is not modal; i.e. it must be programmed every time a corner rounding is needed.

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

The rounding radius must be always positive (R 4.3 or R3.4).

Examples: X in diameters

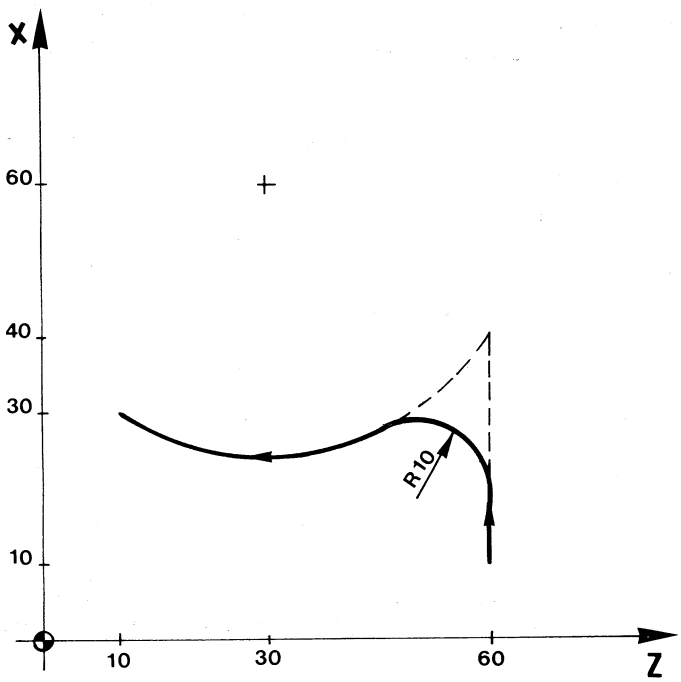
1. Straight-straight rounding



Starting point X20 Z60

```
N100 G90 G01 G36 R10 X80
N110 Z10
```

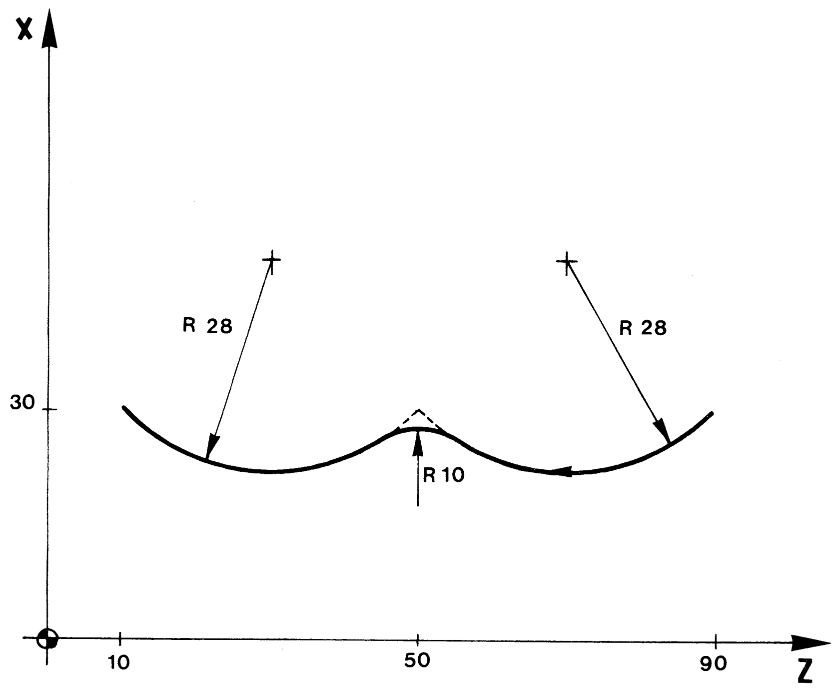
2. Straight-arc rounding



Starting point X20 Z60

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

3. Arc-arc rounding



Starting point X60 Z90

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

6.12. G37. Tangential approach at the start of machining

The preparatory function G37 can be used to link two paths tangentially without having to calculate the intersection points.

Function G37 is not modal, i.e., it has to be programmed every time two paths are to be linked tangentially. There paths may be straight-straight or straight-arc.

The radius, R4.3 in mm or R3.4 in inches, of the entry arc must be programmed following G37.

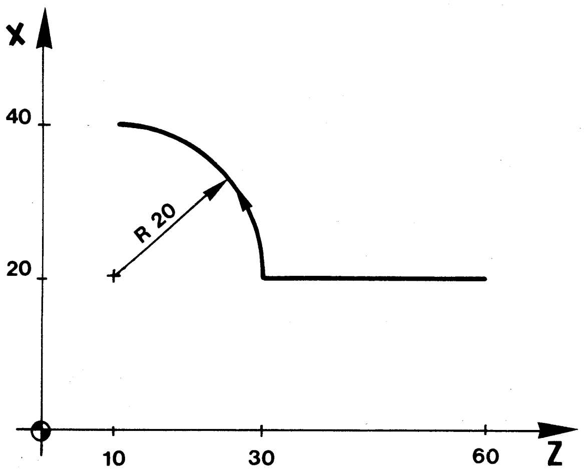
The value of the radius must be positive.

That programming has to be carried out in the block which incorporates the movement whose path is to be altered.

The movement must be rectilinear (G00 or G01).

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

Example: X in radius

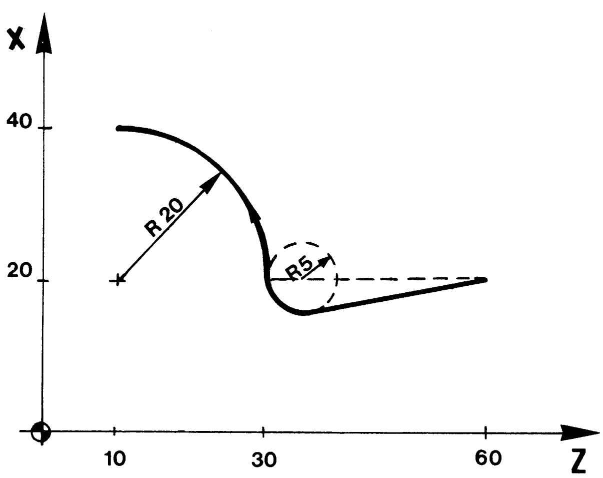


Let us suppose that the starting point is X20, Z60, program:

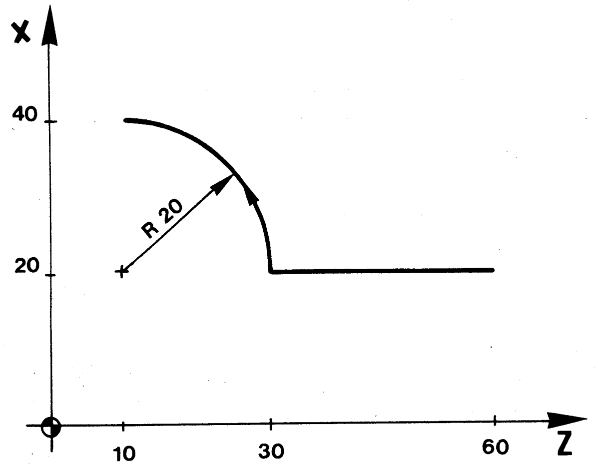
```
N100 G90 G01 X20 Z30
N110 G03     X40 Z10 R20
```

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

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



Example: X in radius

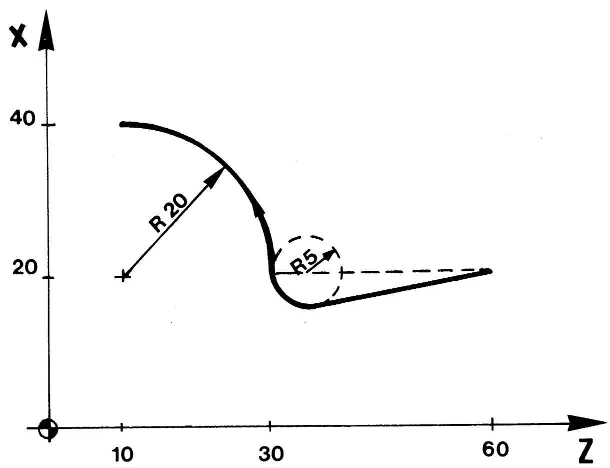


Let us suppose that the starting point is X20, Z60, program:

```
N100 G90 G01 X20 Z30
N110 G03     X40 Z10 R20
```

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

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



6.12. G37. Tangential approach at the start of machining

The preparatory function G37 can be used to link two paths tangentially without having to calculate the intersection points.

Function G37 is not modal, i.e., it has to be programmed every time two paths are to be linked tangentially. There paths may be straight-straight or straight-arc.

The radius, R4.3 in mm or R3.4 in inches, of the entry arc must be programmed following G37.

The value of the radius must be positive.

That programming has to be carried out in the block which incorporates the movement whose path is to be altered.

The movement must be rectilinear (G00 or G01).

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

6.13. G38. Tangential exit on completion of machining

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

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

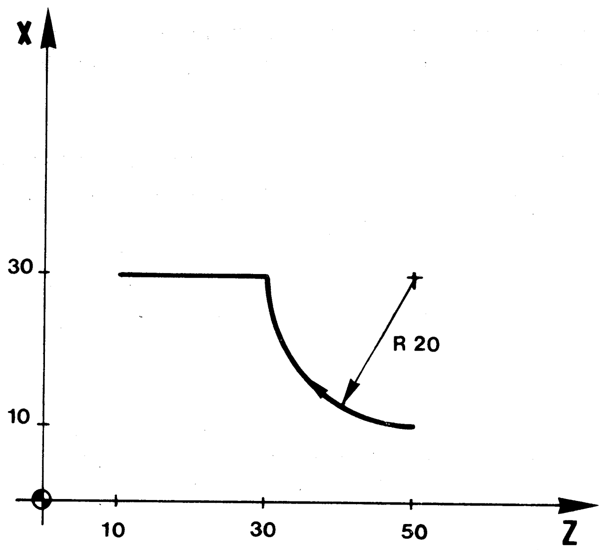
The radius R4.3 in mm or R3.4 in inches, of the exit arc must be programmed following G38.

The value must be positive.

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

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

Example: X in radius

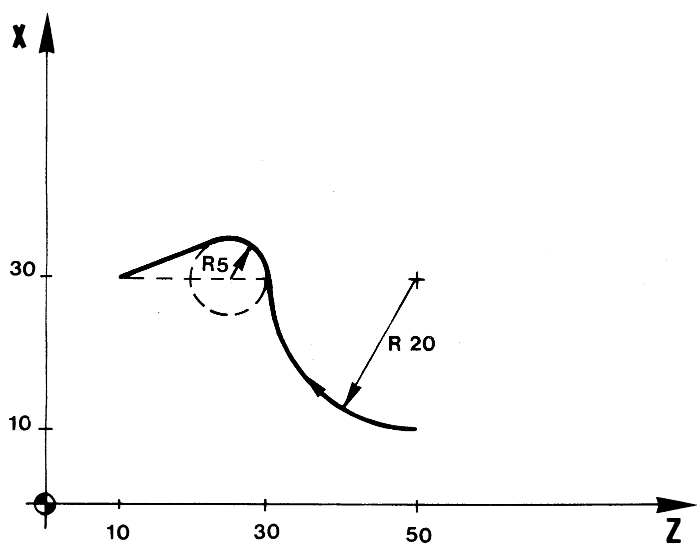


Let us suppose the starting point is X10, Z50. Program:

```
N100 G90 G02 X30 Z30 R20
N110 G01     X30 Z10
```

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

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



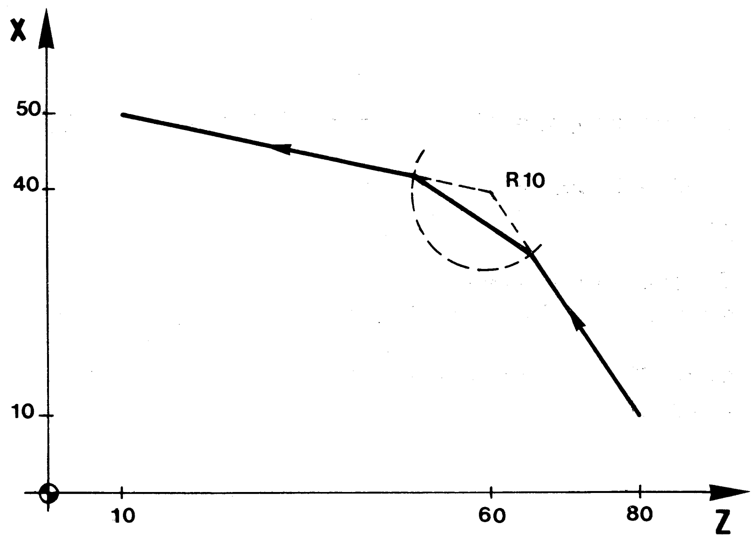
6.14. G39. Chamfering

This function chamfers the corner between two straight lines without the need to calculate the coordinates of the two intersections.

G39 is not modal; i.e. it must be programmed every time a chamfering is need. It must be programmed in the same block as the movement whose end must be chamfered.

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

Example: X in radius



Starting point X20 Z80

```
N100 G90 G01 G39 R10 X80  Z60
N110                 X100 Z10
```

6.15. Tool radius compensation

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

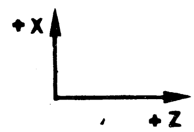
Tool compensation enables the contour of the part to be programmed directly without taking the dimensions of the tool into account.

The CNC automatically calculates the path to be followed by the tool, based on the contour of the part and the tool dimensions stored in the tool table. Every time a tool (T2.2) is selected the CNC automatically applies the tool length compensation (X,Z,I,K) stored in the table, without having to program any G code. If P98(5) is 1 the tool length compensation is effective when **M06** is executed.

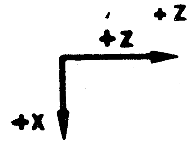
There are three preparatory functions for tool radius compensation:

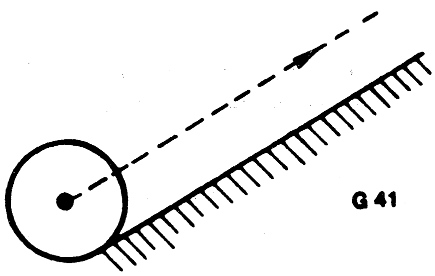
G40 : Cancellation of tool radius compensation
G41 : Left hand tool radius compensation
G42 : Right hand tool radius compensation

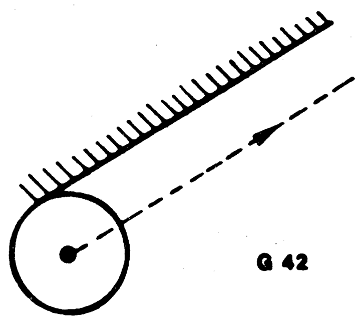
Parameter P94(1) = 0 and axes



Parameter P94(1) = 1 and axes



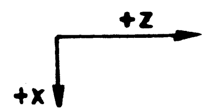




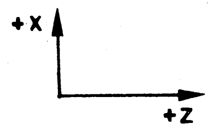
NOTE: The tool is on the part's left side as seen following the direction of the movement (G41).

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

If parameter P94(1) = 0 and axes



or parameter P94(1) = 1 and axes



G41 operates as G42 and viceversa.

The CNC 8020 T has a table of up to 32 tool offsets comprising for each tool length (X,Z,I,K) and radius (R) values plus location codes (F).

The compensation values must be stored in the **tool offsets mode (8)**. The values of **I,K** can also be checked and modified, without stopping the execution of a program (see Operation Manual). The tool table can also be loaded by using G50 in the program.

The max. values are:

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

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

R (Radius) 1000.000 mm (39.3700 inches)

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

Possible codes are F0-F9 (see figure).

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

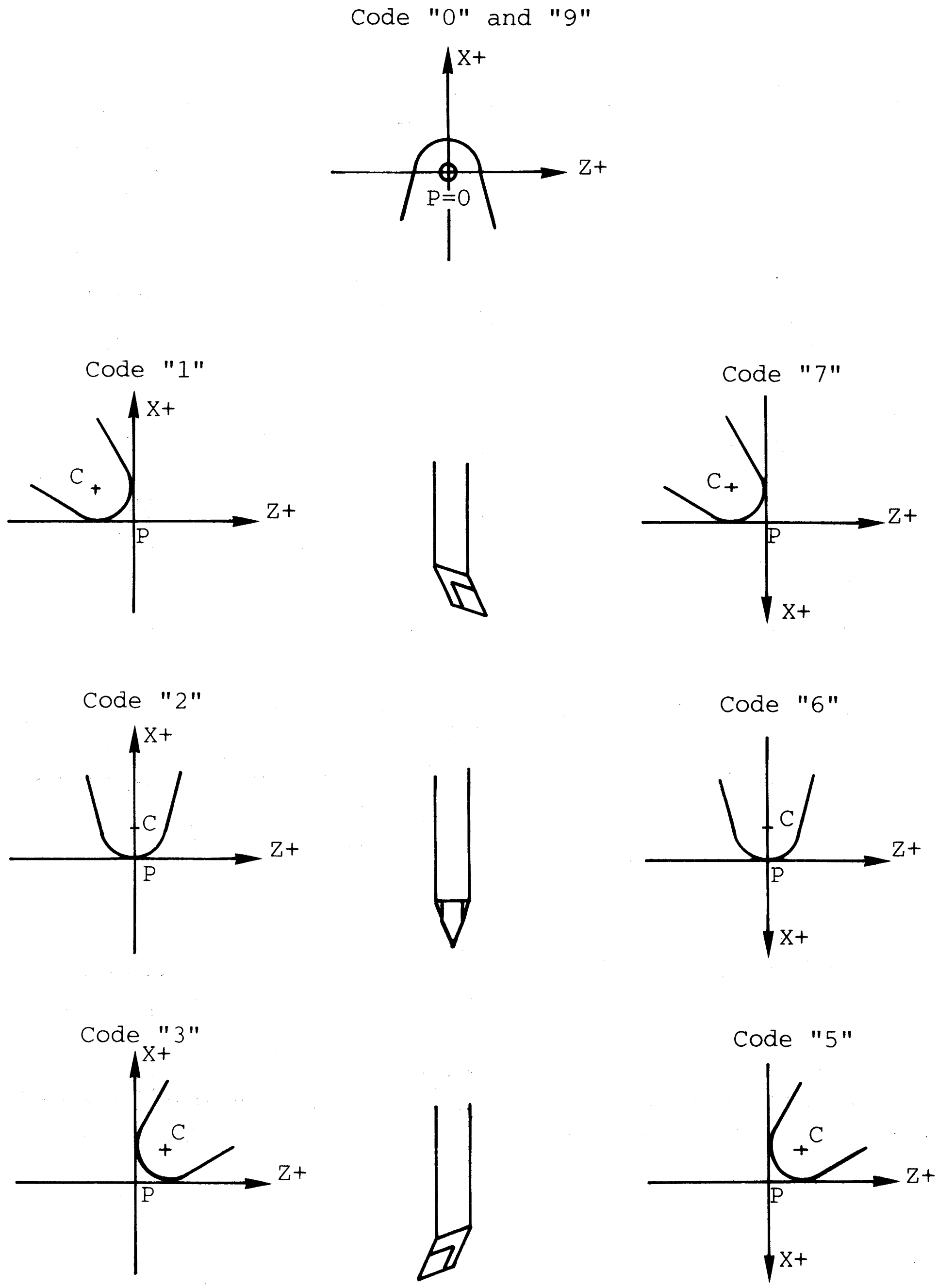
Functions G41 and G42 are modal (persistent) and are cancelled by G40,M02,M30 as well as by an **EMERGENCY** or by a general **RESET**.

The CNC applies tool length compensation (X,Z,I,K) as soon as a tool (Txx.01) is programmed, unless P98(5) is 1. In this case the compensation is effective after **M06**.

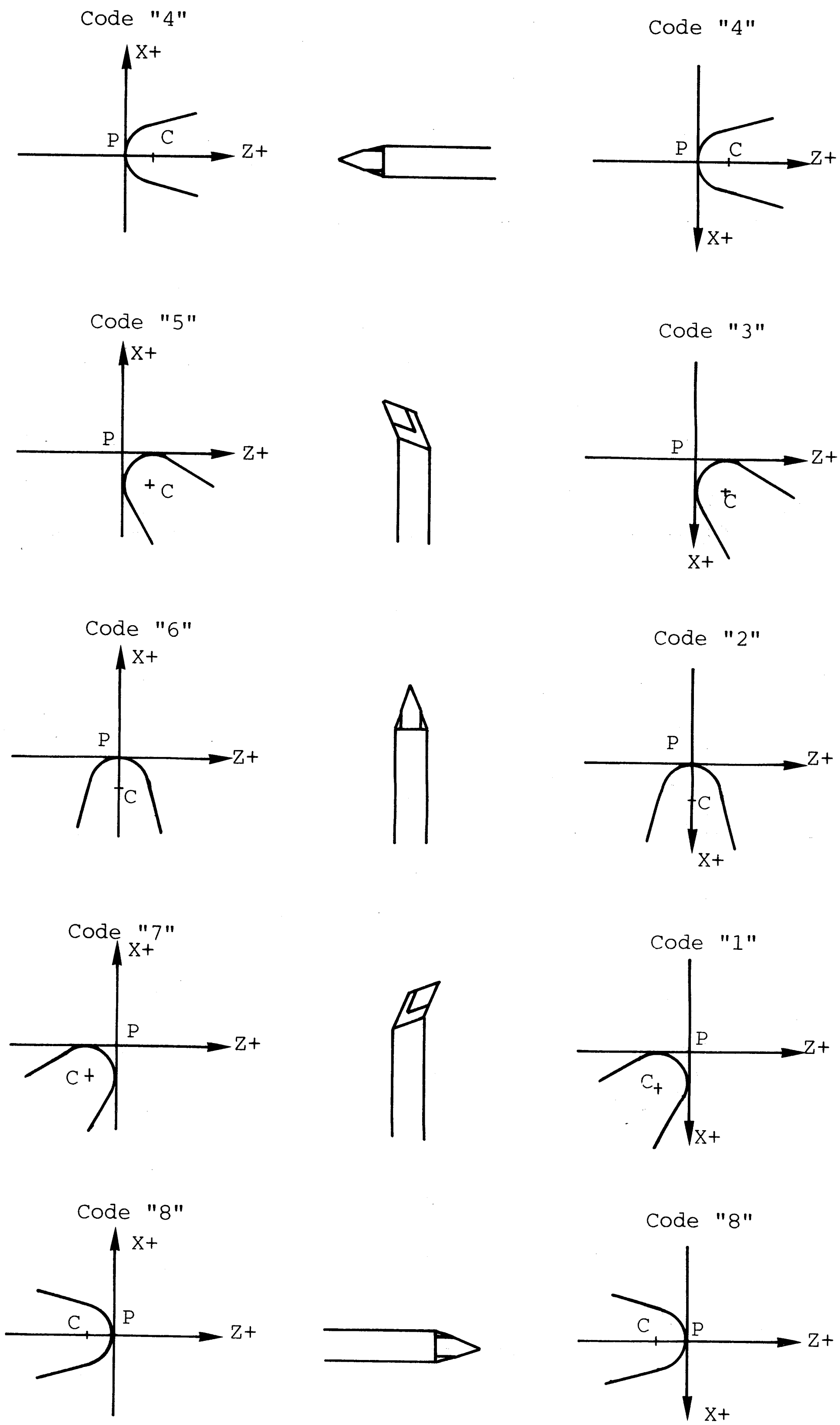
NOTE:

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

LOCATION CODES



P : Tool tip
C : Tool nose radius center


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

6.15.1. Selection and initiation of tool radius compensation

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

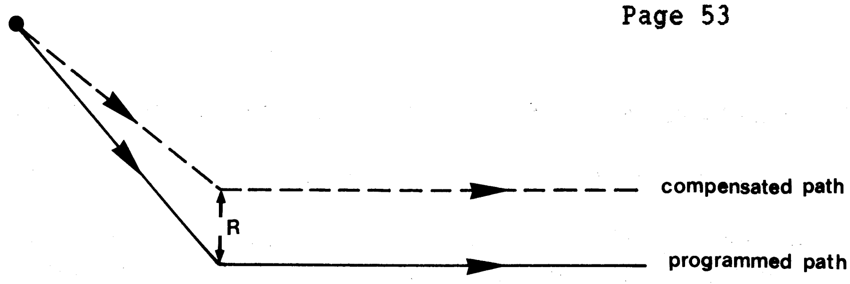
Either the block in which G41/G42 is programmed or a previous block must include programming of function Txx.xx (Txx.00-Txx.32) to select from the tool table the correction value to be applied. If no tool is selected, the CNC assumes the value T00.00.

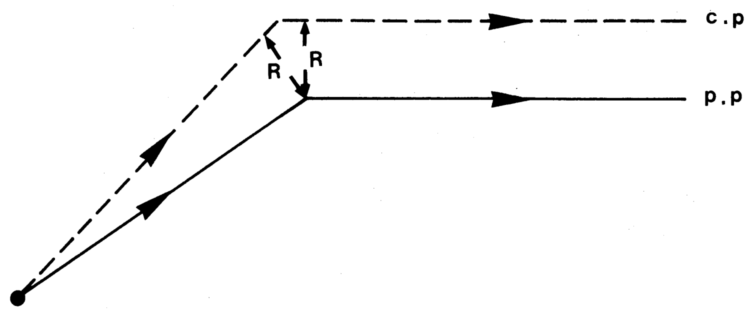
NOTE:

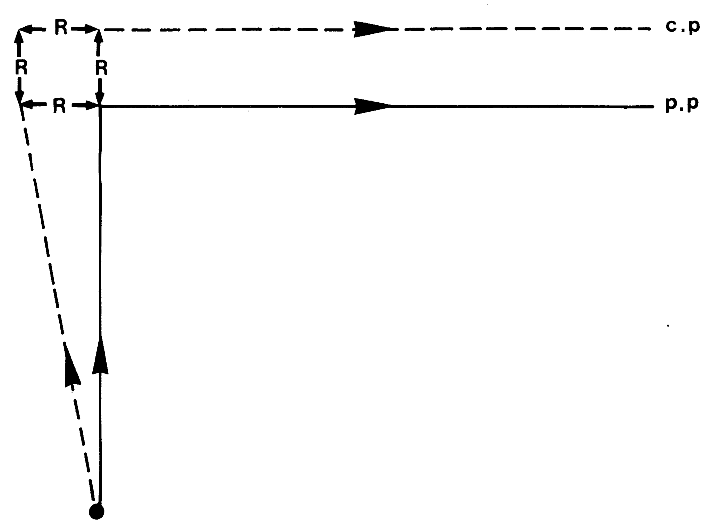
- Tool radius compensation selection (G41/G42) can only be carried out when G00 or G01 (rectilinear movements) is active.

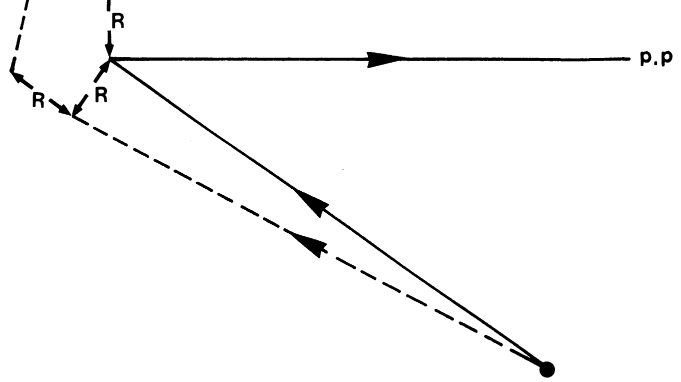
- If the first call for compensation is made when G02 or G03 are active, the CNC will display error code 41.

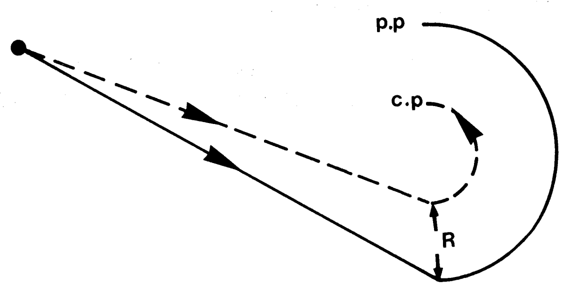
The next page illustrates various cases of initiation of tool radius compensation.

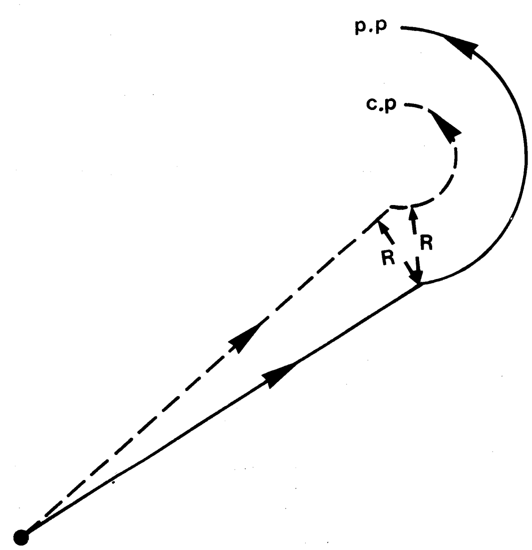

compensated path
R
programmed path

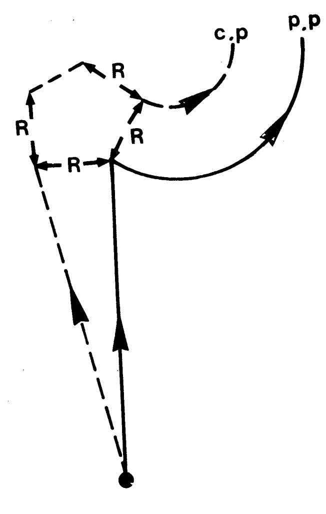

c.p
R
R
p.p

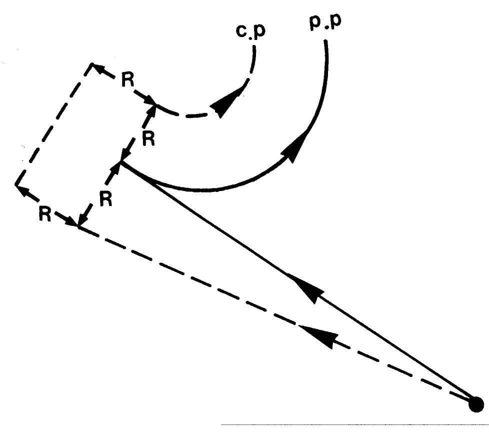

R
c.p
R
R
R
p.p


R
c.p
R
p.p
R
R


p.p
c.p
R

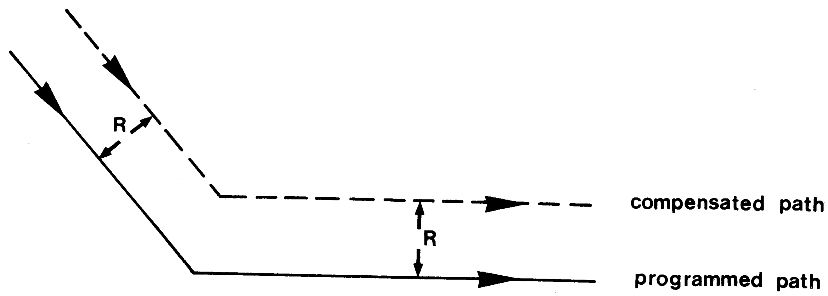

p.p
c.p
R
R

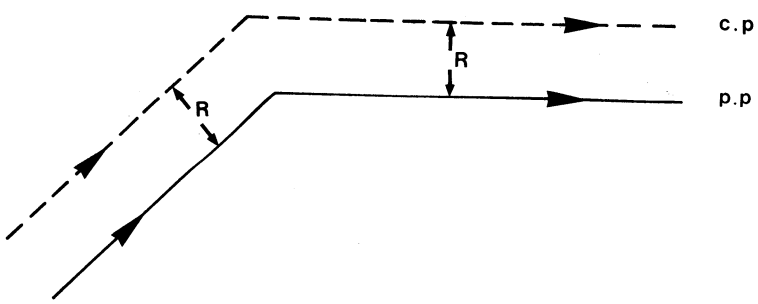

c.p
p.p
R
R
R
R

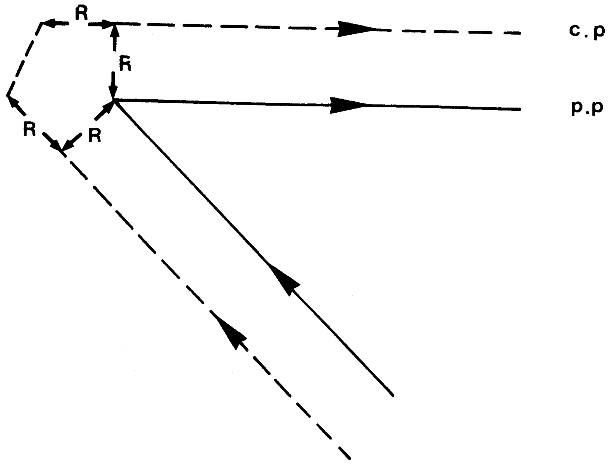

c.p
p.p
R
R
R
R

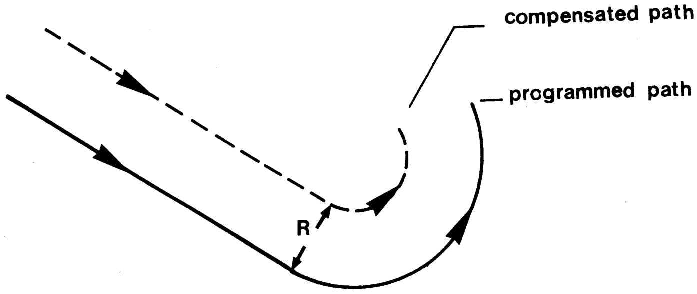
6.15.2. Operating with tool radius compensation

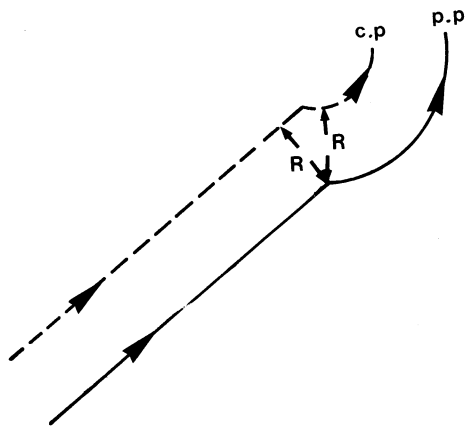
The graphs below illustrate the various paths followed by a tool controlled by a CNC programmed with radius compensation.

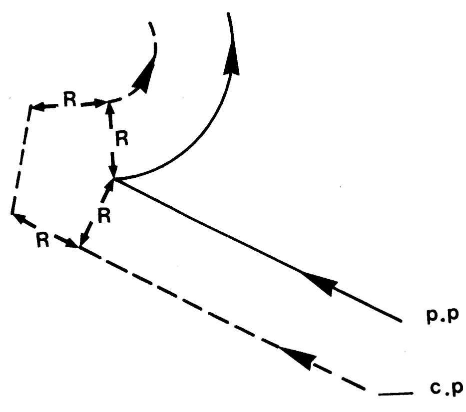


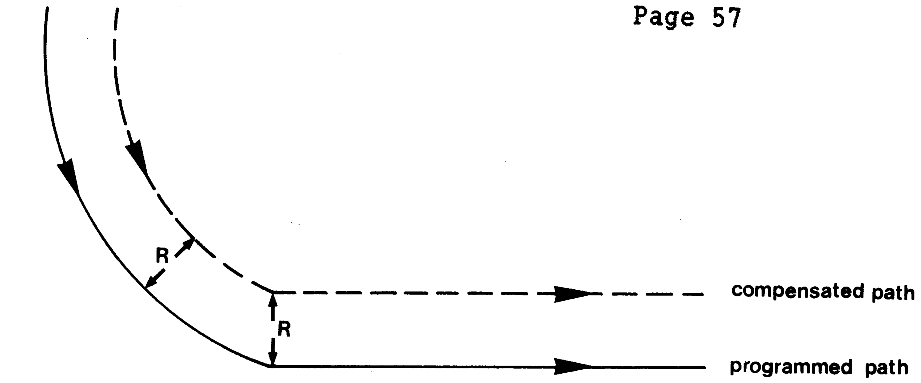


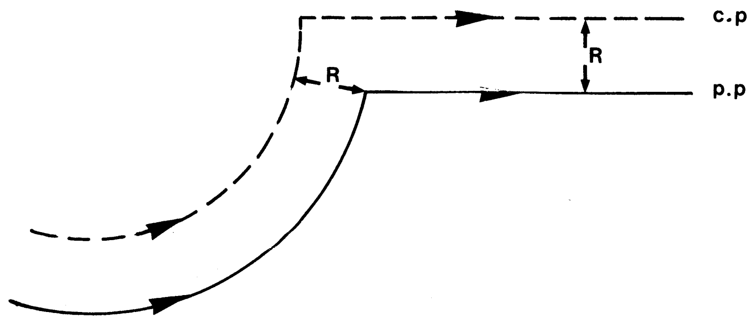


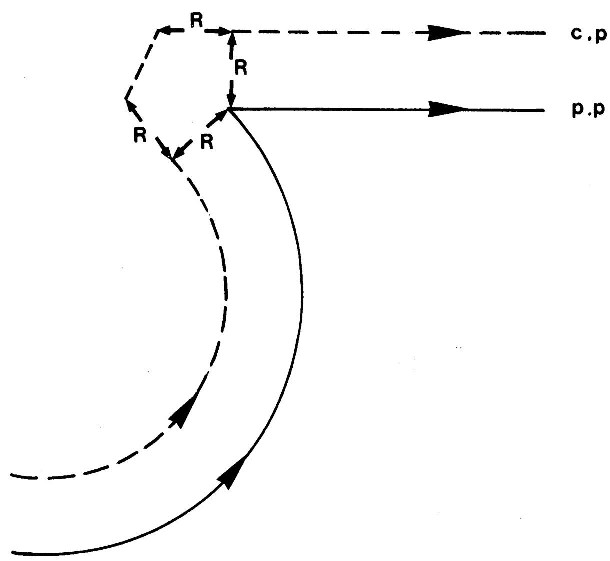

compensated path
programmed path
R

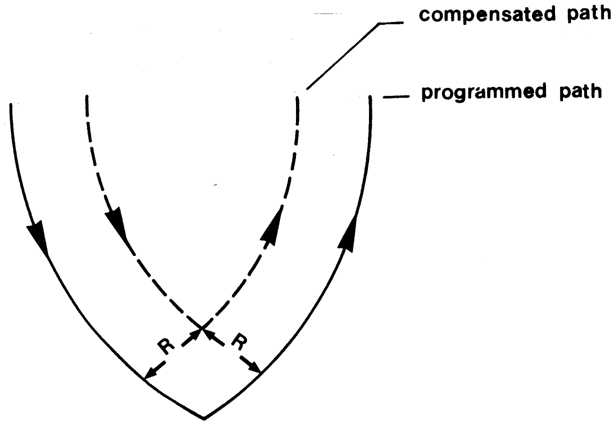

c.p
p.p
R
R

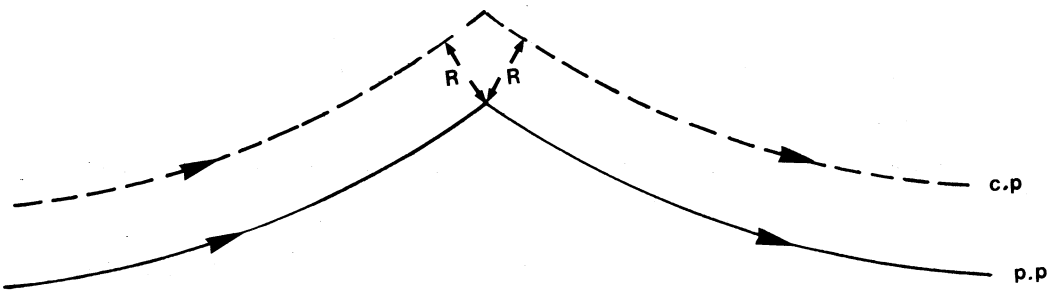

R
R
R
R
p.p
c.p

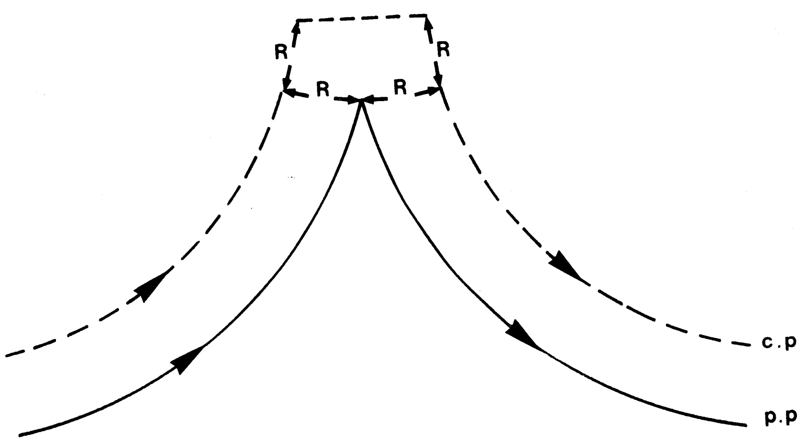

R
R
compensated path
programmed path


c.p
R
R
p.p


R
c.p
R
p.p
R
R

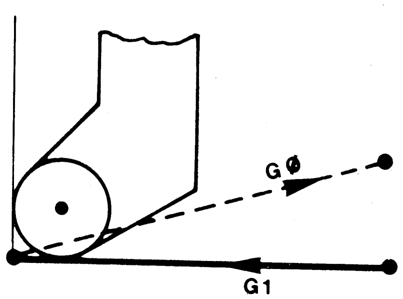

compensated path
programmed path
R
R

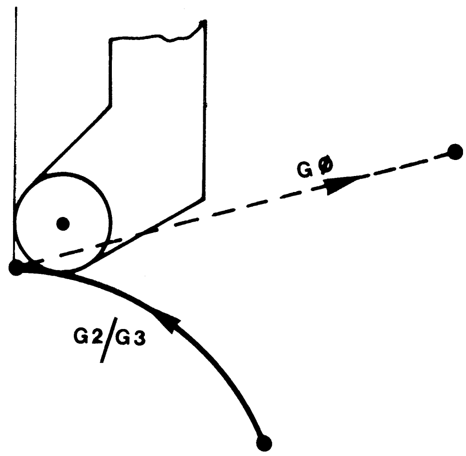

R
R
c.p
p.p


R
R
R
R
c.p
p.p

6.15.3 Tool radius compensation freeze with G00

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





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

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

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

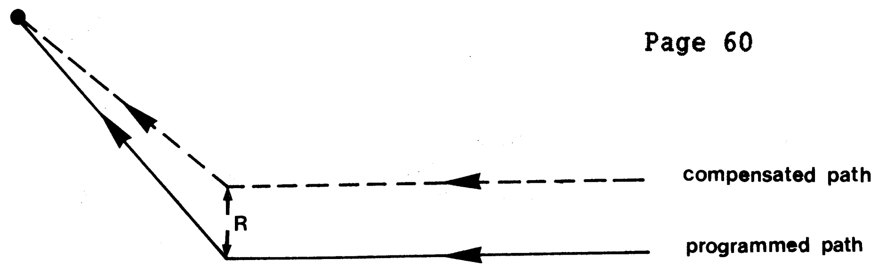
6.14.4. Cancellation of radius compensation

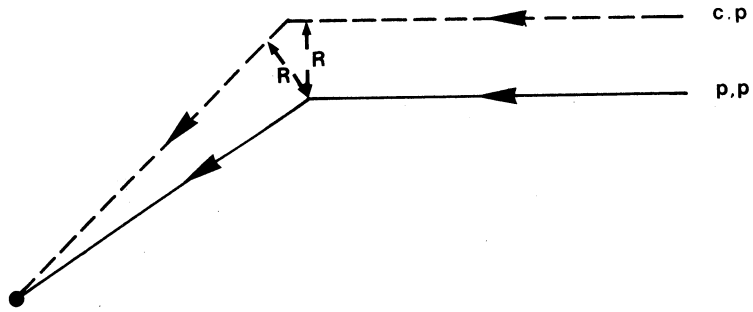
Radius compensation cancellation is achieved by function G40.

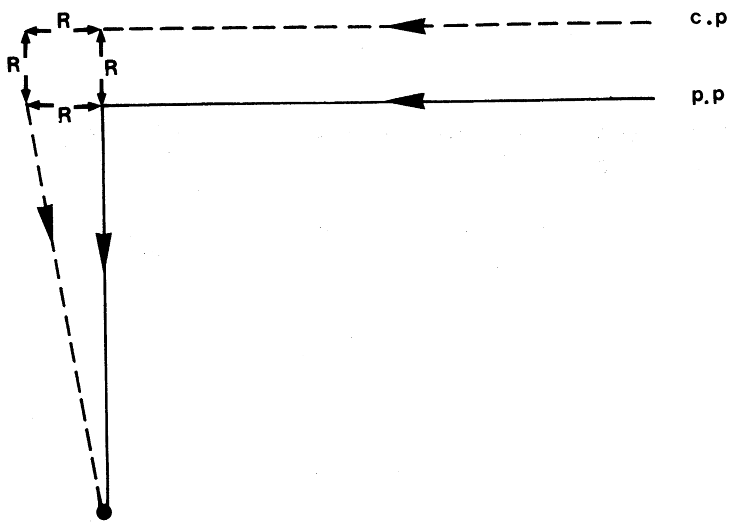
It should be borne in mind that radius compensation cancellation (G40) can only be carried out in a block in which a rectilinear movement is programmed (G00,G01).

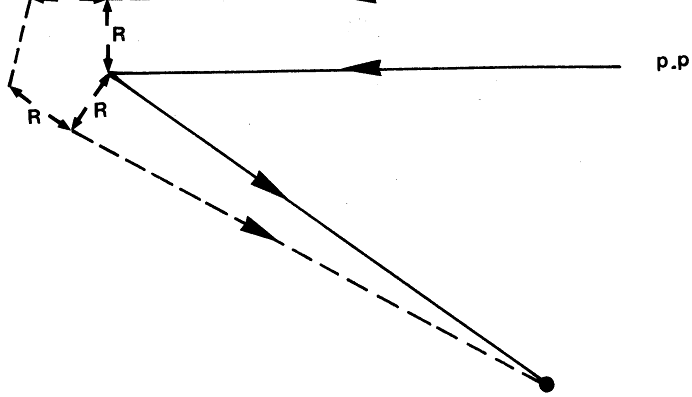
If G40 is programmed in a block containing G02 or G03, the CNC will give error 48.

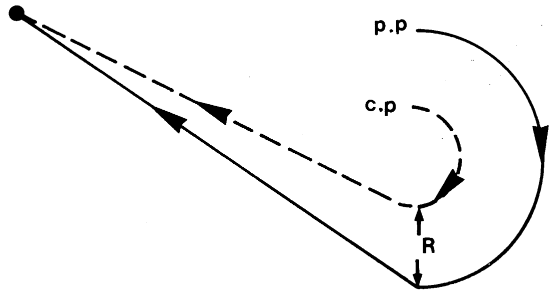
The following is a table of various cases of cancellation of compensation.

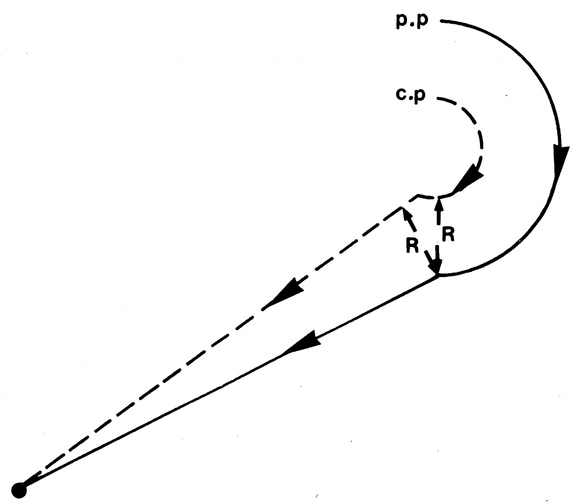

compensated path
R
programmed path

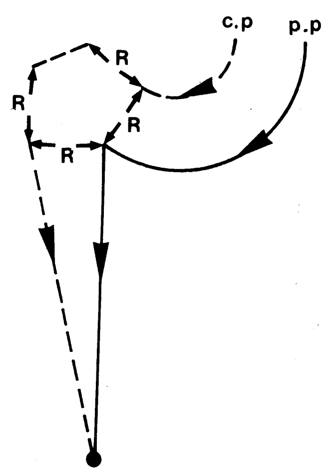

c.p
R
R
p,p

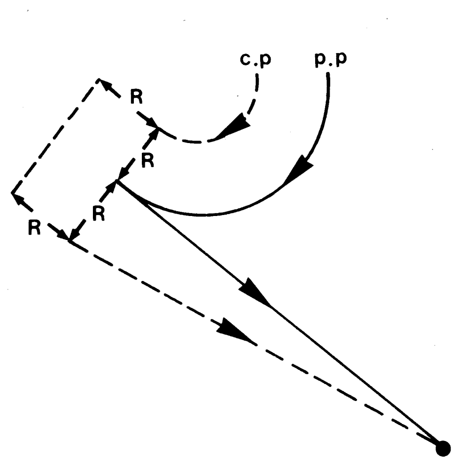

R
R
R
R
c.p
p,p


R
R
R
R
c.p
p.p


p.p
c.p
R


p.p
c.p
R
R


c.p
p.p
R
R
R
R


c.p
p.p
R
R
R
R

6.16. G50. Loading of the values in the tool offset table

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

There are many method to program the function G50:

a) Entering of **all** the values

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

X+/-3.4 Z+/-3.4 F1 R2.4 I+/-1.4 K+/-1.4 inches

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

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

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

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

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

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

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

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

6.17. G51. Alteration of the I and K values of the engaged tool

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

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

artificially alters the values of I,K.

N4 - Block number

G51 - Tool dimensions alteration code

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

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

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

6.18. G53-G59. Zero offsets

7 different zero offsets can be selected by functions G53,G54,G55,G56,G57,G58 and G59. The values of these offsets are stored in the CNC memory after the tool dimensions table and are referred to the **machine-reference-zero** point.

The values can be entered in **operation mode** 8 via the keyboard or by program, using codes G53-G59.

The functions G53-G59 can be used in two different ways:

a) To load the zero offset table.

. Absolute loading of values

X+/-4.3 Z+/-4.3 in mm (or X+/-3.4 Z+/-3.4 in inches) the values identified by X,Z are loaded in the table address defined by G5? (G53-G59).

N4	: Block number
G5?	: Offset code (G53,G54,G55,G56,G57,G58,G59)
X+/-4.3 (X+/-3.4)	: Zero offset value referred to the machine reference zero point on the X axis.
Z+/-4.3 (Z+/-3.4)	: Zero offsets value referred to the machine reference zero point on the Z axis.

- Incremental loading of values

Block N4 G5? I+/-4.3 K+/-4.3 in mm (or N4 G5? I+/-3.4 K+/-3.4 in inches) will increment the contents of tool table address indicated by G5? (G53/G59) by an amount determined by I,K.

N4	:	Block number.
G5?	:	Zero offset code (G53,G54,G55,G56,G57,G58,G59)
I+/-4.3 I+/-3.4	:	Amount added or subtracted to the X value previously stored in the tool table.
K+/-4.3 K+/-3.4	:	Amount added or subtracted to the Z value previously stored in the tool table.

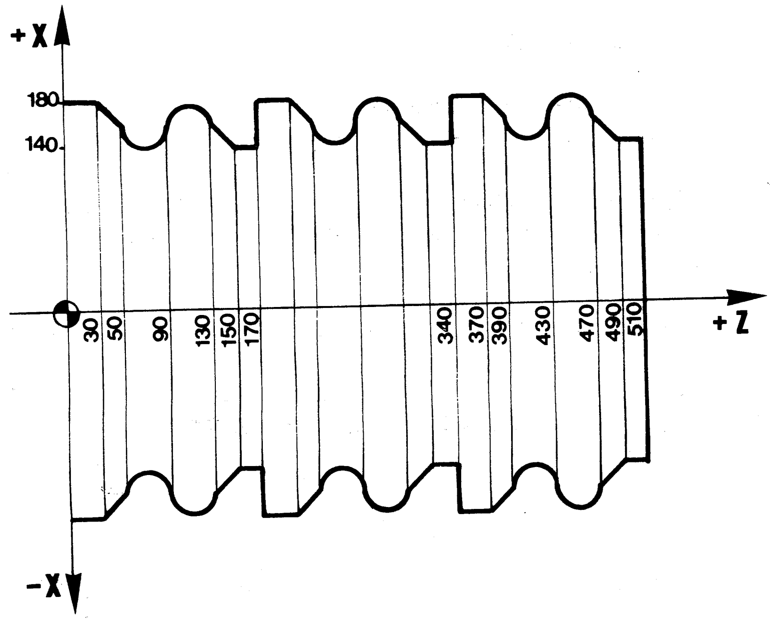
b) To apply a zero offset to the current program:

A block like N4 G5? is used to carry out a zero offset on the current program, according to the values stored in the G5? position of the zero offset table (G53-G59).Example: Starting point X200 Z20

N4 : Block number

G5?: (G53,G54,G55,G56,G57,G58,G59): Memory address in which the zero offset values are stored.

Example:



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

In the G53/G59 table we will enter:

```
G53 X0 Z340
G54 X0 Z170
G55 X0 Z0
```

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

6.19. Units of measurement

G70 : Programming in inches
G71 : Programming in millimeters

Depending on whether G70 or G71 is programmed, the CNC takes the subsequent coordinates as being in inches or millimeters respectively.

Functions G70/G71 are modal and incompatible with one another.

The CNC assumes the units set by parameter P13 when being turned **ON**, after M02,M30, **EMERGENCY** or **RESET**.

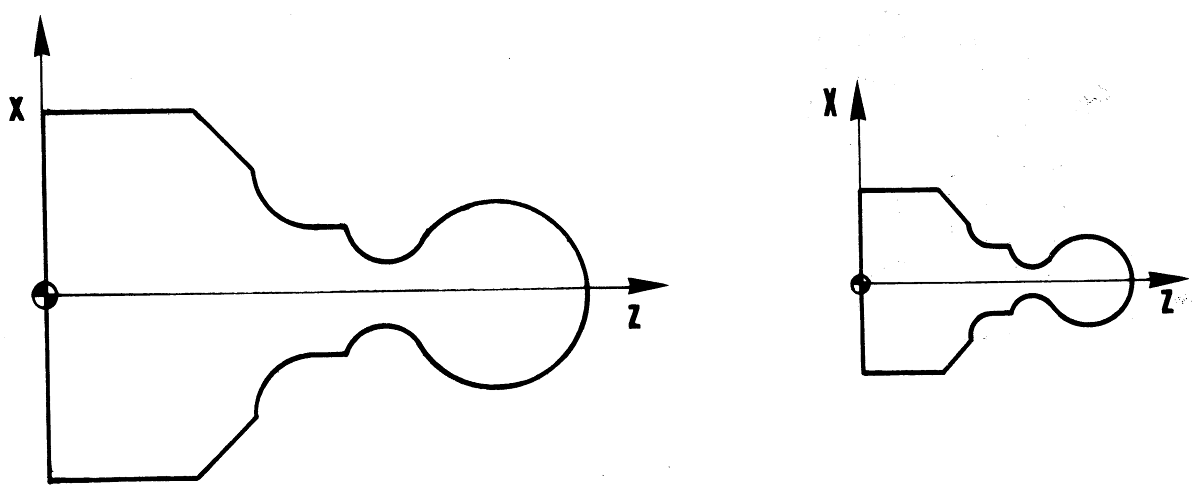
6.20. G72. Scaling

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

Format: N4 G72 K2.4

N4 : Block number
G72 : Scaling code
K2.4: Value of scaling factor
Min. value K0.0001 (X0.0001)
Max. value K99.9999 (X99.9999)

All coordinate values programmed after G72 will be multiplied by K until the scaling is cancelled by K1, or after M02,M30, **EMERGENCY** or **RESET**.



6.21. G74. Machine-reference-search

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

There are two possible cases:

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

b) **One or two axes referencing (Z X)**. If machine reference search is required in an order other than the above, G74 is programmed, followed by the axes in the required order.

No other function can be programmed in a block in which G74 is programmed.

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

6.22. G75. Probing

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

Format: N4 G75 X+/-4.3 Z+/-4.3

The axes will move until the probe signal is received. The CNC will then consider the block to be completed and the real position of the axes will be stored as theoretical position.

Neither the feedrate will be changed by turning the **FEEDRATE** knob (frozen at 100%) nor the movement of the axes will be displayed until the probe signal is received.

If the axes arrive in position before the probe pins the part the CNC will give error code 65. After executing this block, the values of the different axes can be allocated to parameters.

The combination of this feature with mathematical operations with parameters allows the creation of special subroutines to measure parts or tools.

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

NOTE:

Besides the probing canned cycles described next, the CNC allows the manual measuring and loading of tool dimensions with a probe. Refer to the **OPERATING MANUAL**.

6.22.1. Probing canned cycles

The CNC 8020 T offers various probing canned cycles to measure tool and part dimensions.

The programming format is as follows:

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

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

The CNC's probing canned cycles are:

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

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

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

. If any parameter that corresponds to a cycle is not programmed, the CNC will assume the latest value assigned to that parameter. The cycles do not modified the calling parameters (which can be used in later cycles) but do alter the contents of parameters P70 thru P99.

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

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

. Parameter P3 must be greater than zero.

. Parameter P5 must be equal or greater than zero.

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

BASIC OPERATION

The movements of the axes during a probing cycle are:

Aproach

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

Probing

It is executed at a feedrate determined by P4 until the CNC receives the probe signal.

If before moving a maximum distance of 2P3 the CNC has not received the probe signal, error 65 will be displayed and all axes will be stopped.

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

Withdrawl

Once the probing corresponding to selected cycle is finished, the axes will withdraw, in rapid move G00, back to the starting point.

Depending on the selected cycle, the CNC will update, if necessary, the tool table, by the same token, the values of the parameters will have a specific meaning which will be described in the sections for each cycle.

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

DIS MODE
3
P

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

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

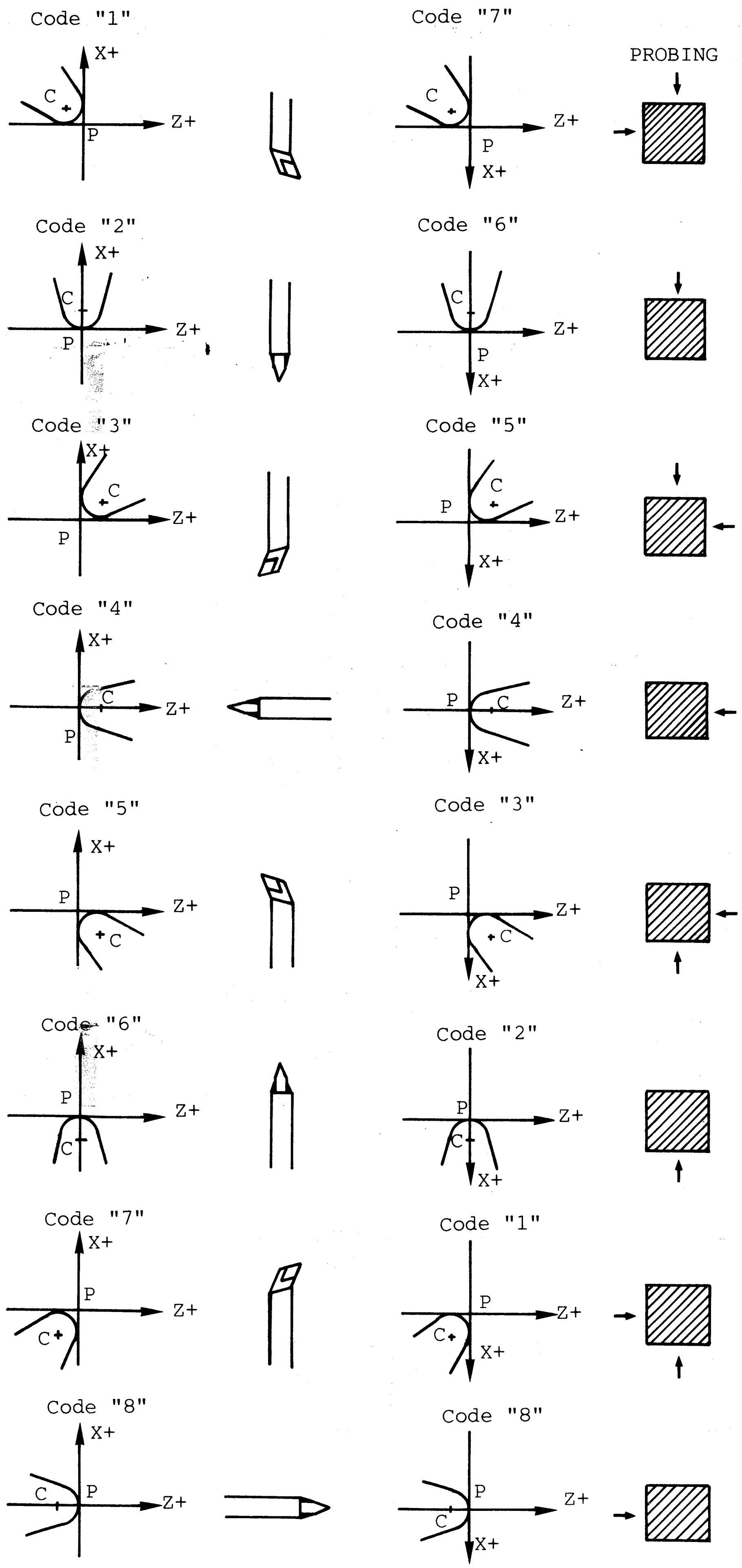
The type of probe used in this cycles may be either one located in a fixed position on the machine (used to calibrate the tools) or one placed on the turret (used to measure parts).

The latter probe will act as if it were a tool and must be calibrated prior to the execution of the cycle and the **X,Z,F,R,** values entered in the appropriate tool table position.

While executing a probing canned cycle, if the CNC receives the probe signal without the probing movement itself being executed, it will issue an error 65 stopping all axes(collision).

When the probe uses an infrared system to send the signal, it is necessary to indicate, with machine-parameter P116, which **M** function must the CNC send to activate the probe.

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

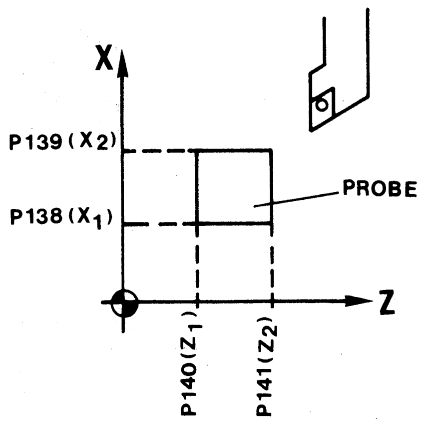

Code "1"
Code "7"
PROBING
Code "2"
Code "6"
Code "3"
Code "5"
Code "4"
Code "4"
Code "5"
Code "3"
Code "6"
Code "2"
Code "7"
Code "1"
Code "8"
Code "8"
X+
Z+
C
P

NO. TOOL CALIBRATION CYCLE

To execute this cycle, a probe must be placed in a fixed position on the machine and with its sides parallel to the axes.

The CNC must know this position on each axis and with respect to the machine-reference-zero. These values must be entered in the following parameters:

P138 Minimum (X1) value (in radius)
P139 Maximum (X2) value (in radius)
P140 Minimum (Z1) value
P141 Maximum (Z2) value



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

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

Cycle programming format:

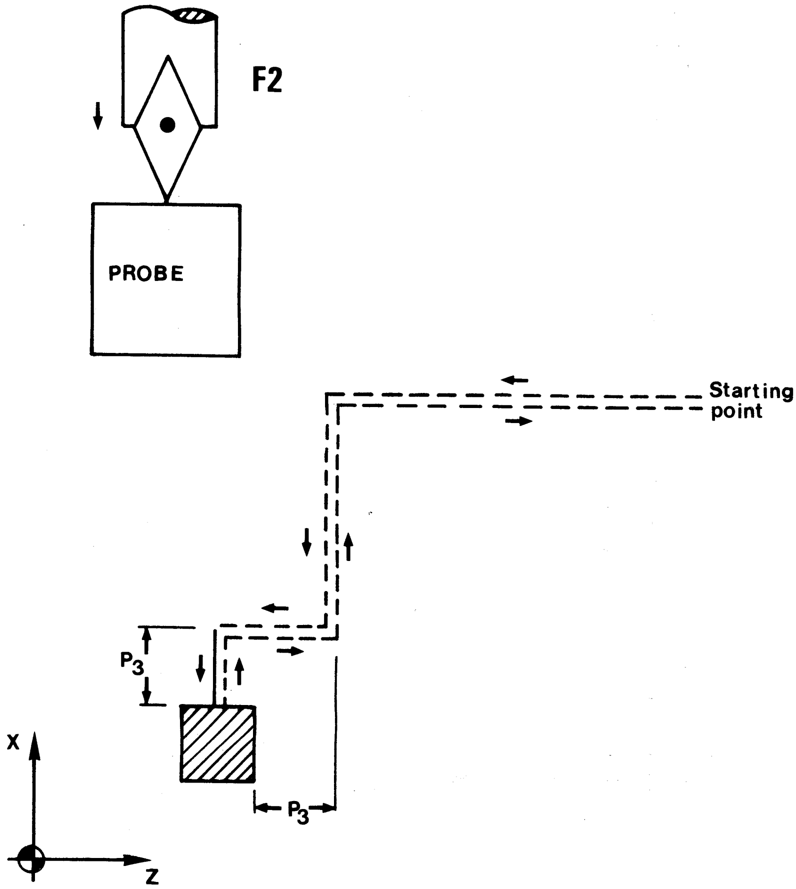
G75 NO P3=K-- P4=K--

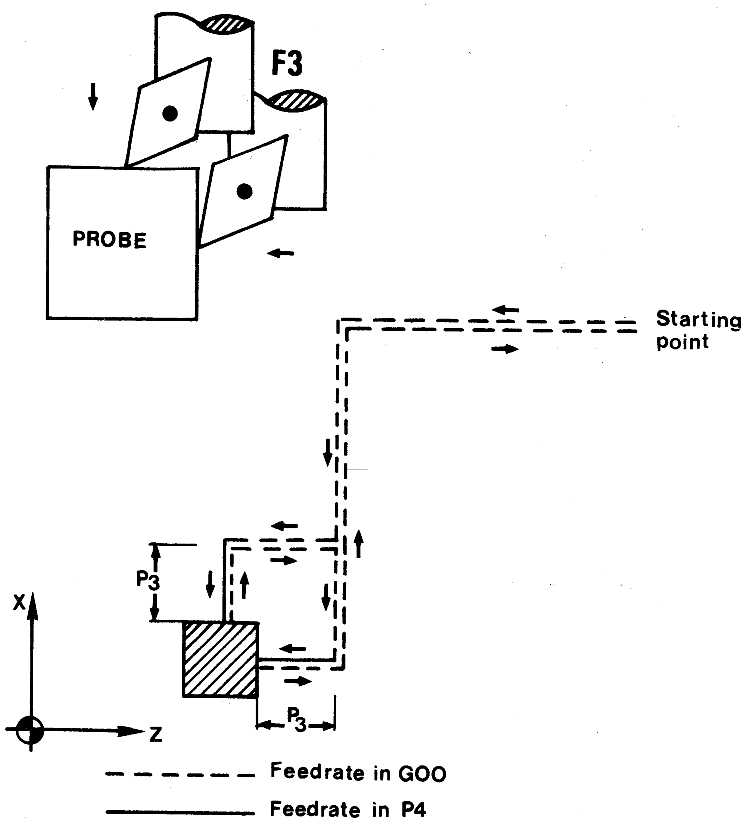
G75 NO = Tool calibration cycle code.
P3 = Safety distance (in radius).
P4 = Probing feedrate.

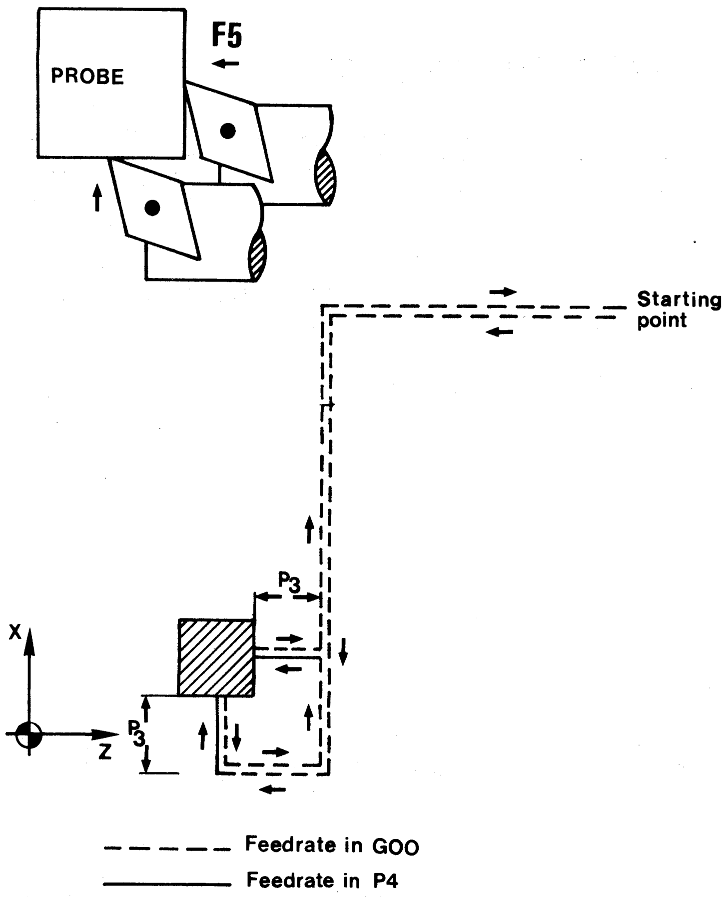
The CNC will execute one or two probings depending on the tool's location code **F**. (see fig.)

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

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







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

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

Also, Parameters P93 and P95 will indicate:

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

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

N1. PROBE CALIBRATION CYCLE

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

The approximate values of the sides of the probe are given to the CNC by entering them in the machine parameters P138, P139, P140, P141.

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

cycle programming format:

G75 N1 P3=K-- P4=K--

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

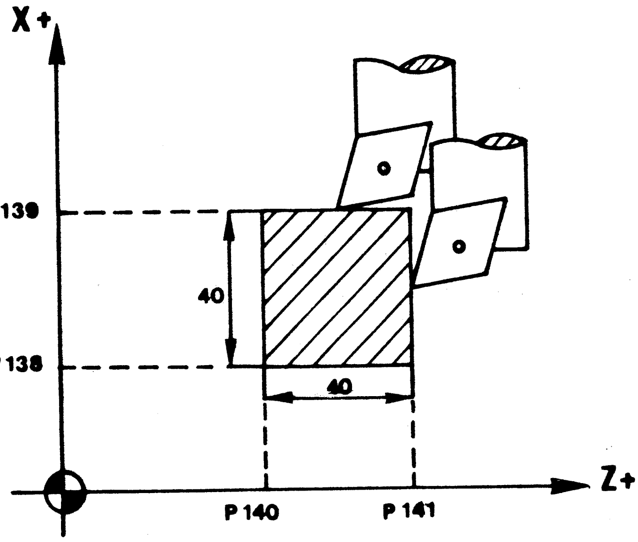
The different movements of this cycle are identical to those described for the tool calibration cycle **N0.**

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

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

Knowing these values and the probe's dimensions, the operator must calculate the values of the other sides of the probe and update, with those values, the machine-parameters P138, P139, P140 and P141.

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

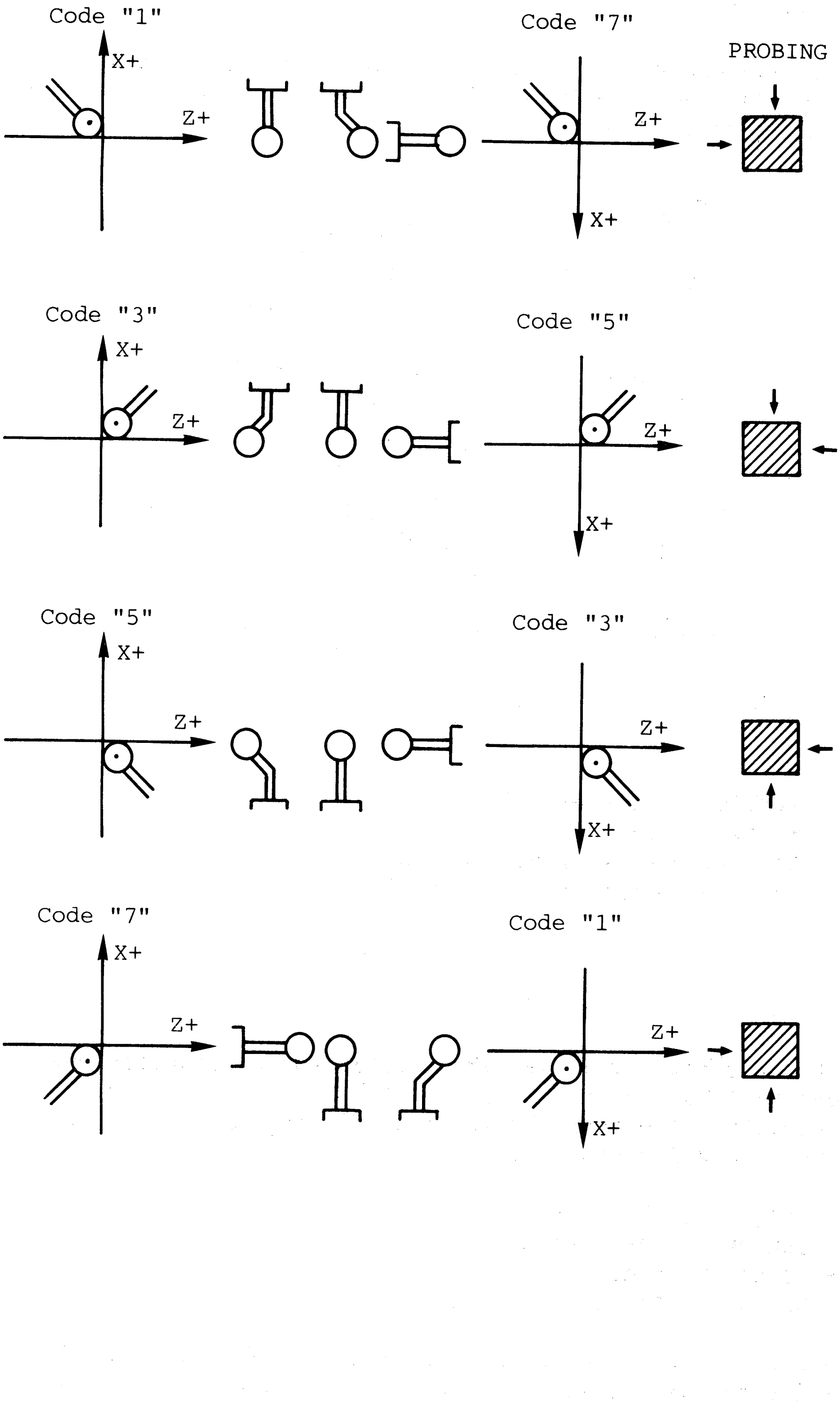


Machine-parameter P138 = P90 - 40
Machine-parameter P139 = P90
Machine-parameter P140 = P92 - 40
Machine-parameter P141 = P92

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

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

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


Code "1"
X+
Z+
Code "7"
Z+
X+
PROBING
Code "3"
X+
Z+
Code "5"
Z+
X+
Code "5"
X+
Z+
Code "3"
Z+
X+
Code "7"
X+
Z+
Code "1"
Z+
X+

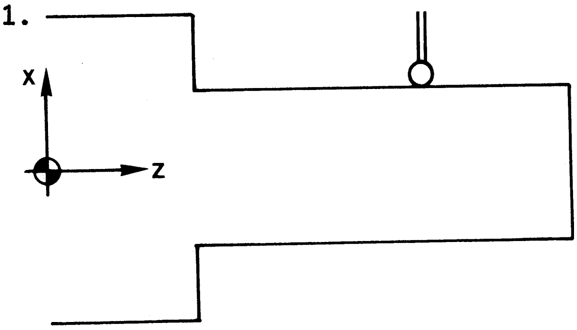
N2. PART MEASUREMET CYCLE FOR THE X AXIS

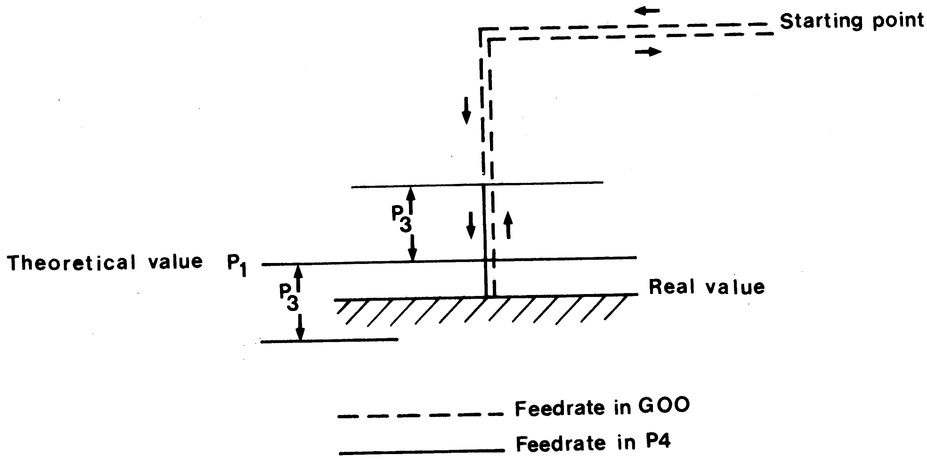
Cycle programming format:

G75 N2 P1= K-- P2= K-- P3= K-- P4= K--

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

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





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

P90 = Real value measured on the X axis.

P93 = Measurement error.

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

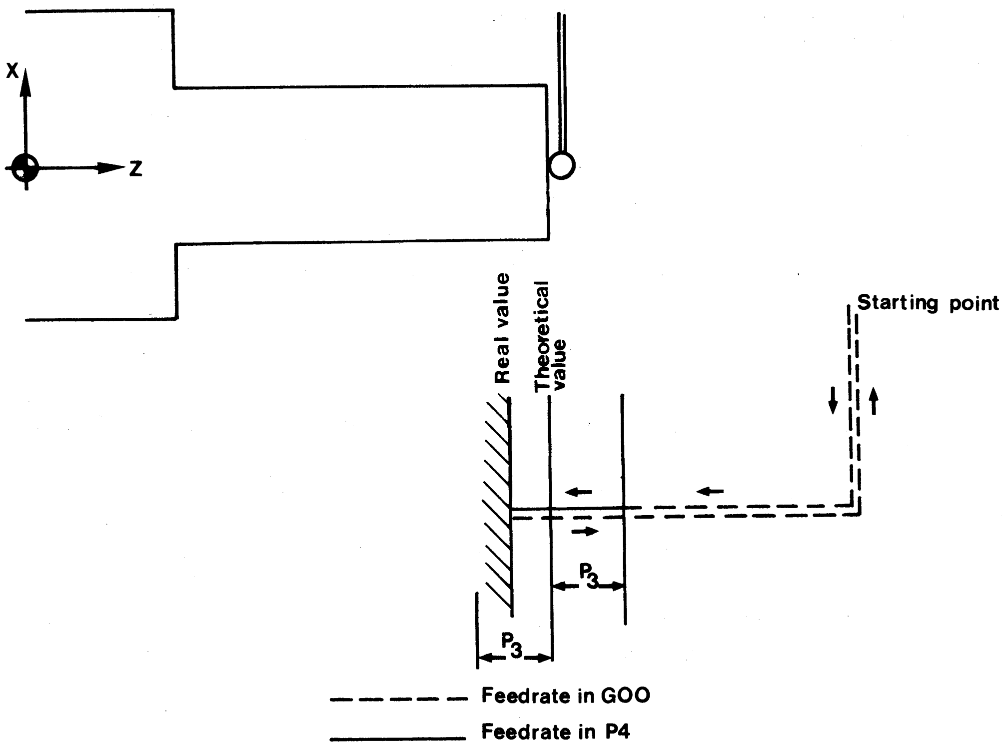
The values of **P93** will always be in diameters.

N3. PART MEASUREMENT CYCLE FOR THE Z AXIS.

Cycle programming format:

G75 N3 P1= K-- P2=K-- P3=K-- P4=K--

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



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

P92 = Real value measured on the Z axis.

P95 = Measurement error.

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

Cycle programming format:

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

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

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

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

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

Cycle programming format:

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

G75 N5 = Part measurement and tool calibration cycle code for the Z axis.

P1 = Theoretical X value.
P2 = Theoretical Z value.
P3 = Safety distance.
P4 = Probing feedrate.
P5 = Tolerance.
P6 = Corrector number for the tool to be calibrated.

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

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

6.23. G76. AUTOMATIC BLOCK GENERATION

This function is used to generate blocks that are automatically loaded into the CNC or to a computer (via **DNC**).

If the new program is going to be loaded into the CNC, a block of the type **G76 P5** must be previously written.

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

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

The programming format is:

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

The contents of the block to be created are similar to the normal programming except that the preparatory functions G22 and G23 cannot be programmed.

After G76, the coordinates can be programmed in different ways:

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

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

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

Example:

```
N10 G76 P00345
N20 G76 G1 X F500 M3
N30 P2=P3 F2 K1
N40 G76 XP2 ZP5 M7
N50 G76 G0 X14 Z20 M5
```

and if in block 40 the parameter values are: P2=14.853 and P5=154.37, the CNC will generate the following program P00345.

```
N100 G1 X78.35 F500 M3
N101 X14.853 Z154.37 M7
N102 G0X14 Z20 M5
```

It is necessary to program all five digits of the program number in blocks of type **G76 P5** or **G76 N5**

The CNC must be in **DNC ON** (operating mode 7) in order to load the new program into a computer (see DNC manual).

If the number of the program to be generated exists already in memory (e.g. **P12345**) it must be in the last position of the program map; but if **G76 P12345** is executed, the old program is erased and the new one can be generated.

When the program number exists in memory but is not the last one in the memory map, the CNC will issue error 56.

NOTE:

When a program is edited it goes to the last position in memory map and when it is executed it goes to the first position.

When a program is being generated, another program cannot be generated until the generation of the previous one is cancelled by means of **M2, M30, RESET** or **EMERGENCY**.

Some of the applications of the G76 function are, for example, the creation of a program after the calculation of a trajectory by means of a parametric program, or the **DIGITIZING** of a model with a measuring probe (G75) generating a point-to-point program as large as desired.

Example G76: DIGITIZING

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

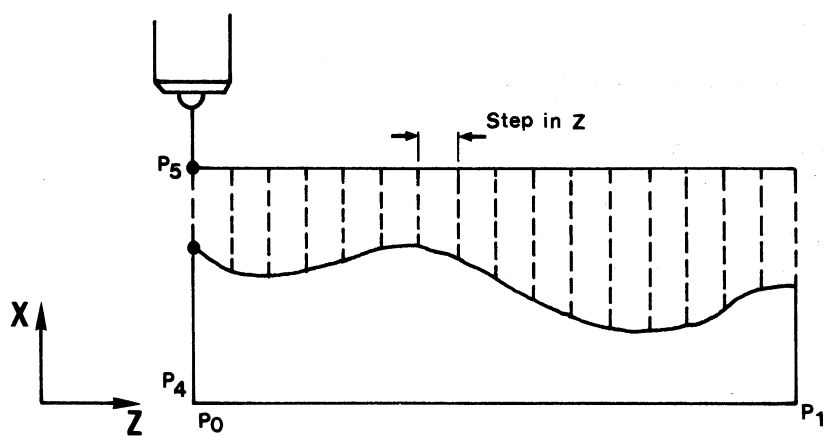
Calling parameters:

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

Parameters used for calculations

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

```
N10  G76 N12345                 -------->  (Program to be loaded
                                            into computer)
N20  G76 G1 F500

N30  P0=K-- P1=K--                         |(Parameter
     P4=K-- P5=K-- P6=K--                  | definition)

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

N50  G26 N80

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

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

N90  G0 G5 G90 XP5 ZP0

N100 P14=K0                     -------->  (P14=Step counter on Z)

N110 G90 G75 XP8                -------->  (Probing on X)

N120 G76 X Z                    -------->  (Load values)

N130 G0 XP5                     -------->  (Withdrawl on X)

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

N150 G28 N180

N160 G91 ZP6                    -------->  (Next step on Z)

N170 G25 N110

N180 G0XP13ZP11                 -------->  (Back to initial point)

N190 M30
```



After the execution of this program, the CNC will have generated and loaded into the computer the following P12345 program:

```
N100 G1 F500
N101 X-- Z--
N102 X-- Z--
N103 X-- Z--
N--  X-- Z--
Etc.
```

If the machining must be done in various passes, the program will have to be executed applying successive zero-offsets or changes in tool length compensation.

All preparatory functions (square corner, scaling factor) that will affect the whole program can be defined in a previous block. The CNC reserves automatically 100 blocks.

Geometrical functions can also be included in a G76 type block:

. G08 Arc tangent to the previous path.
. G09 Arc defined by three points.

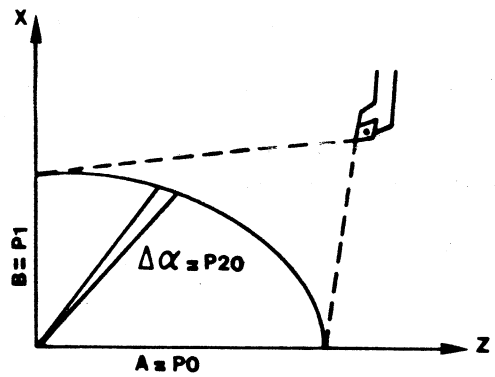
With these functions is possible to smoothen the point-to-point machining profile.

Example G76

This is a parametric program which, when executed, will calculate the different points of an ellipse and load them into a new program by means of G76 for later machining.

The calling parameters are the following:

P0 = Half the long axis (A).
P1 = Half the short axis (B).
P3 = Starting point's angle.
P20= Angular increment.



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

Z = P0 SIN P3
X = P1 COS P3

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

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

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

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

6.24. G90, G91. Absolute programming. Incremental programming

The coordinates of a point can be programmed either with absolute coordinates (G90) or with incremental coordinates (G91).

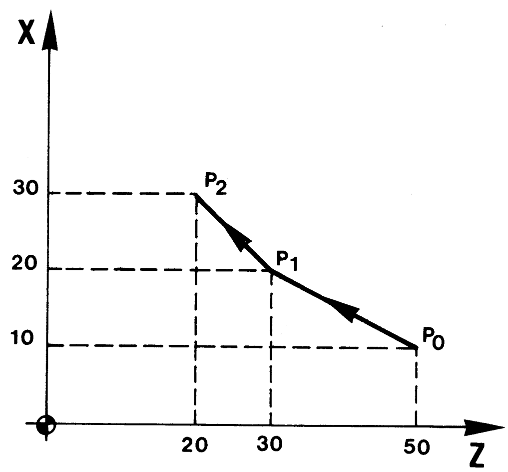
When operating on G90, the coordinates of the point programmed are referred to the coordinate origin point.

When operating on G91, they are referred to the path's previous point, so the programmed values define the distance to go along the relevant axis.

After turning on and having executed **M02**, **M30**, **EMERGENCY** or **RESET**, the CNC assumes the function G90.

The functions G90 and G91, are incompatible each other when being in the same block.

Examples: Let us suppose the X is programmed in diameters and that the starting point is P0(X20 Z50).

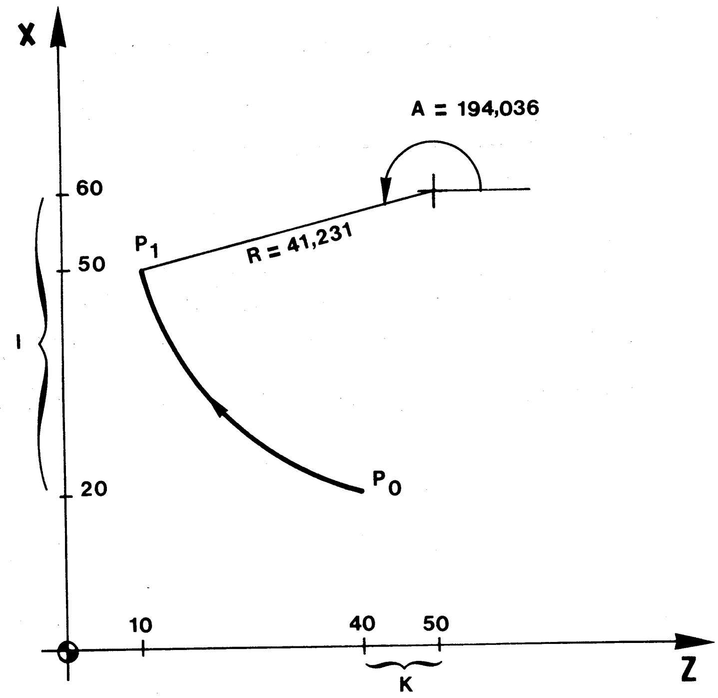


Absolute programming G90

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

Incremental programming G91

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



Starting point P0(X40 Z40)

Absolute programming G90

N100 G90 G02 X100 Z10 I40 K10

or

N100 G90 G02 X100 Z10 R41.231

Incremental programming G91

N100 G91 G02 X60 Z-30 I40 K10

or

N100 G91 G02 X60 Z-30 R41.231

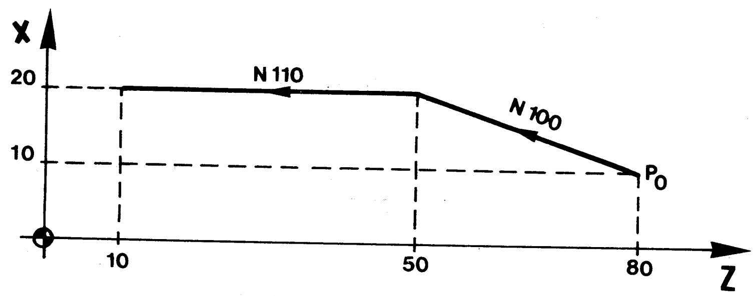
6.25. G92. Preselection of coordinate values and setting of max. S value at constant surface speed G96

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

a) Preselection of coordinate values

When function G92 is programmed, there is no movement of the axes, and the CNC accepts the values of the axes programmed after G92 as the new coordinate values of those axes.

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



The program to describe the path drawn will be:

```
N100 G01 G90 X40 Z50
N110             Z10
```

When using the function G92, the program will be:

```
N90  G92 X20 Z0 --> The point X20 Z0 replaces the point P0
N100 G90 X40 Z-30
N110         Z-70
```

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

Coordinate values preselection by G92 is referred always to the theoretical position in which the axes are; i.e. if G92 is executed while tool compensation is active, the preselected coordinate value is corrected by the compensation value.

b) Setting of max. spindle S value

By means of the block N4 G92 S4 the max. spindle speed is limited to the value set by S4, when working on G96 (constant surface speed). The CNC will not accept the programming of S values higher than this.

This limit can neither be overrun when operating in constant surface speed nor by using the spindle override keys. S4 is programmed in rev./min.

6.26. G93. Preselection of polar origin

Function G93 can be used to preselect any point as the origin of polar coordinates.

There are two ways of preselecting an origin of polar coordinates:

a) G93 I+/-4.3 K+/-4.3 (mm)(Always absolute coordinate values)
I+/-3.4 K+/-3.4 (inches)

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

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

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

b) The programming of G93 in a block determines that, prior to the movement programmed, the actual position of the tool becomes the polar origin.

NOTE:

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

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

6.27. G94. Feedrate F in mm/min (inches/min)

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

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

6.28. G95. Feedrate F in mm/rev. (inches/rev.)

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

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

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

6.29. G96. S speed in m/min. (feet/min.) at constant surface speed

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

It is recommended that G96 and S4 spindle speed be programmed in the same block.

G96 must be programmed in the same block as the spindle range (M41,M42,M43,M44) is it has not been programmed before.

If S4 is not programmed in the same block as G96, the CNC will assume as spindle speed in **constant surface speed** mode, the last value used in this mode.

The CNC will generate error 10 if G96 or the spindle range has not been previously programmed.

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

G00 movements prior to G01,G02 or G03, if any, will not affect this calculation.

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

6.30. G97. S speed in rev./minute

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

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

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

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

7. PROGRAMMING OF COORDINATE VALUES

A point can be programmed in the CNC 8020 T using different data:

- cartesian coordinates
- polar coordinates
- Two angles
- One angle and one cartesian value

7.1. Cartesian coordinates

The format of the axis coordinate values is as follows:

. In mm

X+/-4.3 Z+/-4.3

. In inches:

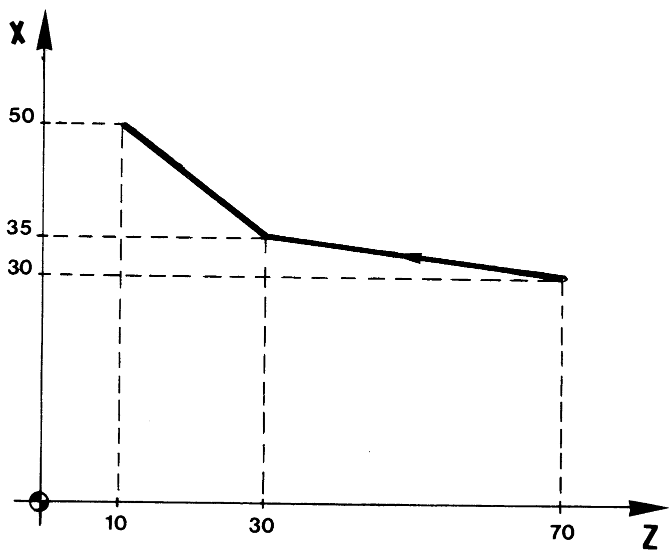
X+/-3.4 Z+/-3.4

In other words, the axis coordinate values are programmed by the letters X,Z followed by the coordinate values.

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

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

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



Absolute coordinate values

```
N100 G90 X70  Z30
N110      X100 Z10
```

Incremental coordinate values

```
N100 G91 X10 Z-40
N110      X30 Z-20
```

7.2. Polar coordinates

The format of the axis coordinate values is as follows:

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

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

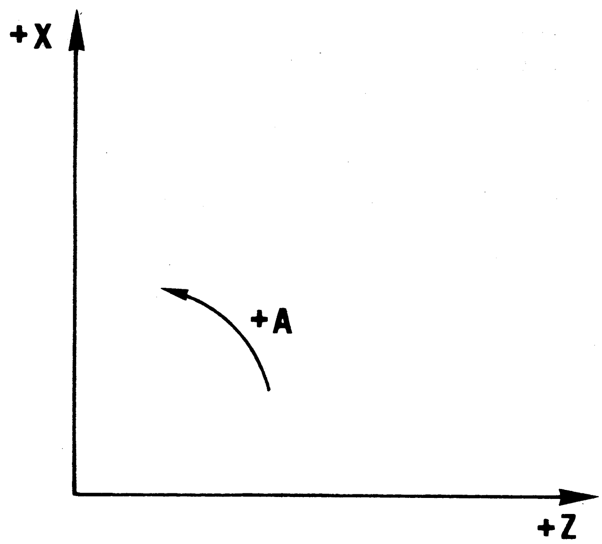
R being the value of the radius and A the value of the angle (A in degrees), referred to the polar center.

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

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

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

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

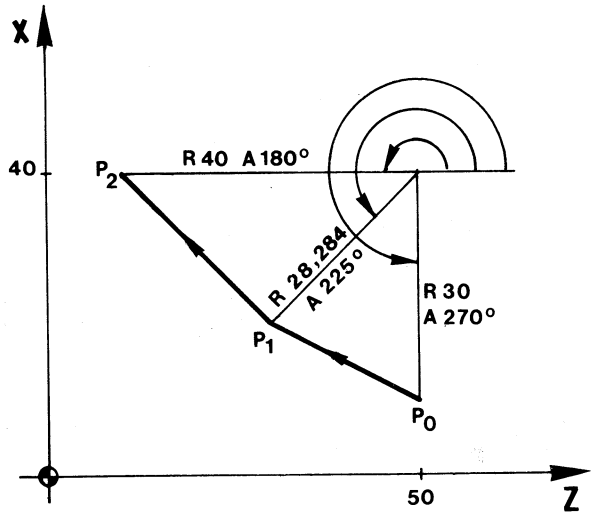


When working with polar coordinates, the center of the circle on circular interpolations (G02,G03) is defined with I,K same as working with cartesian coordinates.

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

Examples: X in diameters

1.



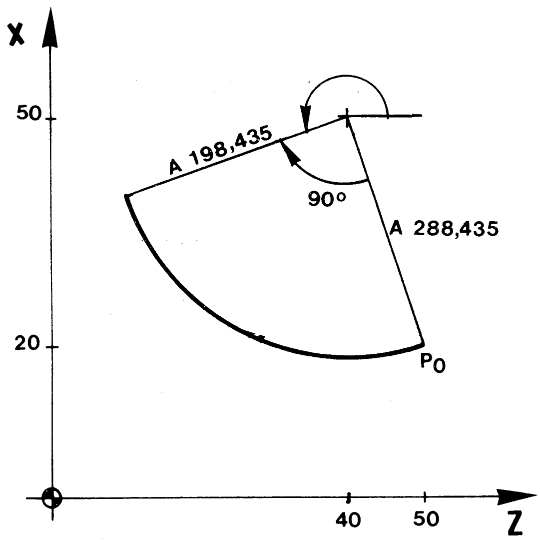
In absolute coordinate values G90

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

In incremental coordinate values G91

```
N100 G93 I80 K50              Preselection of polar origin
N110 G01 G90 R30     A270 --> P0
N120     G91 R-1.76  A-45 --> P1
N130         R11.716 A-45 --> P2
```

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



In absolute coordinate values G90

N100 G90 G02 A198.435 I30 K-10

or

In incremental coordinate values G91

N100 G91 G02 A-90 I30 K-10

or

N100 G93 I100 K40
N110 G91 G02 A-90

7.3. Two angles (A1,A2)

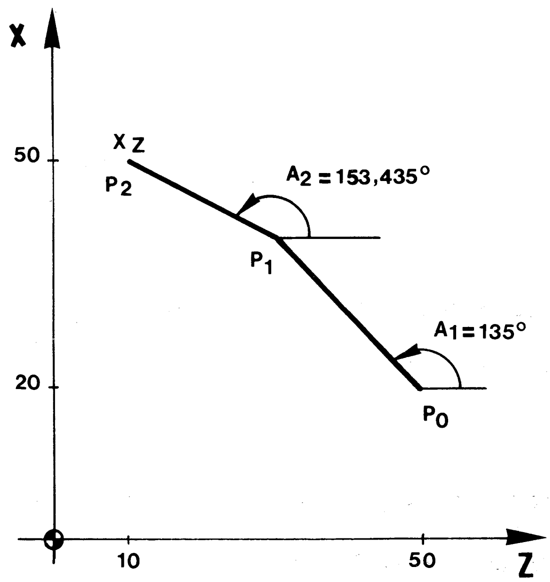
An intermediate point in a path can also be defined by: A1 A2 (X,Z).

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

A2 is the departure angle of the intermediate point P1.

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

The CNC calculates automatically the coordinates of P1.



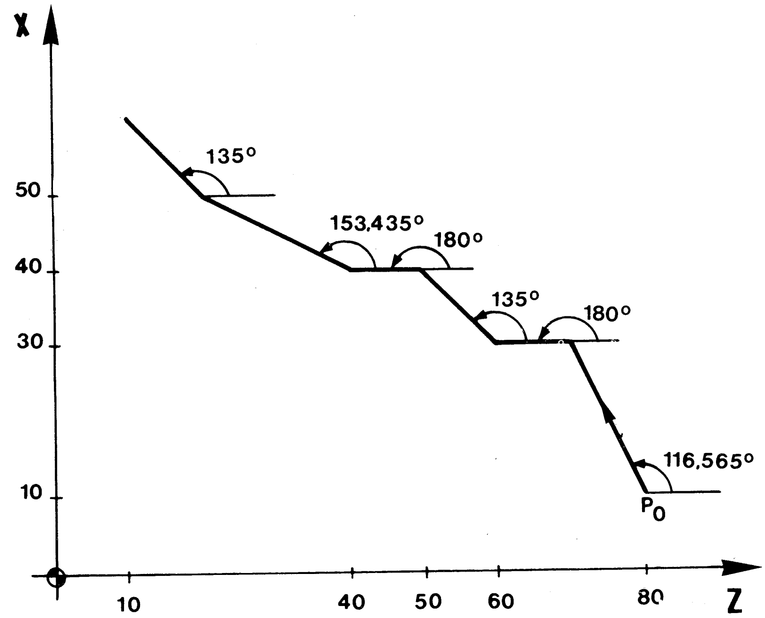
Let us suppose that the starting point is P0 (X40 Z50) and the X axis is in diameters.

```
N100 A135 A153.435
N110 X100 Z10
```

7.4. One angle and one cartesian value

A point can also be defined by an angle (A, see fig.) and a cartesian value.

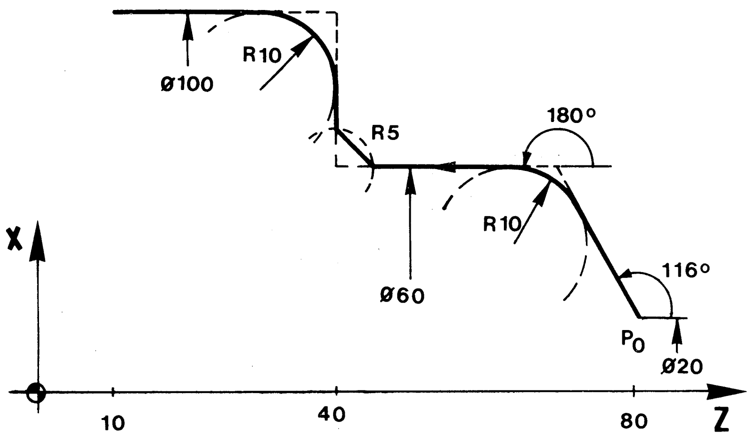
Starting in X70 Z0, the points are defined as follows:



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

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

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



Starting point P0 (X20 Z80)

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

8. F. FEEDRATE PROGRAMMING

F (programmable feedrate) has different meanings whether we are working in G94 or G95 and whether we are working in mm or in inches. The differences are indicated in the table below.

G code	Format in mm	Program unit mm	Maximum value mm	Format inches	Program unit inch	Maximum Value inch
G94	F4	F1= 1mm/min.	F9999= 9999 mm/min.	F4	F1= 0.1 inch/min.	F3937= 393.7 inch/min.
G95	F3.4	F1= 1mm/rev.	F500= 500 mm/rev.	F2.4	F1= 0.1 inch/rev.	F19.685= 19.685 inch/rev.

The machine's actual maximum feedrate may be limited to a lower value (see instruction book of the machine).

The machine's maximum working feedrate can be programmed directly or by using code F0.

Example:

On a machine with a maximum programmable working feedrate of 3000 mm/min. it makes no difference whether F3000 or F0 is programmed.

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

When operating on positioning (G00), the machine will move in rapid regardless of the F programmed.

The rapid speed is set for each axis during the final adjustment of the machine, the maximum possible value being 30 m/min. (See instruction book of the machine).

The programmed feedrate can be varied between 0% and 120% or between 0% and 100% depending on P94(3) by means of the knob on the front panel of the CNC unless a threadcutting operation is being performed with G33, G86 or G87.

9. S. SPINDLE SPEED AND SPINDLE ORIENTATION

Code S can have two different meanings:

a) Spindle speed

The spindle speed is programmed directly in rev./min. or m/min. (feet/min.) by means of code S4 m/min. (feet/min.) must be programmed when constant surface speed is required.

Any value may be programmed between S0 and S9999; i.e. between 0 and 9999 rev./min. This value is limited by the max. speed permitted by the machine, this limit is set by a machine parameter.

The instruction book of the machine must be consulted in each particular case.

The controls on the front panel of the CNC may be used to achieve between 50 and 120% variation in programmed spindle speed.

The possible values of S are S0-S3047 (0 m/min. to 3047 m/min.) or, S0-S 9999 (0 feet/min. to 9999 feet/min.).

b) Spindle orientation

When after code M19,S4.3 is programmed, S4.3 identifies the position of the spindle in degrees, referred to the reference zero marker. The CNC will send an analog S signal defined by machine-parameters P100(2) and P112 until the spindle is positioned at the point determined by S4.3.

10. T. TOOL PROGRAMMING

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

- **Tool number**. The two digits to the left of the decimal point may have any value between 00 and 99. This value is used for selecting the required tool in the case of a machine with automatic turret, and may be limited to a value lower than 99 according to parameter P106.

- **Tool compensation (tool offset table)**. The two digits to the right of the decimal point may have any value between 01 and 32.

  With these figures the desired values are selected in the tool offset table.

  As soon as T2.2 is read, the CNC applies the X,Z,I,K values stored in the table except when P98(5) is 1, in which case these values will be applied after an **M06**.

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

  If no T has been programmed, the CNC applies the address 00.00, which corresponds to a tool of zero dimensions.

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

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

The max. values are:

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

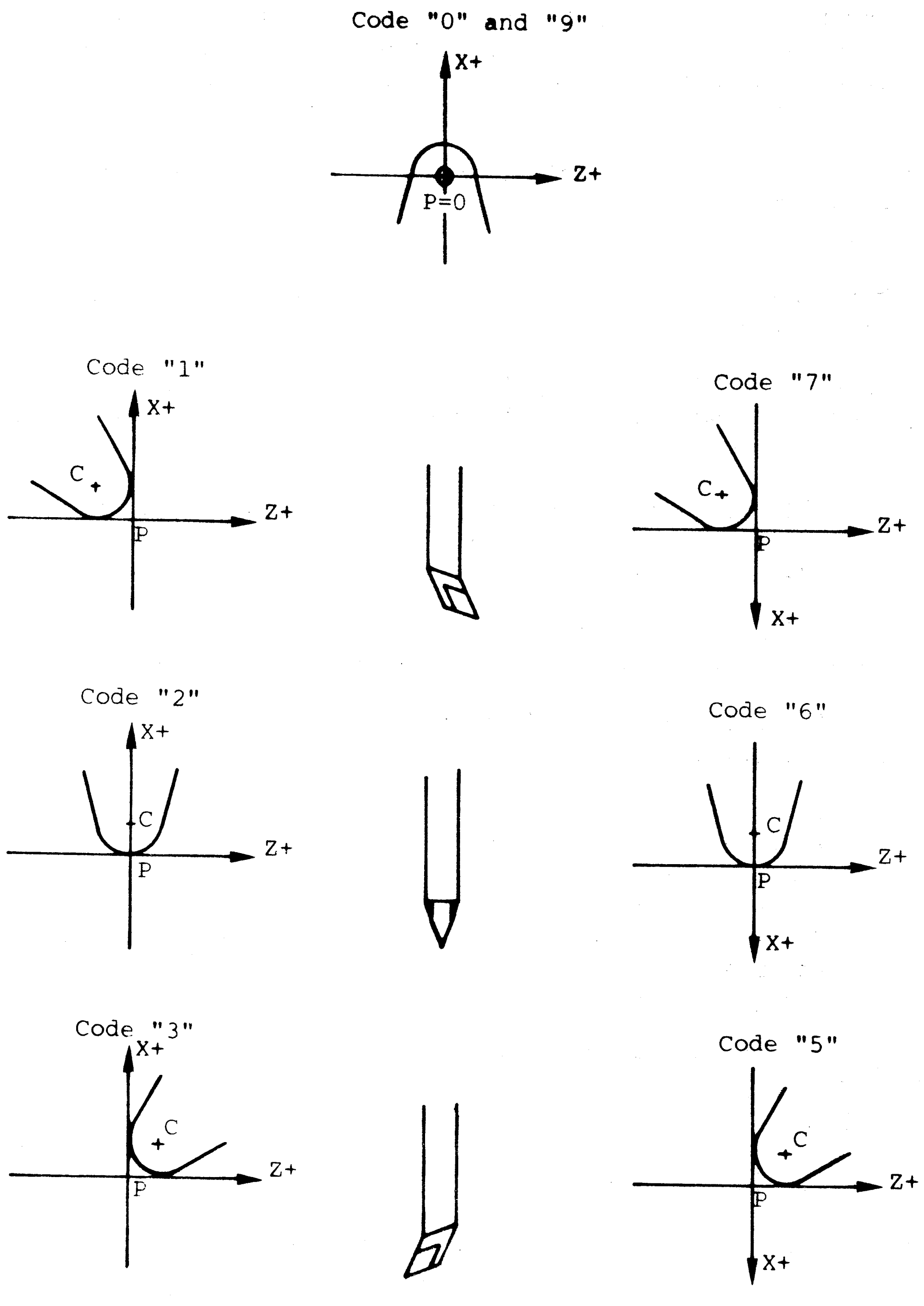
I,K (tool length offset) +/-32.766 mm (+/-1.2900 inches).

R (Radius) 1000.000 mm (39.3700 inches).

To offset the tool nose radius the location code (F) of the tool must also be stored.

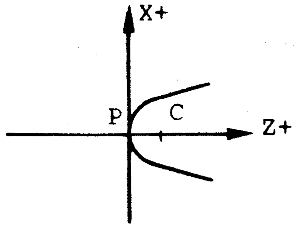
Possible codes are: F0-F9 (see figure).

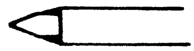
LOCATION CODES



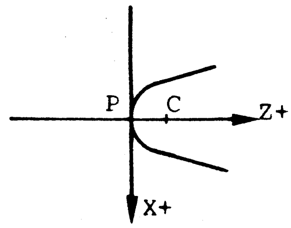
P : Tool tip
C : Tool nose radius center

Code "4"

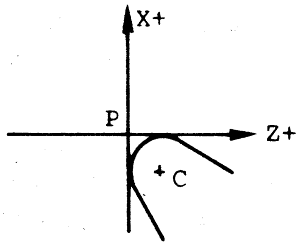


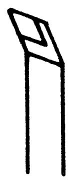


Code "4"

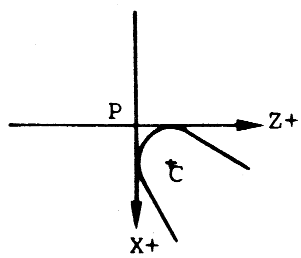


Code "5"

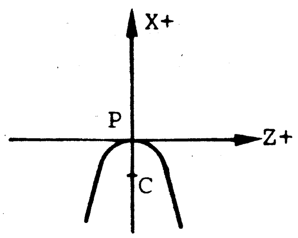




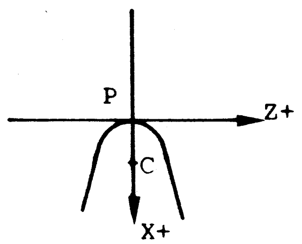
Code "3"



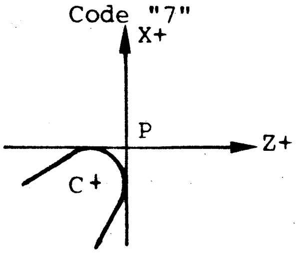
Code "6"



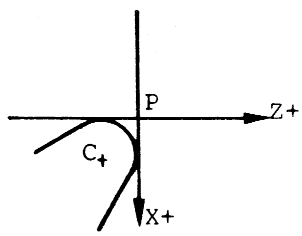
Code "2"



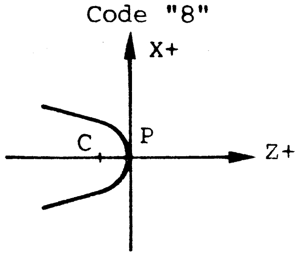
Code "7"

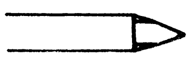


Code "1"

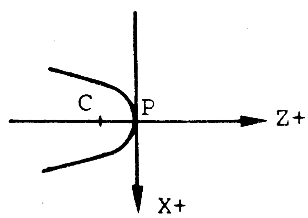


Code "8"





Code "8"



11. M. MISCELLANEOUS FUNCTIONS

The miscellaneous functions are programmed by means of code M2.

96 different miscellaneous functions (M00-M99) can be programmed except M41,M42,M43,M44 pegged to the S function if P95(1) is set to 1. If P95(1) is set to 0, M41,M42,M43 must be programmed.

The miscellaneous functions come out in BCD code.

The CNC also has 15 decoded outputs for miscellaneous functions. These outputs will be assigned to the required functions during the final adjustment of the CNC to the machine.

The miscellaneous functions which are not assigned a decoded output are always performed at the beginning of the block in wich they are programmed.

In assigning a decoded output to an miscellaneous functions, a decision is also made as to whether it is to be performed at the beginning or at the end of the block in which it is programmed.

Up to a maximum of seven miscellaneous functions may be programmed in one block.

When more than one miscellaneous function is programmed in a block, the CNC performs then consecutively in the order in which they are programmed.

Some of the 100 miscellaneous functions have an internal meaning assigned to them in the CNC.

11.1. M00. Program stop

When the CNC reads code M00 in a block, it halts the program. The key has to be pressed for it to resume.

It is recommended that this function be set in the table of decoded M functions so that it is performed at the end of the block in which it is programmed (see Installation and Start up Manual).

11.2. M01. Conditional stop of program

Same as M00 except that the CNC only takes it into account if the **conditional stop** input is activated.

11.3. M02. End of program

This code indicates end of program and performs a **general reset** function of the CNC (reversion to initial state). It also acts as an M05.

As in the case of M00, it is recommended that this function be set so that it is executed at the end of the block in which it is programmed.

11.4. M30. End of program with return to beginning

Same as M02 except that the CNC goes back to the first block at the beginning of the program. It also acts as an M05.

11.5. M03. Clockwise start of the spindle

This code means that the spindle starts running clokwise. It is recommended that this function be set so that it is executed at the beginning of the block in which it is programmed.

11.6. M04. Counter-clockwise start of the spindle

Same as M03 except that the spindle rotates in the opposite direction.

11.7. M05. Spindle stop

It is recommended that this be set so that the CNC executes it at the end of the block in which it is programmed.

11.8. M19. Spindle orientation

If M19 S4.3 is programmed, the spindle will rotate at the speed and direction set by machine-parameters P100(2) and P112 and will stop at the point identified by S4.3 in degrees, referred to the reference zero marker.

When the spindle is positioned within the dead band (P113), the CNC sends out the spindle blocking signal (decoded M15) and the spindle is kept in closed loop; gain as per P114, min. analog voltage as per P115.

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

11.9. M41,M42,M43,M44 Spindle range selection

When P95(1) has been set to 1, these codes are automatically generated by the CNC, when an S function is programmed. If this parameter is 0, M41,M42,M43 and M44 must be programmed.

When operating at constant surface speed (G96) these functions must **necessarily** be programmed even if P95(1) is set to 1.

11.10. M45. Selection of rotation speed of the LIVE TOOL and that of the SYNCHRONIZED TOOL.

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

a) LIVE TOOL

Programming format:

N4 M45 S+/-4

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

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

b) SYNCHRONIZED TOOL (with the rotation speed of the spindle)

To take advantage of this feature, an encoder must be conected to the live tool. This encoder will be connected to the CNC via the feedback connector **A2**.

Programming format: **N4 M45 K+/-3.4**

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

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

Example:

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

```
N - P1=K1 F4 K3
N - M45 KP1
```

If the rpm's are greater than the limit established by machine-parameter P126, the CNC will display error 17. Also, when the following error of the synchronized tool is too large, the CNC will display error 71.

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

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

11.9. M41,M42,M43,M44 Spindle range selection

When P95(1) has been set to 1, these codes are automatically generated by the CNC, when an S function is programmed. If this parameter is 0, M41,M42,M43 and M44 must be programmed.

When operating at constant surface speed (G96) these functions must **necessarily** be programmed even if P95(1) is set to 1.

11.10. M45. Selection of rotation speed of the LIVE TOOL and that of the SYNCHRONIZED TOOL.

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

a) LIVE TOOL

Programming format:

N4 M45 S+/-4

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

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

b) SYNCHRONIZED TOOL (with the rotation speed of the spindle)

To take advantage of this feature, an encoder must be conected to the live tool. This encoder will be connected to the CNC via the feedback connector **A2**.

Programming format: **N4 M45 K+/-3.4**

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

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

Example:

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

```
N - P1=K1 F4 K3
N - M45 KP1
```

If the rpm's are greater than the limit established by machine-parameter P126, the CNC will display error 17. Also, when the following error of the synchronized tool is too large, the CNC will display error 71.

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

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

12. STANDARD AND PARAMETRIC SUBROUTINES

A subroutine is a part of a program which is suitably identified and can be called in for execution from any position in a program.

A subroutine may be called in several times from different positions in the program or from different programs.

A single call can be used to repeat the execution of a subroutine up to 255 times.

A subroutine may be stored in the memory of the CNC as an independent program or as part of a program.

Standard and parametric subroutines are basically identical. The difference between them is that up to 15 parameters can be defined in the call block (G21 N2.2) of parametric subroutines.

In standard subroutines the parameters cannot be defined in the call block (G20 N2.2).

The max. number of parameters of a subroutine (standard or parametric) is 100 (P0-P99).

12.1. Identification of a standard subroutine

A standard (non-parametric) subroutine always begins with a block which contains function G22. The structure of the subroutine opening block is: N4 G22 N2

N4 : Block number
G22: Defines the beginning of a subroutine
N2 : Identifies the subroutine (may be any number between N0 and N99)

This block cannot contain additional information.

NOTE: Two subroutines having the same identification number but belonging to different programs cannot be present at the same time in the memory of the CNC, although a standard subroutine and a parametric subroutine may be identified by the same number.

The subroutine opening block is followed by programming the blocks required.

A standard subroutine can contain parametric blocks.

```
Example: N0   G22   N25
         N10  X20
         N15  P0    F1  P1
         N20  G24
```

A subroutine must always end with a block of the form N4 G24.

N4 : Block number
G24: End of subroutine

No other additional information can be programmed in that block.

12.2. Calling in a standard subroutine

A standard subroutine may be called in from any program or other subroutine (standard or parametric). Calling in a standard subroutine is achieved by function G20.

The structure of a call block is: N4 G20 N2.2

N4 : Block number
G20 : Subroutine call
N2.2: The two figures to the left of the decimal point identify the number of the subroutine called in (00-99).

The two figures to the right of the decimal point indicate the number of times the subroutine is to be repeated (00-99).

If a parameter is programmed instead the two figures on the left of the decimal point, it can have a value between 0 and 255.

No other additional information can be programmed in the block calling in a standard subroutine.

12.3. Parametric subroutines

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

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

12.3.1. Identification of a parametric subroutine

A parametric subroutine always starts by G23.

The structure of the first block is:

N4 G23 N2

N4 : Block number
G23 : Defines the start of a parametric subroutine
N2 : It identifies the parametric subroutine (it may be any number between 00 and 99).

NOTE:

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

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

N4 : Block number

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

No additional information can be programmed in this block.

12.4. Calling a parametric subroutine

A parametric subroutine can be called from a main program or from another subroutine (standard or parametric).

The calling of a parametric subroutine is carried out by means of function G21.

The structure of the call block is:

N4 G21 N2.2 P2=K+/-5.4 P2=K+/-5.4

N4 : Block number

G21 : Call code for a parametric subroutine

N2.2: The two numbers to the left of the decimal point identify the number of the parametric subroutine being called (00-99). The two figures to the right indicate the number of times the subroutine is to be repeated (00-99). If a parameter is programmed instead of the two figures on the left of the decimal point, it can have a value between 0 and 255.

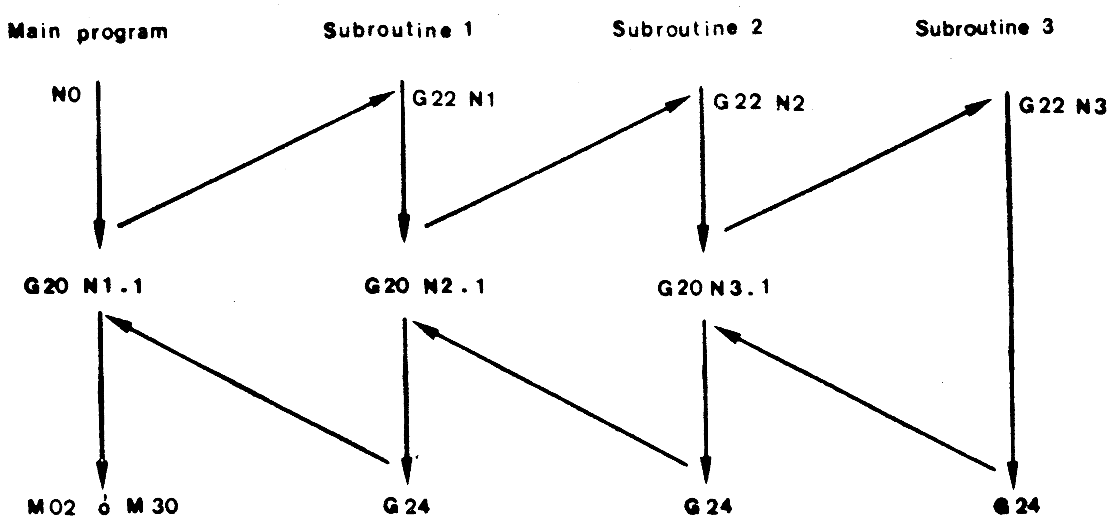
P2 : Number of the parameter (P00-P99).
K+/-5.4: Values given to the parameters.

NOTE: The symbol = is written with the key *.

12.5. Levels of nesting

From a main program or from a subroutine (standard or parametric) it is possible to call in a subroutine, from the latter a second subroutine, from the second a third, and so on up to a maximum of 15 levels of nesting. Each level may be repeated 255 times.

Subroutine sequencing diagram



13. PARAMETRIC PROGRAMMING. OPERATIONS WITH PARAMETERS

The CNC 8020 T has 100 parameters (P0-99) with which the following operations can be performed.

- Programming of parametric blocks
- Different operations
- Jumps within a program

The parametric blocks can be written in any part of the program.

In the **AUTOMATIC, SINGLE BLOCK, TEACH-IN** and **DRY RUN** operating modes, when being in display mode 3, the **P** key is depressed, the parameters will be displayed.
29 operations (F1-F29) can be carried out with parameters.

F1 : Addition
F2 : Subtraction
F3 : Multiplication
F4 : Division
F5 : Square root
F6 : Square root of the addition of the squares
F7 : Sinus
F8 : Cosinus
F9 : Tangent
F10 : Arc tangent
F11 : Comparison
F12 : Entire part
F13 : Entire part plus one
F14 : Entire part minus one
F15 : Absolute value
F16 : Complementation
F17 : Special functions
F18 : Special functions
F19 : Special functions
F20 : Special functions
F21 : Special functions
F22 : Special functions
F23 : Special functions
F24 : Special Functions
F25 : Special functions
F26 : Special functions
F27 : Special functions
F28 : Special functions
F29 : Special functions

The use of the parameter is described next:

Assignments

Any value can be assigned to a parameter.

a) N4 P1 = P2

The indicates that P1 takes the value of P2, while P2 keeps the value it had.
NOTE: The key * is used to program =

b) N4 P1 = K1.5

P1 takes the value 1.5

K identifies a constant. Constants can have values comprised between +/-0.0001 and +/-99999.9999.

c) N4 P1 = X

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

d) N4 P1 = Z

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

e) N4 P1 = R

P1 takes the value 1 if P11 (radius/diameter) is set to radius and the value 2 if in diameters.

f) N4 P1 = T

P1 takes the actual position of the clock (execution time) in hundredths of asecond. This assignation means the cancellation of radius compensation (G41 or G42).

Operations

F1 Addition

Example: N4 P1 = P2 F1 P3

P1 takes the value of the addition of P2 and P3;
i.e. P1= P2 + P3.

N4 P1 = P2 F1 K2 can also be programmed; i.e. P1 takes the value of P2 + 2.

The letter K identifies a constant for instance:

K1 means **value 1**
K1000 means **value 1000**

The same parameter can be as an addend and as the result; i.e.,

N4 P1 = P1 F1 K2

This means that P1 = P1 + 2

NOTE: The key * is used to write =

F2 Subtraction

```
N4 P10 = P2 F2 P3 -->  P10 = P2 - P3
N4 P10 = P2 F2 K3 -->  P10 = P2 - 3
N4 P10 = P10 F2 K1 -> P10 = P10 - 1
```

F3 Multiplication

```
N4 P17 = P2 F3 P30 -->  P17 = P2 x P30
N4 P17 = P2 F3 K4  -->  P17 = P2 x 4
N4 P17 = P17 F3 K8 -->  P17 = P17x 8
```

F4 Division

```
N4 P8 = P7 F4 P35 -->  P8 = P7 : P35
N4 P8 = P2 F4 K5  -->  P8 = P2 : 5
N4 P8 = P8 F4 K2  -->  P8 = P8 : 2
```

F5 Square root

N4 P15 = F5 P23 --> P15 $=\sqrt{P23}$
N4 P14 = F5 K9 --> P14 $=\sqrt{9}$
N4 P18 = F5 P18 --> P18 $=\sqrt{P18}$

F6 Square root of the addition of the squares

N4 P60 = P2 F6 P3 --> P60 $=\sqrt{P2^2 + P3^2}$
N4 P50 = P40 F6 K5 --> P50 $=\sqrt{P40^2 + 5^2}$
N4 P1 = P1 F6 K4 --> P1 $=\sqrt{P1^2 + 4^2}$

F7 Sinus

N4 P1 = F7 P2 --> P1 = Sin P2

The angle has to be programmed in degrees.

N4 P1 = F7 K5 --> P1 = Sin 5 degrees

F8 Cosinus

```
N4 P1 = F8 P2  -->  P1 = Cos P2
N4 P1 = F8 K75 -->  P1 = Cos 75 degrees
```

F9 Tangent

```
N4 P1 = F9 P2  -->  P1 = tg P2
N5 P1 = F9 K30 -->  P1 = tg 30 degrees
```

F10 Arc tangent

```
N4 P1 = F10 P2   -->  P1 = arc tg P2 (result in degrees)
N4 P1 = F10 K0.5 -->  P1 = arc tg 0.5
```

F11 Comparison

It compares different parameters, or a parameter and a constant, and activates the conditional jumps flags. Its application will be described in the conditional jumps section.

```
N4 P1 = F11 P2
```

```
If P1 = P2 the if zero jump flag is activated
If P1 => P2 the if => zero jump flag is activated
If P1 =< P2 the if =< zero jump flag is activated
```

N4 P1 = F11 K6 can also be programmed.

F12 Entire part

```
N4 P1 = F12 P2   -->  P1 takes the entire part of P2 as its
value
N4 P1 = F12 K5,4 -->  P1 = 5
```

F13 Entire part plus one

```
N4 P1 = F13 P2   -->  P1 takes the entire part of P2 plus one as
                      its value
N4 P1 = F13 K5,4 -->  P1 = 5 + 1 = 6
```

F14 Entire part minus one

```
N4 P1 = F14 P27  -->  P1 takes the entire part of P2 minus one
                      as its value
N4 P5 = F14 K5,4 -->  P5 = 5 - 1 = 4
```

F15 Absolute value

```
N4 P1 = F15 P2  -->   P1 takes the absolute value of P2
N4 P1 = F15 K-8 -->   P1 = 8
```

F16 Complementation

```
N4 P7 = F16 P20  -->  P7 takes the complemented value of P20;
                      i.e., P7 = -P20
N4 P7 = F16 K10  -->  P7 = -10
```

F17-F29 Special functions

They do not affect the jump flags.

F17

```
N4 P1 = F17 P2  --> P1 takes the value of the memory address in
                    which the P2 block is located

Example N4 P1 = F17 K12 --> P1 takes the value of the memory
                            address in which the block N12 is
                            located.
```

F18

```
N4 P1 = F18 P2 -->  P1 takes the value of the X coordinate value
                    in the block located at P2.
```

F18 does not accept a constant as operand.
Example: P1 = F18 K2 is not valid.

F19

```
N4 P1 = F19 P2 -->  P1 takes the value of the Z coordinate value
                    in the block located at P2.
```

F19 does not accept a constant as operand.
Example: P1 = F19 K3 is not valid.

F20

```
N4 P1 = F20 P2 -->  P2 takes the value of the memory address in
                    the block previous to the one defined by P2.
```

F20 does not accept a constant as operand.
Example: P1 = F20 K4 is not valid.

F21

```
N4 P1 = F21 P2 -->  P1 takes the value of the I coordinate value
                    which is in the block located at P2.
```

F21 does not accept a constant as operand.
Example: P1 = F21 K2 is not valid.

F22

```
N4 P1 = F22 P2 -->  P1 takes the value of the K coordinate value
                    which is in the block located at P2.
```

F22 does not accept a constant as operand.
Example: P1 = F22 K3 is not valid.

F23

N4 P1 =F23 ----------> P1 takes the value of the tool table that is being used at the moment.

F24

This function can be programmed in two different ways:

Example a) N4 P9 = F24 K2 --> P9 takes the **X** value that is located in the position **2** of the tool offset.

Example b) N4 P8 = F24 P12 --> P8 takes the **X** value that is located in the tool offset in the position indicated by parameter **P12**.

F25

This function can be programmed in two different ways:

Example a) N4 P15= F25 K16 --> P15 takes the **Z** value that is located in the position **16** of the tool table.

Example b) N4 P13= F25 P34 --> P13 takes the **Z** value that is in the tool table in the position indicated by parameter **P34**.

F26

This function can be programmed in two different ways:

Example a) N4 P6 = F26 K32 --> P6 takes the **F** value that is located in the position **32** of the tool table.

Example b) N4 P14= F26 P15 --> P14 takes the **F** value that is located in the tool table in the position indicated by parameter **P15**.

F27

This function can be programmed in two different ways:

Example a) N4 P90= F27 K13 --> P90 takes the **R** value that is located in the position 13 of the tool table.

Example b) N4 P28 = F27 P5 --> P28 takes the **R** value that is located in the tool table in the position inicated by parameter **P5**.

F28

This function can be programmed in two different ways:

Example a) N4 P17= F28 K10 --> P17 takes the **I** value that is located in the position **10** of the tool table.

Example b) N4 P19= F29 P63 --> P13 takes the **I** value that is located in the tool table in the position indicated by parameter **P63**.

F29

This function can be programmed in two different ways:

Example a) N4 P15= F29 K27 --> P15 takes the **K** value that is located in the position **27** of the tool table.

Example b) N4 P13= F29 P25 --> P13 takes the **K** value that is located in the tool table in the position indicated by parameter **P25**.

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

Jumps/calls within a program

Functions G25,G26,G27,G28 and G29 can be used to jump to another block of the current program.

There are two possibilities:

a) N4 (G25,G26,G27,G28,G29) N4

N4 : Block number
G25,G26,G27,G28,G29 : Codes for different jumps
N4 : Number of the block the jump is aimed at.

When the CNC reads this block, it jumps to the targeted block and the program continues.

Example:

```
N0  G00  X100
N5            Z50
N10 G25  N50
N15      X50
N20      Z70
N50 G01  X20
```

When the block 10 is reached the CNC jumps to block 50 and then the program continues until it is finished.

b) N4 (G25,G26,G27,G28,G29) N4.4.2.

N4 : Block number
G25,G26,G27,G28,G29 : Codes for different jumps

```
N4.4.2 --> Number of repetitions
 | |-----> Number of the last block to be executed
 |-------> Number of the block to which the jump is targeted
```

When the CNC reads such a block, it jumps to the block identified between the N and the first decimal point. It then executes the section of the program between this block and the one identified between the two decimal points as many times as set by the last digit. This last digit can take a value within 0 and 99, unless it is programmed using a parameter in which case the limits are 0 and 255.

If only N4.4 is written the CNC will assume N4.4.1.

When the execution of this section is finished, the CNC goes to the block next to the one in which G25 N4.4.2. was programmed.

Example:

```
N0  G00  X10
N5       Z20
N10 G01  X50  M3
N15 G00  Z0
N20      X0
N25 G25       N0.20.8
N30 M30
```

When block 25 is reached, the CNC will jump to block o and will execute 8 times the section N0-N20. On completion of this, it will go to block 30.

G25 Non-conditional jump/call

As soon as the CNC reads code G25, it jumps to the block identified by N4 or N4.4.2.

Programming

N4 G25 N4 or N4 G25 N4.4.2

G25 must stand alone in a block.

Two **flags** can be activated according to the result of the following operation:
F1,F2,F3,F4,F5,F6,F7,F8,F9,F10,F11,F12,F13,F14,F15,F16.

The assignments do not affect the state of these flags.

Flag 1 (zero, equal)

If the result of an operation is zero, flag 1 is activated.

If the result of an operation is not zero, flag 1 is not activated.

If the result of a comparison is **equal**, flat 1 is activated.

If the result of a comparison is **different**, flag 1 is not activated.

Flag 2 (negative, smaller)

If the result of an operation is smaller than zero, flag 2 is activated.

If the result of an operation is greater than or equal to zero, flag 2 is not activated.

If, in a comparison, the first operand is smaller than the second, flag 2 is activated.

If, in a comparison, the first operand is greater than or equal to the second flag 2 is not activated.

The conditions for the program to jump to the targeted block, after reading G26,G27,G28 or G29 are:

With G27 the program will jump if flag 1 is activated.
With G28 the program will jump if flag 1 is not activated.
With G29 the program will jump if flag 2 is activated.
With G30 the program will jump if flag 2 is not activated.

G26 Conditional jump/call if = 0

When the CNC reads a block with the code G26, if the condition = 0 is met, it jumps to the block indicated by N4 or N4.4.2; if the condition = 0 is not met, the CNC will disregard this block.

Programming

N4 G26 N4 or N4 G26 N4.4.2

G26 must stand alone in a block.

Examples:

```
a) N0  G00  X10
   N5  P2 = K3
   N10 P1 = P2 F1 K5
   N15 G01  Z5
   N20 G26  N50
   N25
    "
    "
    "
   N50 G1  Z10
```

The last operation with parameters being P1 = P2+K5=3+5=8 (result = 0), the = 0 flag will not be activated and the CNC will disregard block N20.

```
b) N0  G00  X10
   N5  P2 = K3
   N10 P1 = P2 F1 K5
   N15 G01  Z5
   N20 P3 = K7
   N25 P4 = P3 F2 K7
   N30 G26  N50
    "
    "
    "
   N50 M30
```

The last operation with parameters being
P4 = P3+K2 K7= 7-7=0, the =0 flag will be activated and the CNC will jump to block 50 when reading block 30.

G27 Conditional jump/call if not equal to 0

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

Programming

N4 G27 N4 or N4 G27 N4.4.2

G27 must stand alone in a block.

G28 Conditional jump/call if smaller than

When the CNC reads a block with the code G28, if the condition < is met, it jumps to the block identified by N4 or N4.4.2. If the condition < is not met, the CNC will disregard this block.

Programming

N4 G28 N4 or N4 G28 N4.4.2

G28 must stand alone in a block.

Example: A program to define a parabolic path whose formula is:
$Z = -K X^2$

X in diameters

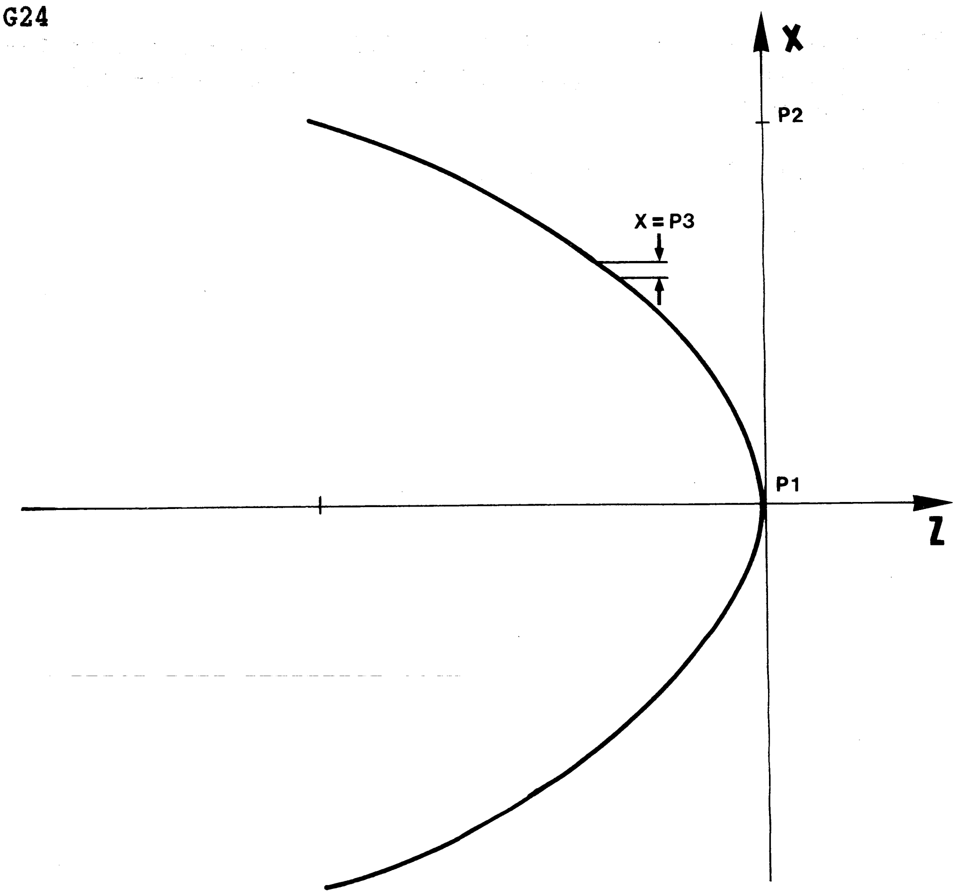
The calling parameters are:

```
P0 --> K
P1 --> Starting X coordinate values
P2 --> Final X coordinate values
P3 --> X axis increment
```

Calculated parameters

```
P4 --> Coordinate X
P5 --> Coordinate Z
```

```
N80 G21 N56.1 P0=K0.01 P1=K00 P2=K100 P3=K1
N90 M30

N110 G23  N56
N120 P4=P1 --> X=X initial
N130 P4=P4 F1 P3 P4=F11 P2
N140 G28  N160
N150 P4=P2
N160 P5=P4 F3 P4 P5=P5 F3 P0 P5=F16 P5
N170 G01 XP4 ZP5 --> Movement block
N180 P4=F11 P2
N190 G27 N130
N200 G24
```



G29 Conditional jump/call if equal or greater than

When the CNC reads a block with G29, if the condition "equal or greater than" is met, it jumps to the block identified by N4 or N4.4.2. If this condition is not met, the CNC will disregard this block.

Programming:

N4 G29 N4 or N4 G29 N4.4.2

G29 must stand alone in a block.

G30 Display of error code defined by K

When the CNC reads a block with G30, it stops the program and displays the contents of this block.

Programming:

N4 G30 K2

N4 : Block number
G30 : Code identifying programming of an error
K2(0-99) : Programmed error code

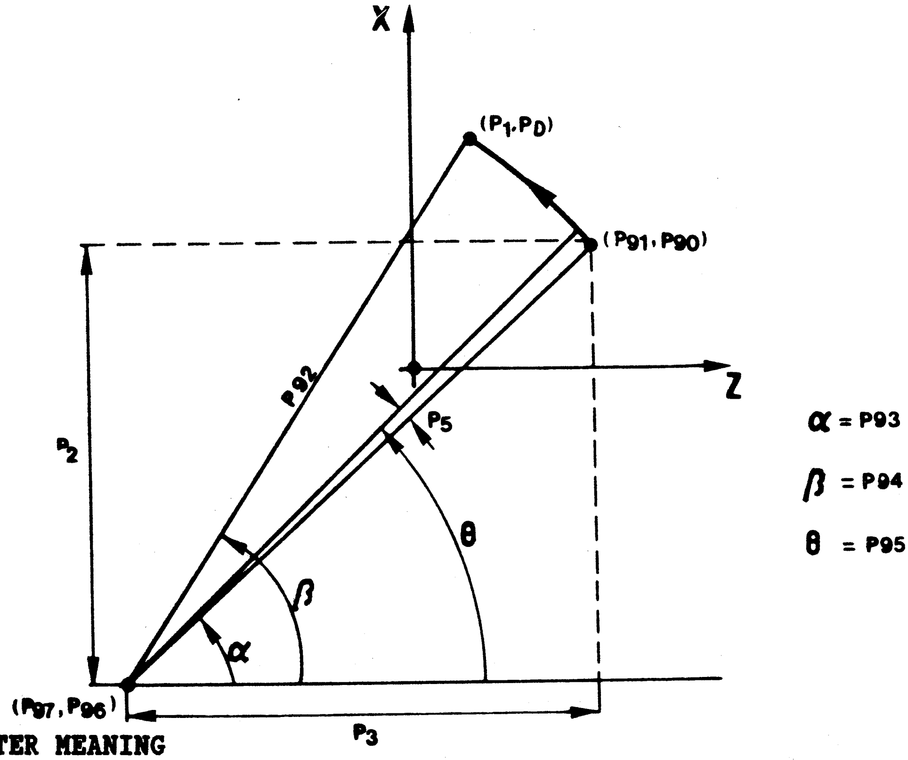
However, if the error code K is programmed by means of a parameter, it can have a value from 0 to 255.

This code, combined with G26,G27,G28,G29 enables, for instance, the stopping of the program and the detection of possible measuring error.

G30 must stand alone in a block.

PROGRAMMING EXAMPLE OF AN ARC WHOSE RADIUS IS LARGER THAN 8388.607 mm

If X axis is in radius and the starting point is at X2000 Z3000, when programming the following arc: G03 X3774.964 Z1000 I-7000 K-8000 the CNC will issue error code 33 which indicates that a movement of more than 8388 mm has been programmed. Thus, to execute the arc, we are forced to use parametric programming.



PARAMETER MEANING

Calling parameters

P0 : X value of the destination point (radius or diameter).
P1 : Z value of the destination point.
P2 : Distance from the starting point to the center on X (in radius).
P3 : Distance from the starting point to the center on Z.
P4 : Feedrate.
P5 : Angular increment value in degrees with a (+) sign for counter-clockwise and (-) sign for clockwise.

Parameters used in the subroutine

P90 : X value of the starting point (in radius).
P91 : Z value of the starting point.
P92 : Radius.
P93 : Initial angle α.
P94 : Final angle β.
P95 : Working or moving angle θ.
P96 : X value of the arc's center (in radius).
P97 : Z value of the arc's center.
P98 : Calculations.
P99 : Calculations.

Flux diagram

```
                          -----
                         | N98 |
                          -----
                            |
              --------------------------------
             | Assign P90 and P91 coordinate |
             | values of the starting point  |
              --------------------------------
                            |
                  -------------------------
                 |Calculate arc's center   |
                 |    values P96 and P97   |
                  -------------------------
                            |
                  -------------------------
                 | Calculate arc's radius  |
                  -------------------------
                            |
               -----------------------------
              | Calculate  initial angle  α |
               -----------------------------
                            |
               -----------------------------
              |   Calculate final angle  β  |
               -----------------------------
                            |
 ---------------------------------------------------
|Adjust α  β if the arc is in the 3rd  4th quadrant |
 ---------------------------------------------------
                            |
                    -----------------
                   |Angle  θ = α + P5|
                    -----------------
 ------------>-------------|
|            --------------------------------------
|           | Calculate point (X,Z) for new angle  |
|            --------------------------------------
|                            |
|              ------------------------------
|             |      Travel to that point    |
|              ------------------------------
|                           |
|                      -------------  Yes      -----
|                     | Arc's end?  |----------| END |
|                      -------------           -----
|                            |No
|                   Yes  ---------  No
|          -------------|¿α > β?  |---------------
|         |              ---------                |
|     -----------                          -----------
|    |θ = θ + P5 |                        |θ = θ + P5 |
|     -----------                          -----------
|         |                                     |
|     ---------  No                        ---------  No
|    |¿θ < β?  |---                       |¿θ > β?  |---
|     ---------    |                       ---------    |
|         |Yes     |                           |Yes     |
|       -----      |                         -----      |
|      |θ = β|     |                        |θ = β|     |
|       -----      |                         -----      |
|         |---<----                             |---<-----
 --------------------------------------------------
```

SUBROUTINE N98

```
N01 P99=R P90=X P90=P90 F4 P99 P91=Z          Takes point's values
    P96=P90 F1 P2  P97=P91 F1 P3              Calculates center
    P92=P2 F6 P3                              Calculates radius
    P98=P2 F4 P3  P93=F10 P98                |Calculates angle α
    P98=P91 F2 P97  P98=F11 K0               |
N02 G29 N4
N03 P93=P93 F1 K180
N04 P98=R P98=P0 F4 P98 P98=P98 F2 P96 P99=P1 F2 P97 |Calculates angle β
N05 P94=P98 F4 P99 P94=F10 P94 P99=F11 K0            |
N06 G29 N8
N07 P94=P94 F1 K180
N08 P5=F11 K0                                |
N09 G29 N16                                  |
N10 P93=F11 K0                               |
N11 G29 N21                                  |Adjusts values of
N12 P94=F11 K0                               |α  β if the
N13 G28 N21                                  |arc goes from the 3rd
N14 P93=P93 F1 K360                          |to the 4th quadrant
N15 G25 N21                                  |from 4th to 3rd
N16 P94=F11 K0                               |
N17 G29 N21                                  |
N18 P93=F11 K0                               |
N19 G28 N21                                  |
N20 P94=P94 F1 K360                          |
N21 P95=P93 F1 P5                             Angle θ = α + P5
N22 P98=F7 P95 P98=P98 F3 P92 P98=P98 F1 P96  point's X value
    P99=R P98=P98 F3 P99
    P99=F8 P95 P99=P99 F3 P92 P99=P99 F1 P97  Point's Z value
N23 G1 XP98 ZP99 FP4                          Travel to point
N24 P95=F11 P94                               End of arc?
N25 G26 N37
N26 P94=F11 P93                               Compares α  β
N27 G26 N37                                   If α = β end
N28 G28 N33
N29 P95=P95 F1 P5  P95=F11 P94                If β>α increase θ
                                                 look if = to β

N30 G28 N32
N31 P95=P94                                   If reached or passed θ=β
N32 G25 N22                                   Calculate new point
N33 P95=P95 F1 P5  P94=F11 P95                If α>β decrease θ
                                                 look if = to β)

N34 G28 N36
N35 P95=P94                                   If reached or passed θ=β
N36 G25 N22                                   Calculate new point
N37 G24
```

With this subroutine any arcs of radius greater than 8388.607 mm can be made both clockwise and counter-clockwise.
The program to make the arc defined before is the following:

Programming the X axis in radius

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

Programming the X axis in diameters

N10 P0=7549.928 P1=K1000 P2=K-7000 P3=K-8000 P4=K100 P5=K0.5
N20 G1 G41 X4000 Z3000 T1.1
N30 G21 N98.01

NOTE: If tool compensation is used, the following programming sequence must be observed:

1st. Recall-parameter definition.

2nd. Positioning at the arc's initial point.

3rd. Calling the Subroutine.

14. CANNED CYCLES

The CNC 8020 T includes the following canned cycles:

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

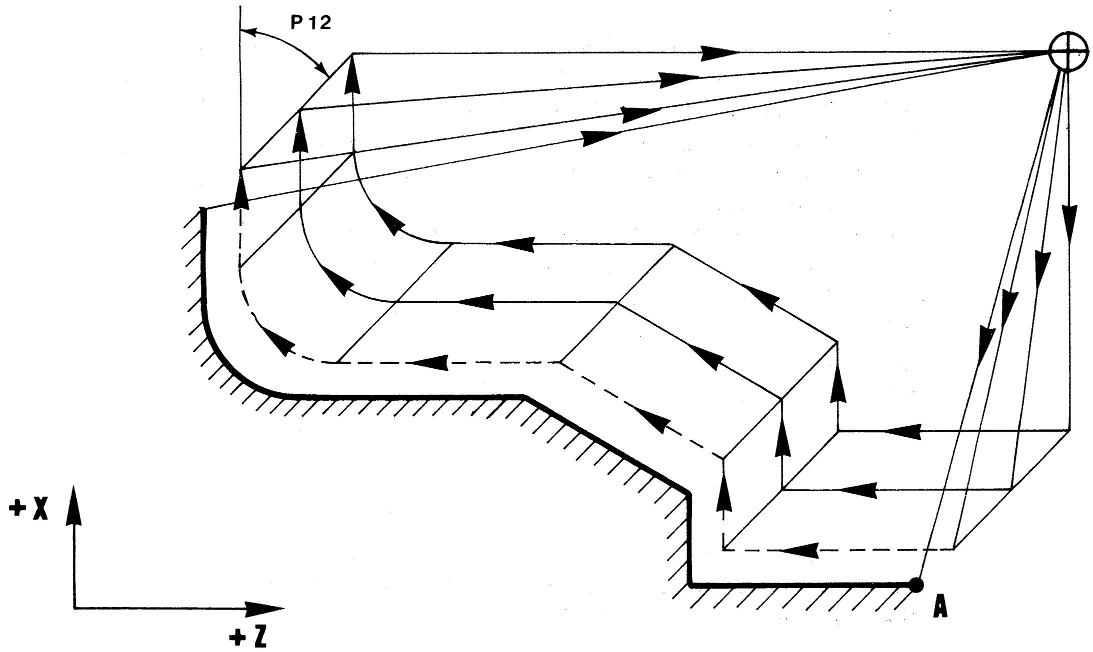
NOTE:

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

If the value of a parameter is a constant, when programming canned cycles, it is necessary to key-in K after =.

Example: N4 G66 P0 = K25 ......

14.1. G66. Pattern repeating



Format:

N4 G66 P0=K P1=K P4=K P5=K P7=K P8=K P9=K P12=K P13=K P14=K

Meaning of the parameters:

P0 : X coodinate value of the initial point A, in radius or diameter.

P1 : Z coordinate value of the initial point A.

P4 : Residual stock. It must be 0 and than the finishing stock allowance or error 3 will be displayed. According to P12 it will be identified as residual in X or Z.

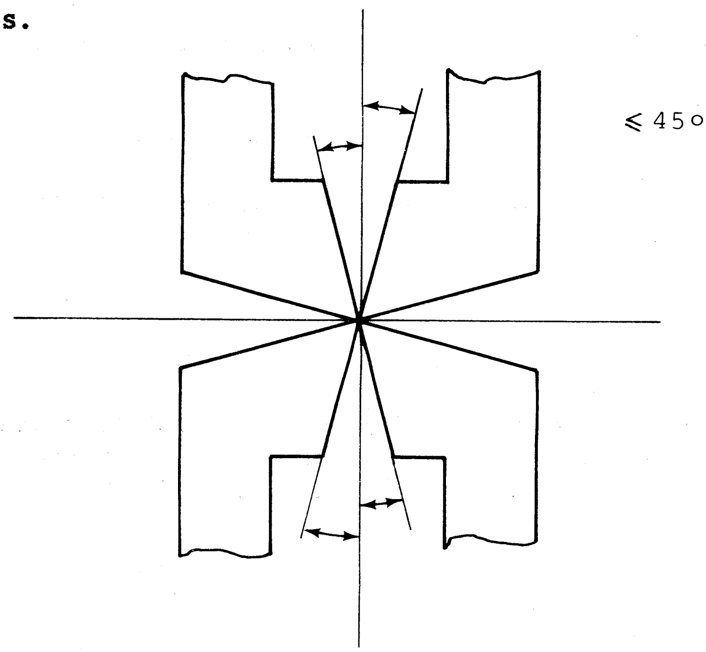
P5 : Max. step. It must be greater than zero or error code 3 will be displayed. According to P12 it will be identified as step along X or Z axis. The real step calculated by the CNC will be equal or smaller than the max. step.

P7 : Finishing stock allowance on the X axis. It must be 0, or error 3 will be displayed.

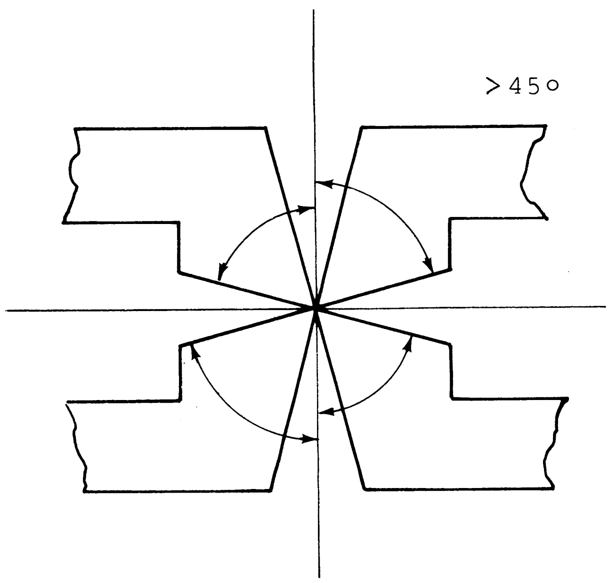
P8 : Finishing stock allowance on the Z axis. It must be 0, or error 3 will be displayed.

P9 : Feedrate for the finishing pass. If it is = 0, there is no finishing pass. If it is negative, error code will be displayed.

P12 : Tool angle. Its value must be comprised between 0 degrees and 90 degrees or error 3 will be displayed. If it is equal or smaller than 45 degree P4 will be taken as residual stock on the X axis and P5 as max. step along X axis.



If it is greater than 45 degree, P4 will be taken as residual stock on the Z axis and P5 as max. step along Z axis.



P13 : Number of the first block to define the pattern.

P14 : Number of the last block to define the pattern.

The definition of the pattern must not include point A because it is identified by P0 and P1.

The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle. The parameters must be entered either in the cycle calling block or in previous blocks. The exit conditions of the cycle are G00 and G90. If the tool's position is not correct to execute the cycle, error code 4 will be displayed. The pattern can be made up by straight lines, circles, roundings, tangential approaches, tangential exits and chamfers. Absolute or incremental programming can be used.

In the definition of the pattern no T function can exist.

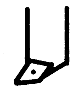
The approaching and withdrawing movements are carried out in rapid and the rest at the programmed feedrate.

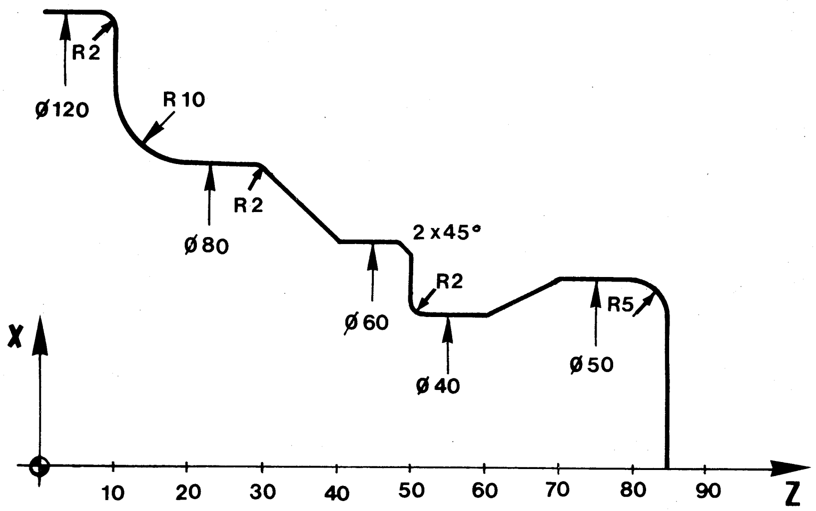
The cycle is completed on the starting position of the tool.

Tool radius compensation (G41,G42) can be used.

The coordinates values (X,Z) of the point from where the cycle is called must be different from P0 and P1 respectively. Otherwise error code 4 will be generated.

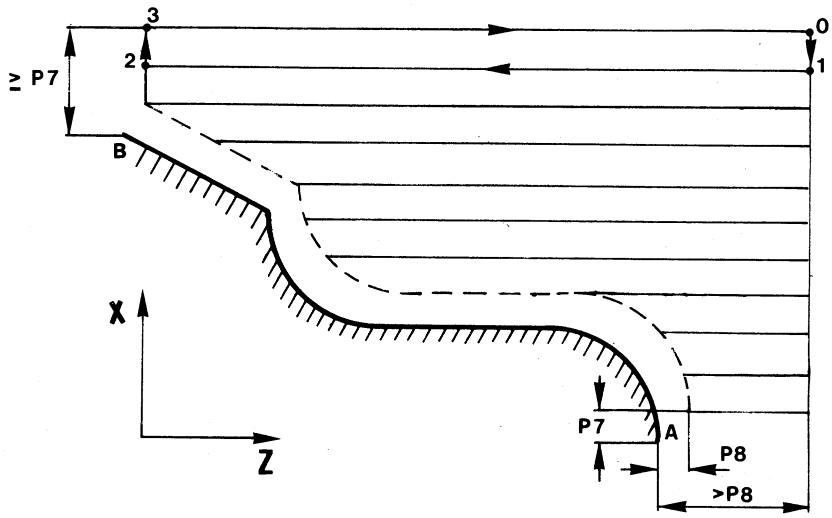
Example G66: X in diameters.






```
N100 --
N110 G90 G00 G42 X150 Z115
N120 G66 P0=K0 P1=K85 P4=K20 P5=K5 P7=K1 P8=K1
         P9=K100 P12=K40 P13=K200 P14=K290
N130 G40 X160 Z135
N140 M30
N200 G36 R5  X50  Z85
N210         X50  Z70
N220         X40  Z60
N230 G36 R2  X40  Z50
N240 G39 R2  X60  Z50
N250         X60  Z40
N260 G36 R2  X80  Z30
N270 G36 R10 X80  Z10
N280 G36 R2  X120 Z10
N290         X120 Z0
```

14.2. G68. Stock removal on the X axis



Format:

N4 G68 P0=K P1=K P5=K P7=K P8=K P9=K P13=K P14=K

Meaning of the parameters:

P0 : Absolute X coordinate value of the starting point (A) in radius or diameters.

P1 : Absolute Z coordinate value of the starting point A.

P5 : Max. step (radius). It must be greater than zero or error code 3 will be displayed. The real step calculated by the CNC will be equal or smaller than the max. step.

P7 : Finishing stock allowance along X axis (radius). It must be greater or equal to zero, or error code 3 will be displayed.

P8 : Finishing stock allowance along Z axis. It must be greater or equal to zero or error code 3 will be displayed.

P9 : Feedrate of the finishing pass. If it is 0, there will be no finishing pass. If it is negative, error code 3 will be displayed.

P13 : Number of the first block to define the pattern.

P14 : Number of the last block to define the pattern. It must be greater than P13. Otherwise error code 13 is generated.

NOTES:

1 - The distance between the starting point 0 and final point (B) along, the X axis must be equal or greater than P7. To avoid passes that are too thin or generating error 31 when operating with tool compensation the value of this distance (from 0 to B) should be equal to P7+NP5, N being an entire number.

2 - The distance from 0 to A along Z axis should be higher than P8.

3 - The definition of the pattern must not include point A because it is identified by P0 and P1.

4 - The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle. The parameters can be programmed in the calling block or in previous blocks. The exit conditions are G00 and G90.

5 - If the tool's position is not correct to start the cycle, error code 4 will be displayed.

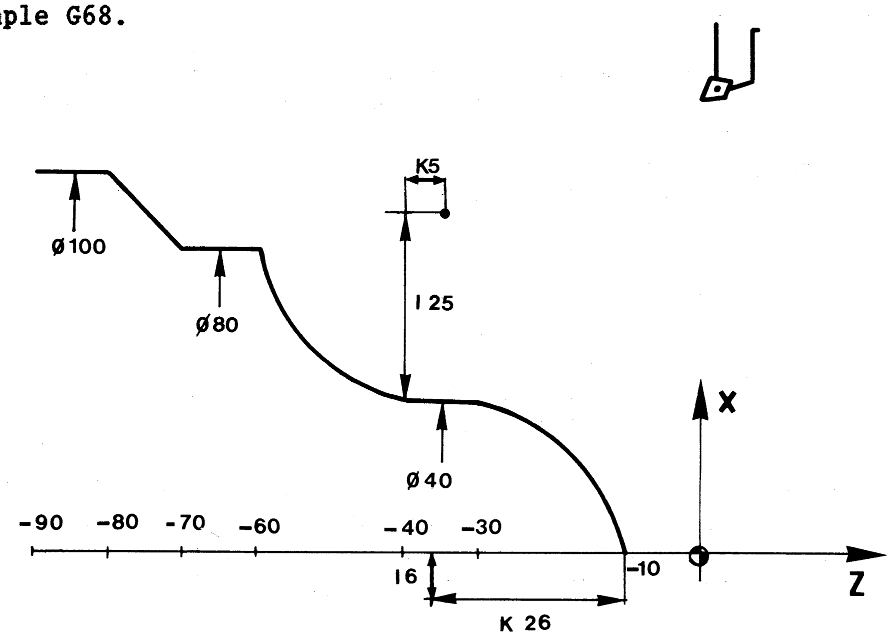
6 - The pattern can be made up of straight lines and arcs. **All** the blocks of pattern definition will be programmed with cartesian coordinates being **mandatory** to program the **two axes** in **absolute**, otherwise, the CRT will display error 21. If arcs are included in the definition, they must be programmed with the center's I,K coordinates, referred to the arc's starting point and with the relevant sign. If functions F,S,T or M are programmed in the definition, they will be ignored except for the finishing pass.

7 - The cycle is completed on the starting position of the tool 0.

8 - If the last movement prior to calling the canned cycle (G68) has been executed in G00, tool radius compensation (G41,G42) can be used. Otherwise error 35 will be displayed.

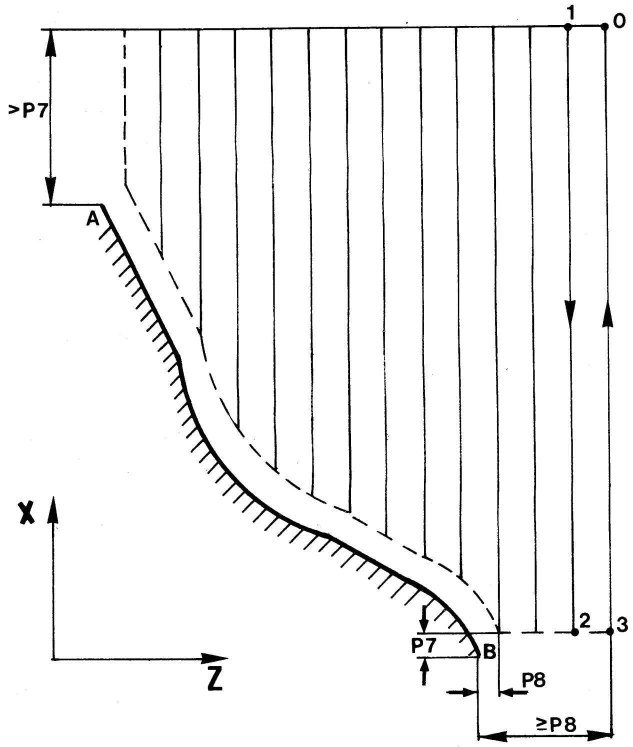
The figure shows the elemental cycle. The movements 1->2 and 2->3 will be performed at the programmed feedrate and the 0->1 and 3->0 in rapid.

Example G68.




```
N100 --
N110 G42 G00 X120 Z0
N120 G68 P0=K0 P1=K-10 P5=K2 P7=K0.8 P8=K0.8 P9=K100
         P13=K200 P14=K250
N130 G40 X130 Z10
N140 M30
N200 G03 X40  Z-30  I-6  K-26
N210 G01 X40  Z-40
N220 G02 X80  Z-60  I25  K5
N230 G01 X80  Z-70
N240     X100 Z-80
N250     X100 Z-90
```

14.3 G69. Stock removal along the Z axis



Format:

N4 G69 P0=K P1=K P5=K P7=K P8=K P9=K P13=K P14=K

Meaning of the parameters:

P0 : Coordinate X value of the starting point (A) in radius or diameters.

P1 : Coordinate Z value of the starting point (A).

P5 : Max. step. It must be greater than zero or error code 3 will be displayed. The real step calculated by the CNC will be smaller or equal to the max. step.

P7 : Finishing stock allowance along X axis. It must be greater or equal to zero or error code 3 will be displayed.

P8 : Finishing stock allonwance along Z axis. It must be greater or equal to zero or error code 3 will be displayed.

P9 : Feedrate of the finishing pass. If it is zero there will be no finishing pass. If it negative, error code 3 will be displayed.

P13 : Number of the first block to define the pattern.

P14 : Number of the last block to define the pattern. It must be higher than P13 or error code 13 will be displayed.

NOTES:

1 - The distance between the starting point 0 and B point along the Z axis must be equal or greater than P8. To avoid passes that are too thin or generating error P31 when operating with tool compensation the value of this distance (from 0 to B) should be equal to P8+NP5, N being an entire number.

2 - The distance from 0 to A along. The axis should be higher than P7.

3 - The definition of the pattern must not include point A because it is identified by P0 and P1.

4 - The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle. The parameters can be programmed in the calling block or in previous blocks. The exit conditions are G00 and G90.

5 - If the tool's position is not correct to start the cycle, error code 4 will be displayed.

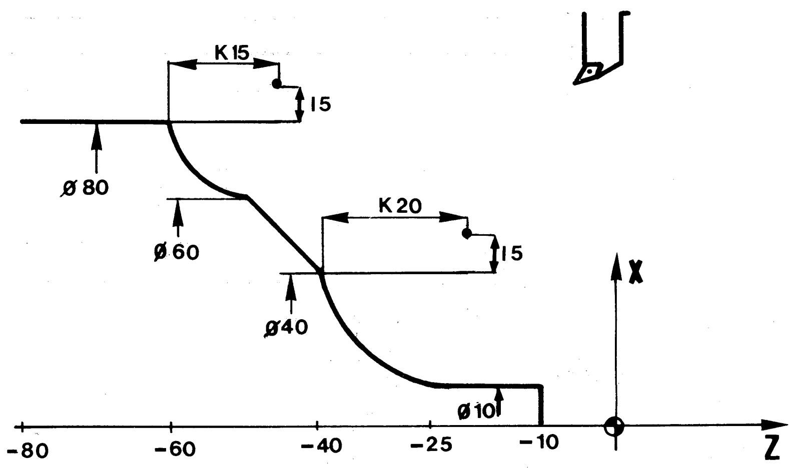
6 - The pattern can be made up of straight lines and arcs. **All** the blocks of pattern definition will be programmed with cartesian coordinates being **mandatory** to program the **two axes in absolute**, otherwise, the CRT will display error 21. If arcs are included in the definition, they must be programmed with the center's I,K coordinates, referred to the arc's starting point and with the relevant sign. If functions F,S,T or M are programmed in the definition, they will be ignored except for the finishing pass.

7 - The cycle is completed on the starting position of the tool (0).

8 - If the last movement prior to calling the canned cycle (G69) has been executed in G00, tool radius compensation (G41,G42) can be used. Otherwise error 35 will be displayed.

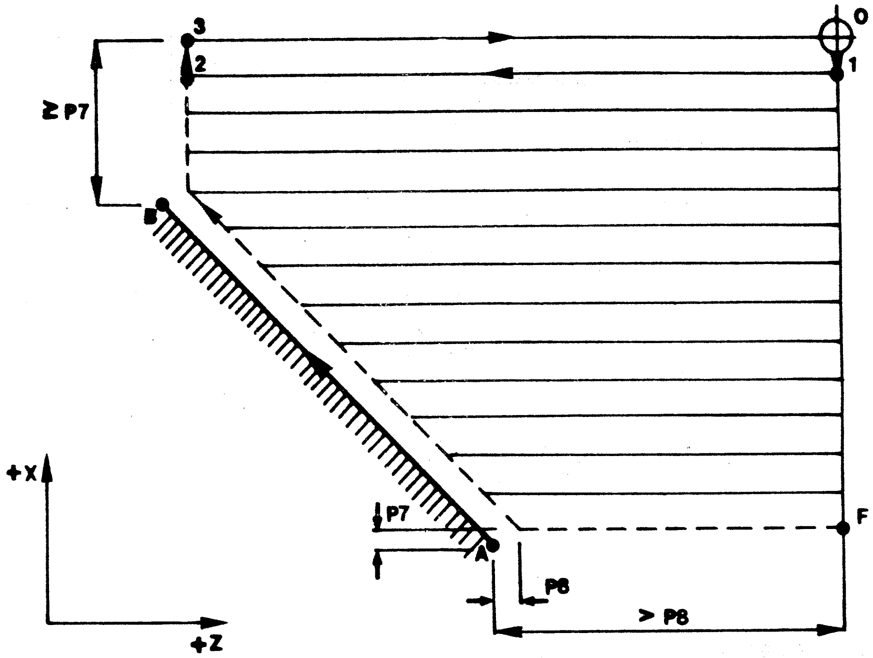
The figure shows the elemental cycle. The movements 1->2 and 2->3 will be performed at the programmed feedrate and the 0->1 and 3->0 in rapid.

Example G69.




```
N190 --
N200 G41 G0 X90 Z-5
N210 G69 P0=K80 P1=K-80 P5=K2 P7=K0.8 P8=K0.8
         P9=K100 P13=K300 P14=K340
N220 G40 X100 Z0
N230 M30
N300 G01 X80 Z-60
N310 G03 X60 Z-50 I5 K15
N320 G01 X40 Z-40
N330 G03 X10 Z-25 I5 K20
N340 G01 X10 Z-10
```

14.4. G81. Canned turning cycle of straight sections



EXAMPLE: Let us suppose the coordinate values of the drawing's points are: A(X0 Z0) B(X90 Z-45) O(X134 Z47) and the programming of X axis is in diameters.

N90 G00 X134 Z47 --> The tool located in point 0.
N100 G81 P0=K0 P1=K0 P2=K90 P3=K-45 P5=K5 P7=K3 P8=K4 P9=K100

Meaning of the parameters:

P0 : X coordinate value of the point A (radius or diameters).

P1 : Z coordinate value of the point A.

P2 : X coordinate value of point B (radius or diameters).

P3 : Z coordinate value of point B.

P5 : Max. step. It must be greater than zero or error code 3 will be displayed. The real value calculated by the CNC will be smaller or equal to the max. step.

P7 : Finishing stock allowance on the X axis. It must be greater or equal to zero or error code 3 will be displayed.

P8 : Finishing stock allowance along the Z axis. It must be greater or equal to zero or error code 3 will be displayed.

P9 : Feedrate of the finishing pass. If it is zero, there will be no finishing pass. If it is negative, error code 3 will be displayed.

NOTES:

1 - The distance between the starting point 0 and final point (B) along the X axis must be equal or greater than P7. To avoid passes that are too thin or generating error 31 when operting with tool compensation the value of this distance (from 0 to B) should be equal to P7+NP5, N being an entire number.

2 - The distance from 0 to A along Z axis should be higher than P8.

3 - The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle. The parameters can be programmed in the calling block or in previous blocks. The exit conditions of the cycle are G00 and G90.

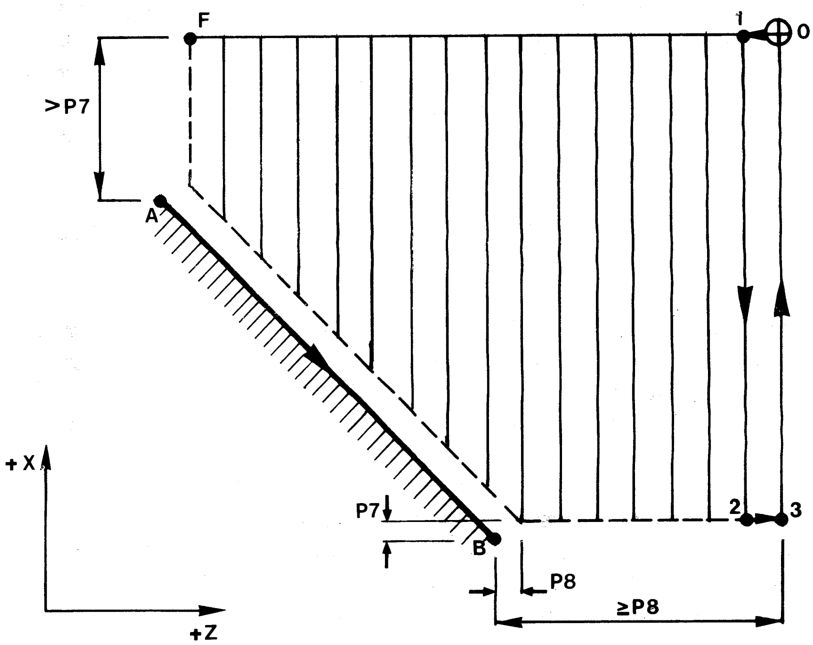
4 - If the position of the tool is not correct to execute the cycle, error code 4 will be displayed. If it is correct but the X value of starting point does not coincide with the X value of point B, an horizontal turning will be carried out until it is reached and then the cycle will be executed.

5 - If there is a finishing pass the cycle will be completed on the starting position of the tool (0). Otherwise the cycle will end on point F.

6 - If the last movement prior to calling the canned cycle (G81) has been executed in G00, tool radius compensation (G41,G42) can be used. Otherwise error 35 will be displayed.

The figure shows the elemental cycle. The movements 1-> 2 and 2->3 will be performed at the programmed feedrate and the 0->1 and 3->0 in rapid.

14.5. G82. Canned facing cycle of straight sections



EXAMPLE: Let us suppose the coordinate values of the drawing's points are: A(X90 Z-45) B(X0 Z0) 0(X136 Z39) and the programming of X axis is in diameters.

N90 G00 X136 Z39 --> The tool located in point 0.
N100 G82 P0=K90 P1=K-45 P2=K0 P3=K0 P5=K5 P7=K3 P8=K4 P9=K100

Meaning of the parameters:

P0 : X coordinate value of point A (radius or diameters).

P1 : Z coordinate value of point A.

P2 : X coordinate value of point B (radius or diameters).

P3 : Z coordinate value of point B.

P5 : Max. step. It must be greater than zero or error code 3 will be one displayed. The real step calculated by the CNC will be smaller or equal to the max. step.

P7 : Finishing stock allowance on the X axis. It must be greater or equal to zero or error code 3 will be displayed.

P8 : Finishing stock allowance along the Z axis. It must be greater or equal to zero or error code 3 will be displayed.

P9 : Feedrate of the finishing pass. If it is zero there will be no finishing pass. If it is negative, error code 3 will be displayed.

NOTES:

1 - The distance between the starting point 0 and final point (B) along the Z axis must be equal or greater than P8. To avoid passes that are too thin or generating error 31 when operting with tool compensation the value of this distance (from 0 to B) should be equal to P8+NP5, N being an entire number.

2 - The distance from 0 to A along the X axis should be higher than P7.

3 - The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle. The parameters can be programmed in the calling block or in previous blocks. The exit conditions of the cycle are G00 and G90.

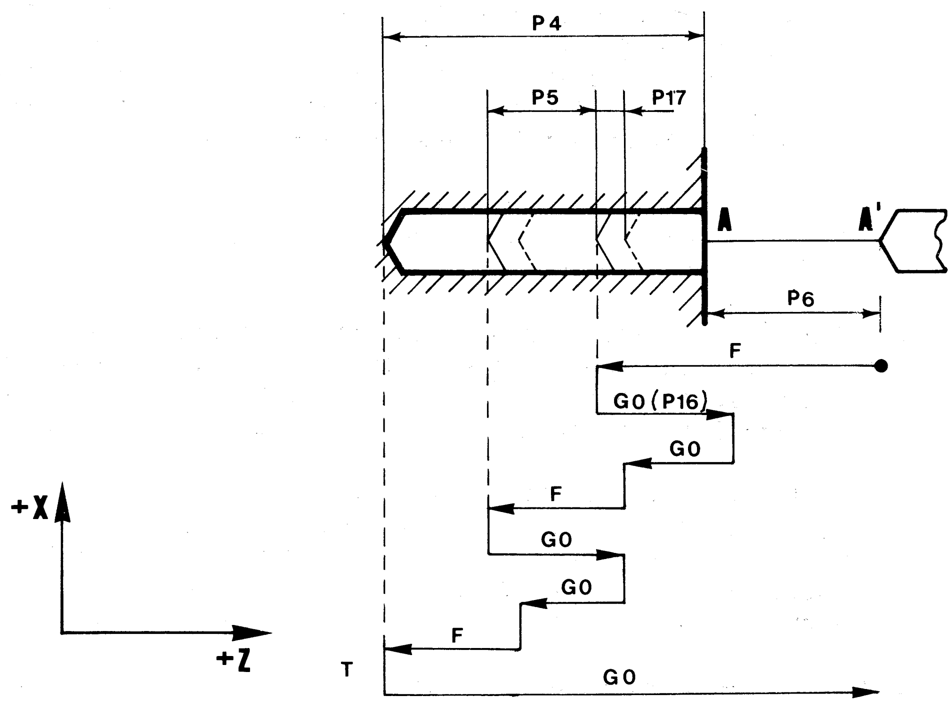
4 - If the position of the tool is not correct to execute the cycle, error code 4 will be displayed. If it is correct but its Z value of starting point does not coincide with the Z value of point B, a vertical facing will be carried out until it is reached and then the cycle will be executed.

5 - If there is a finishing pass, the cycle will be completed on the tool's starting position (0). If there is no finishing pass the cycle will end at point F.

6 - If the last movement prior to calling the canned cycle (G82) has been executed in G00, tool radius compensation (G41,G42) can be used. Otherwise error 35 will be displayed.

The figure shows the elemental cycle. The movements 1->2 and 2->3 will be performed at the programmed feedrate and the 0->1 and 3->0 in rapid.

14.6. G83. Deep hole drilling cycle



Format:

N4 G83 P0=K P1=K P4=K P5=K P6=K P15=K P16=K P17=K

Meaning of the parameters:

P0 : Absolute X value of the point where the drilling or circular groove is desired (if different from zero) in radius or diameters.

P1 : Absolute Z value of the point where the drilling is desired.

P4 : Total depth of the hole. It will have positive value when drilled towards the negative direction of the Z axis and viceversa. If it is zero, error code 3 will be displayed.

P5 : Max. pass. The CNC will execute the minimum number of equal passes, smaller than P5 until the total depth, defined by P4, is reached. If it is equal to or smaller than zero, error code 3 will be displayed.

P6 : Safety distance. It defines the distance to the part from the point where the tool ends the positioning approach. If it is negative, error code 3 will be generated.

P15 : Dwell. It identifies the value in seconds of the dwell at the bottom of the hole. If it is negative, error code 3 will be displayed.

P16 : It indicates the incremental value of the G00 movement after each pass. If it is zero, this movement will be executed up to the A'point. If it is negative, error 3 will be displayed.

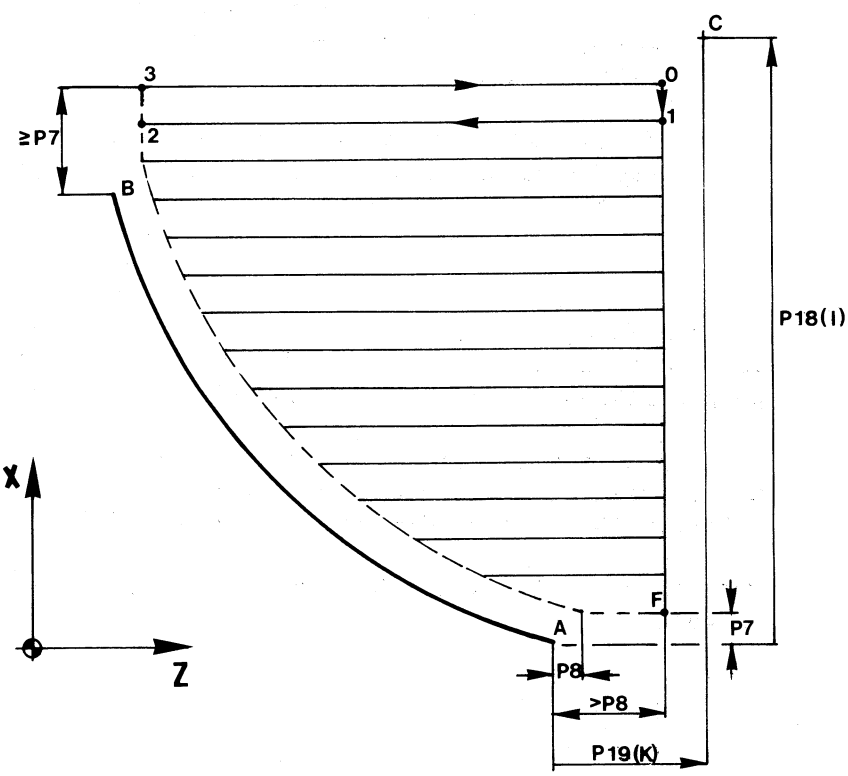
P17 : It indicates the safety distance between the bottom of the previous penetration and the point where the tool ends the rapid approach for a subsequent penetration. If it is negative, error code 3 will be displayed.

The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle. The parameters can be programmed in the calling block or in previous blocks.

The cycle does not alter the calling parameters and thus, they can be used for future cycles. The parameters P90 to P96 are altered. The exit conditions are G00,G07,G40 and G90.

The cycle starts with a G00 approach to point A' and ends at A' as well.

14.7. G84. Turning cycle with arcs



EXAMPLE: Let us suppose the coordinate values of the drawing's point are: 0(X149 Z86) A(X0 Z71) B(X120 Z11) C(X160 Z91) and the programming of X axis is in diameters.

```
N90  G00 X149 Z86 --> The position of the tool
                      is the point 0
N100 G84 P0=K0 P1=K71 P2=K120 P3=K11 P5=K5 P7=K4 P8=K4 P9=K100
         P18=K80 P19=K20
```

Meaning of the parameters:

P0 : X coordinate value of point A (radius or diameters).

P1 : Z coordinate value of point A.

P2 : X coordinate value of point B (radius or diameters).

P3 : Z coordinate value of point B.

P5 : Max. step. It must be greater than zero or error code 3 will be displayed. The real step calculated by the CNC will be smaller or equal to the max. step.

P7 : Finishing stock allowance on the X axis. It must be greater or equal to zero or error code 3 will be displayed.

P8 : Finishing stock allowance along the Z axis. It must be greater or equal to zero or error code 3 will be displayed.

P9 : Feedrate of the finishing pass. If it is zero there will be no finishing pass. If it is negative, error code 3 will be displayed.

P18 : Distance I between the point A and the arc's center along the X axis. Although the X axis is programmed in diameters, the values of I are always programmed in radius.

P19 : K distance between the point A and the arc's center along the Z axis.

When programming this canned cycle, take into account the following aspects:

1. The distance between the starting point 0 and final point (B) along the Z axis must be equal or greater than P8. To avoid passes that are too thin or generating error 31 when operating with tool compensation, the value of this distance (from 0 to B) should be equal to P8+N P5, N being an entire number.

2. The distance between the starting point 0 and the point (A), along the X axis, should be higher than P7.

3. The machining conditions (feedrate, spindle rotation ...) must be programmed before calling the cycle. The parameters can be programmed either in the cycle calling block or in previous blocks. The exit conditions are G00 and G90.

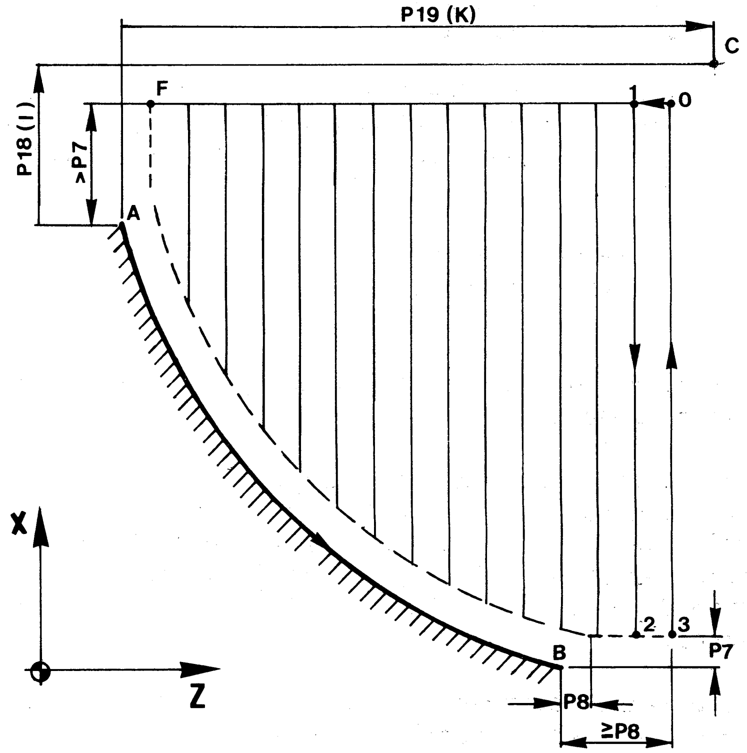
4. If the position of the tool is not correct to execute the cycle, error code 4 will be displayed. If it is correct but its X value of starting point does not coincide with the X value of point B, a horizontal turning will be carried out until it is reached and then the cycle will be executed.

5. If there is a finishing pass, the cycle will be completed on the tool's starting position (0). If there is no finishing pass the cycle will end at point F.

6. If the last movement prior to calling the canned cycle has been executed in G00, tool radius compensation (G41,G42) can be used. Otherwise error 35 will be displayed.

The figure shows the elemental cycle. The movement 1->2 and 2->3 will be performed at the programmed feedrate and the 0->1 and 3->0 in rapid.

14.8. G85. Facing cycle with arcs



EXAMPLE: Let us suppose the coordinate values of the drawing's point are: 0(X150 Z85) A(X118 Z11) B(X0 Z70) C(X160 Z91) and the programming of X axis is in diameters.

```
N90  G00 X150 Z85 --> The position of the tool
                      is the point 0
N100 G85 P0=K118 P1=K11 P2=K0 P3=K70 P5=K5 P7=K4 P8=K4 P9=K100
         P18=K21 P19=K80
```

Meaning of the parameters:

P0 : X coordinate value of point A (radius or diameters).

P1 : Z coordinate value of point A.

P2 : X coordinate value of point B (radius or diameters).

P3 : Z coordinate value of point B.

P5 : Max. step. It must be greater than zero or error code 3 will be displayed. The real step calculated by the CNC will be smaller or equal to the max. step.

P7 : Finishing stock allowance on the X axis. It must be greater or equal to zero or error code 3 will be displayed.

P8 : Finishing stock allowance along the Z axis. It must be greater or equal to zero or error code 3 will be displayed.

P9 : Feedrate of the finishing pass. If it is zero there will be no finishing pass. If it is negative, error code 3 will be displayed.

P18 : Distance I between the point A and the arc's center along the X axis. Although the X axis is programmed in diameters, the values of I are always programmed in radius.

P19 : K distance between the point A and the arc's center along the Z axis.

When programming this canned cycle, take into account the following aspects:

1. The distance between the starting point 0 and final point (B) along the Z axis must be equal or greater than P8. To avoid passes that are too thin or generating error 31 when operating with tool compensation, the value of this distance (from 0 to B) should be equal to P8+N P5, N being an entire number.

2. The distance between the starting point 0 and the point (A), along the X axis, should be higher than P7.

3. The machining conditions (feedrate, spindle rotation ...) must be programmed before calling the cycle. The parameters can be programmed either in the cycle calling block or in previous blocks. The exit conditions are G00 and G90.

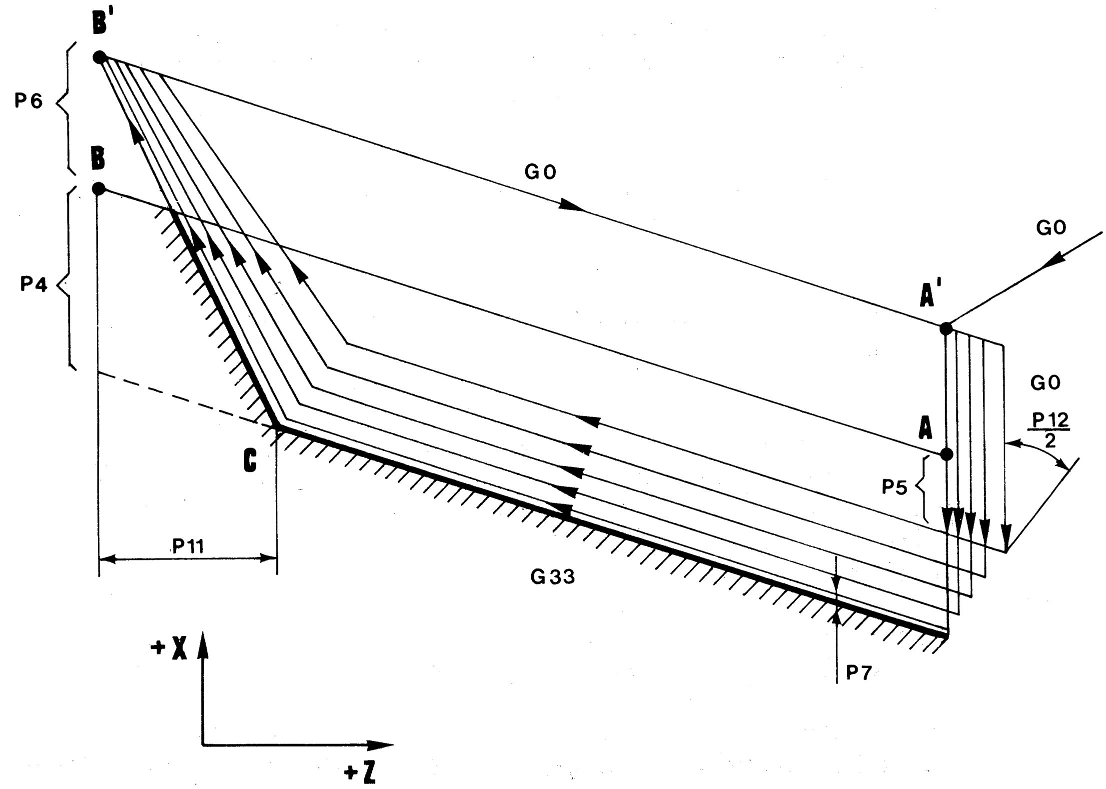
4. If the position of the tool is not correct to execute the cycle, error code 4 will be displayed. If it is correct but its Z value of starting point does not coincide with the Z value of point B, a vertical facing will be carried out until it is reached and then the cycle will be executed.

5. If there is a finishing pass, the cycle will be completed on the tool's starting position (0). If there is no finishing pass the cycle will end a point F.

6. If the last movement prior to calling the canned cycle has been executed in G00, tool radius compensation (G41,G42) can be used. Otherwise error 35 will be displayed.

The figure shows the elemental cycle. The movement 1->2 and 2->3 will be performed at the programmed feedrate and the 0->1 and 3->0 in rapid.

14.9. G86. Threadcutting cycle (Z axis)



Format:

N4 G86 P0=K P1=K P2=K P3=K P4=K P5=K P6=K P7=K P10=K P11=K
P12=K

Meaning of the parameters:

P0 : Absolute X value of the starting point of the thread (A) in radius or diameters.

P1 : Absolute Z value of the starting point of the thread (A).

P2 : Absolute X value of the final point of the thread (B) in radius or diameters.

P3 : Absolute Z value of the final point of the thread.

P4 : Depth of the thread (in radius). If will have a positive value in external threads and a negative one in internal threads. If it is zero error code 3 will be displayed.

P5 : Initial pass (in radius). It defines the depth of the first cutting pass. The subsequent passes will depend on the sign given to the parameter.

- If the sign is **positive**, the depth of the second pass will be $P5\sqrt{2}$ and the depth of the 11th will be $P5\sqrt{n}$, until the finishing depth is reached.

- If the sign is **negative**, the penetration increment will be constant and of a value equal to the absolute value of the parameter.

P6 : Safety distance in radius. It indicates at what distance from the thread the rapid withdrawal to point A' is started. It must be equal to or greater than zero or error code 3 will be displayed.

P7 : Finishing pass (in radius), this pass is carried out with radial approach. If it is zero, the last pass is repeated. If it is negative, error code 3 will be displayed.

P10 : Thread pitch along Z axis.

P11 : Thread exit. It defines the distance from the end of the thread to the point where the exit starts. If it is negative, error code 3 will be displayed. If it is different from zero, the section CB' is a tapered thread whose pitch along Z axis is P10. If it is zero, the section CB' is executed in G00.

P12 : Angle of the tool's nose. It makes the starting points of the successive passes to be at a P12/2 angle with X axis.

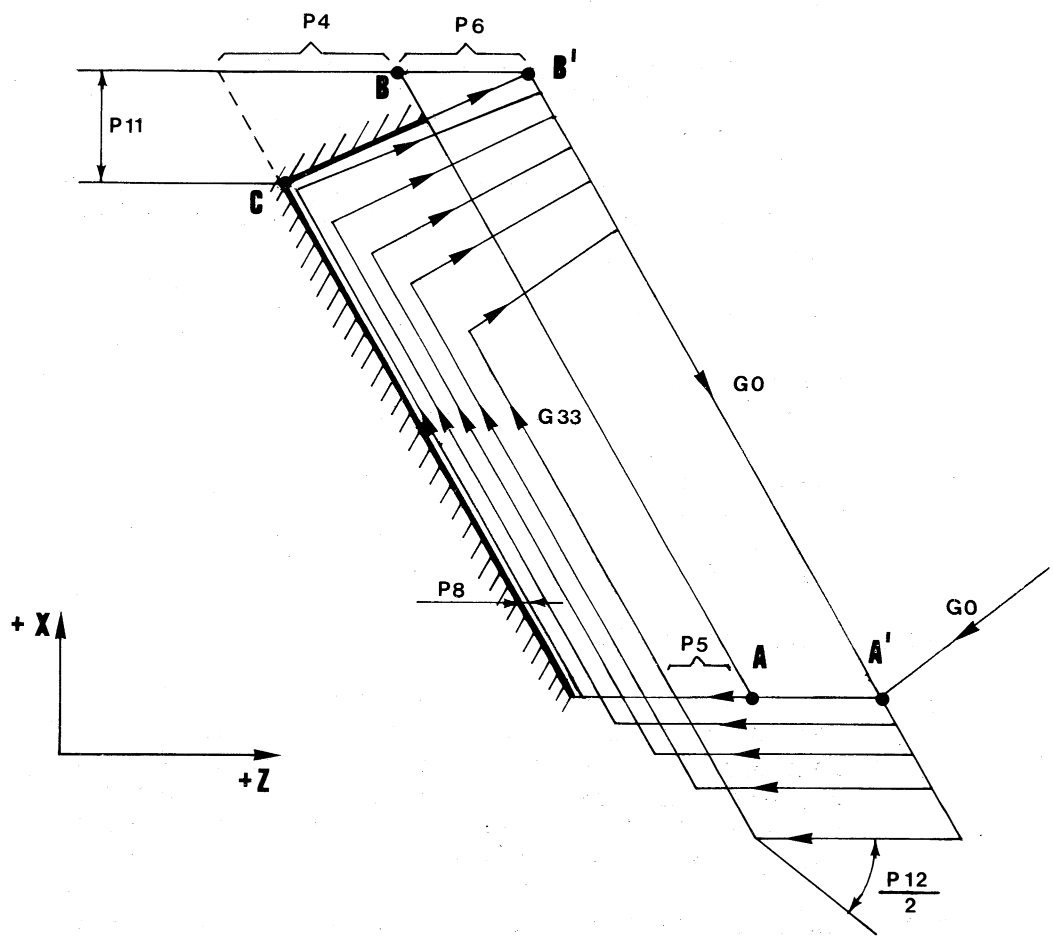
The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle. The parameters can be programmed in the calling block or in previous blocks.

The parameters P80 to P99 are altered. The exit conditions are G00,G07,G40, G90 and G97.

The cycle starts with a G00 approach to point A' and ends at A' as well.

When executing the block, the F feedrate speed cannot be altered by turning the **FEEDRATE** knob whose value will be frozen at 100%.

14.10. G87. Threadcutting cycle (X axis)



Format:

N4 G87 P0=K P1=K P2=K P3=K P4=K P5=K P6=K P8=K P10=K P11=K P12=K

Meaning of the parameters:

P0 : Absolute X coordinate value of the initial point of the thread (A) in radius or diameters.

P1 : Absolute Z coordinate value of the initial point of the thread (A).

P2 : Absolute X coordinate value of the final point of the thread (B) in radius or diameters.

P3 : Absolute Z coordinate value of the final point of the thread (B).

P4 : Depth of the thread. It will have a position value when the cutting takes place towards the negative direction of the Z axis and viceversa. If it is zero, error code 3 will be displayed.

P5 : Initial pass. It defines the depth of the first cutting pass. The subsequent passes will depend on the sign given to the parameter:

- If the sign is **positive**, the depth of the second pass will be P5 $\sqrt{2}$ and the depth of the 11th will be P5 $\sqrt{n}$, until the finishing depth is reached.
- If the sign is **negative**, the deepening increment will be constant and of a value equal to the absolute value of the parameter.
- If the value is equal to zero, error 3 is generated.

P6 : Safety distance. It indicates at what distance from the thread's surface the rapid withdrawal to point A' is started. It must be equal or greater than zero or error code 3 will be displayed.

P8 : Finishing pass, this pass is carried out with radial approach. If it is zero, the last pass is repeated. If it is negative, error code 3 is generated.

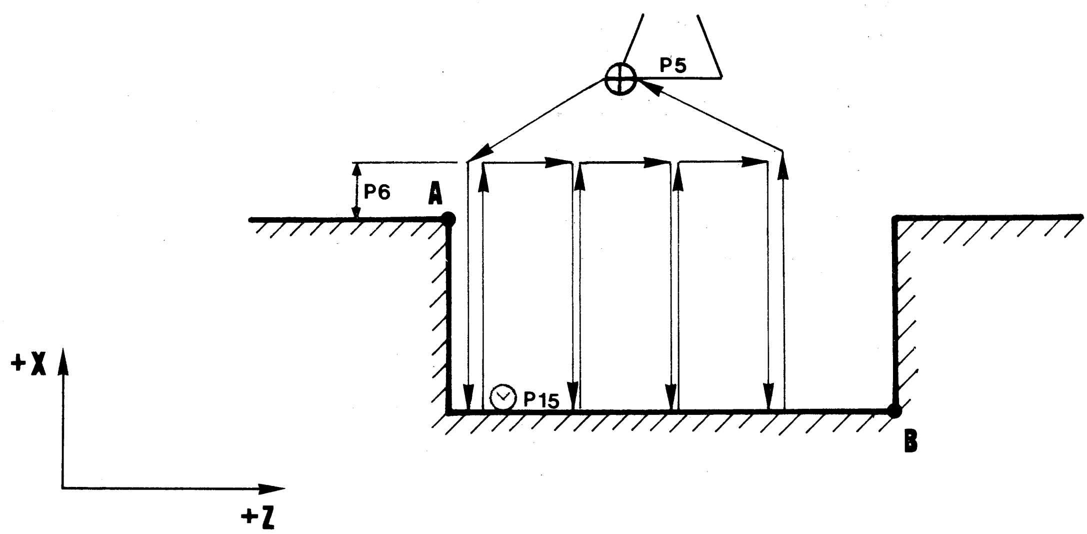
P10 : Thread pitch along X axis in radius.

P11 : Thread exit (in radius). It defines the distance from the end of the thread to the point where the exit starts. If it is negative, error code 3 will be displayed. If it is different from zero, the section CB' is a tapered thread whose pitch along X axis is P10. If it is zero, the section CB' is executed in G00.

P12 : Angle of the tool's nose. It makes the starting points of the successive passes to be at a P12/2 angle with the Z axis.

The machining conditions (feedrate, spindle rotation, etc.) must be programmed before the cycle is called. The parameters can be programmed in the call block or in previous blocks. The cycle does not alter the calling parameters and thus, they can be used for future cycles. The parameters P80 to P99 are altered. The exit conditions are G00, G07, G40, G90 and G97. The cycle starts with a G00 approach to point A' and ends at A' as well. When executing the block, the **F** feedrate speed can't be altered by turning the **FEEDRATE** knob whose value will be frozen at 100%.

14.11. G88. Grooving cycle along X axis



Format:

N4 G88 P0=K P1=K P2=K P3=K P5=K P6=K P15=K

Meaning of the parameters:

P0 : X coordinate value of point A (radius or diameters).

P1 : Z coordinate value of point A.

P2 : X coordinate value of point B (radius or diameters).

P3 : Z coordinate value of point B.

P5 : Tool nose's width. It must be greater than zero or error code 3 will be displayed. The real step calculated by the CNC will be smaller than the tool's width.

P6 : Safety distance. It must be equal or greater than zero or error code 3 will be displayed.

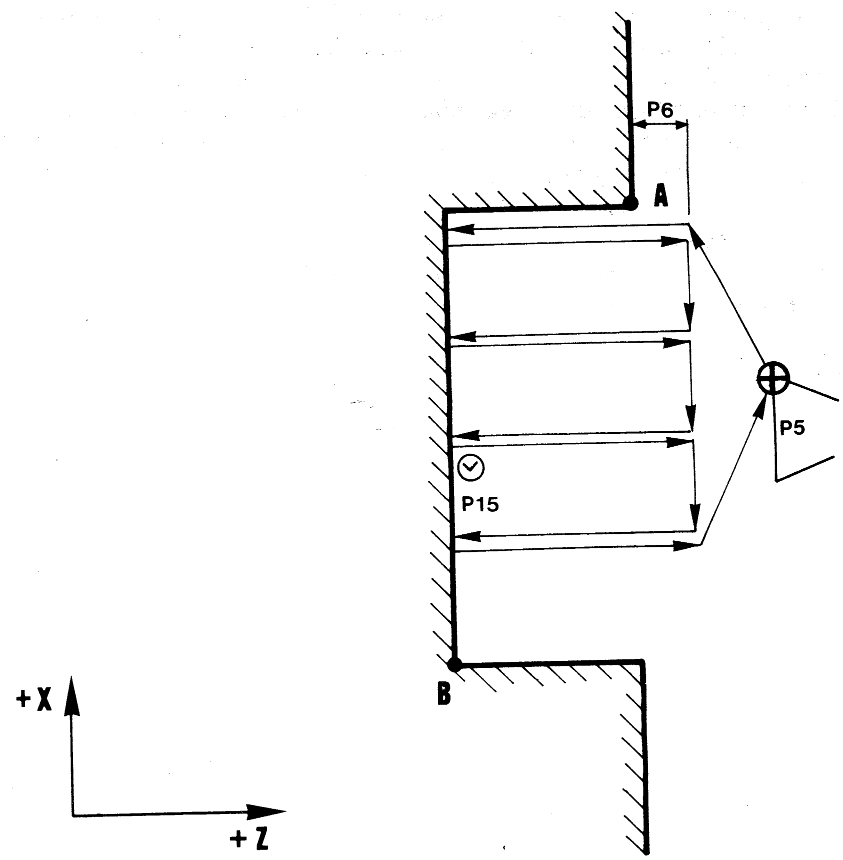
P15 : Dwell at the bottom (seconds). It must be equal or greater than 0 and smaller than 655.36 seconds. Otherwise, error code 3 will be displayed.

The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle. The parameters can be programmed in the calling block or in previous blocks. The exit conditions of the cycle and G00,G07,G40 and G90. If the groove's depth is zero, code 3 will be displayed. If the width of the groove is smaller than the tool nose's width, error code 3 will be displayed. If the position of the tool is not correct to execute the cycle, error code 4 will be displayed.

The movement from the safety distance to the bottom of the groove is carried out at the programmed feedrate. The rest of the movements in rapid.

The cycle ends at the tool's initial position.

14.12. G89. Grooving cycle along Z axis



Format:

N4 G89 P0=K P1=K P2=K P3=K P5=K P6=K P15=K

Meaning of the parameters:

P0 : X coordinate value of point A (radius or diameters).

P1 : Z coordinate value of point A.

P2 : X coordinate value of point B (radius or diameters).

P3 : Z coordinate value of point B.

P5 : Tool nose's width. It must be greater than zero, or error code 3 will be displayed. The real step calculated by the CNC will be smaller than the tool's width.

P6 : Safety distance. It must be equal or greater than zero or error code 3 will be displayed.

P15 : Dwell at the bottom (seconds). It must be equal or greater than zero and smaller than 655.36 seconds. Otherwise error code 3 will be displayed.

The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle. The parameters can be programmed in the call block or in previous blocks. The exit conditions of the cycle and G00,G07,G40 and G90. If the depth of the groove is zero, error code 3 will be displayed. If the width of the groove is smaller than the tool nose's width, error code 3 will be displayed. If the position of the tool is not correct to execute the cycle, error code **4** will be displayed.

The movement from the safety distance to the bottom of the groove is carried out at the programmed feedrate. The rest of the movements in rapid.

The cycle ends at the tool's initial position.

15. ERROR CODES

Code	Meaning
001	N is not the first character of a block. If running the execution of a program, while another program is being **modified** (background programming), a subroutine located in the program being modified or in a subsequent program, is called from the program in execution. The programs are stored as they are entered. Their order can be seen in **Program mapping.** The program being executed is always shifted to the first position. If during the execution of a program, another program is edited with a number not previously stored in the memory, this situation cannot arise.
002	Too many digits to define a function.
003	Negative value (or parameter) assigned to a function which cannot accept minus sign. Incorrect value given to a canned cycle parameter.
004	Calling a canned cycle from a inadequate position.
005	Incorrectly written parametric block.
006	More than 15 parameters affected in a block.
007	Division by zero.
008	Square root of a negative number.
009	Too great a value assigned to a parameter.
010*	The range or the S value, when working on constant surface speed, has not been programmed.
011	More than seven M functions in the same block.
012	. Function G50 improperly programmed. . Tool dimensions too large. . G53/G59 values too large.
013	Canned cycle not properly defined.
014	An incorrect block has been programmed, which is either incorrect by itself or incorrect in relation to the sequence of the program up to that point.
015	Functions G20, G21, G22, G23, G24, G25, G26, G27, G28, G29, G30, G31, G32, G50, G51, G53/G59, G72, G74, G92 or G93 I, K do not stand alone in a block.
016	The subroutine or block called for does not exist or the block looked for with the function **F17** does not exist.
017	Too high or negative threadcutting pitch.

018 Wrong definition when a point is defined by angle plus angle or angle plus cartesian value.

019 After defining G20,G21,G22 or G23 there is no subroutine number to which it refers or too many nesting levels. N is not the first character after G25,G26,G27,G28,G29. Too much mutual internesting.

020 More than one spindle range programmed in a particular block.

021 There is no block in the address defined by the parameter assigned to F18,F19 or F20,F21 or F22.
X axis is not programmed in the block addressed by the parameter assigned to F18.
Z axis is not programmed in the block addressed by the parameter assigned to F19.
I is not programmed in the block addressed by the parameter assigned to F21.
K is not programmed in the block addressed by the parameter assigned to F22.

022 Any one of the axis is repeated when programming on G74.

023 K has not been programmed after G04.

024 Decimal point missing in formats T2.2 or N2.2.

025 Error in a block defining or calling a subroutine or a subprogram, or a jump.

026 Memory capacity overrum. Available tape or CNC memory too small for the program to be stored.

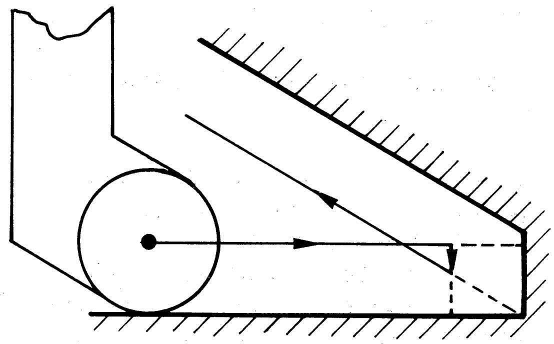
027 I/K not defined in circular interpolation or threadcutting.

028 A tool in the table has been defined as having a number greater than Txx.32 or an external tool greater than the maximum defined by machine parameter.

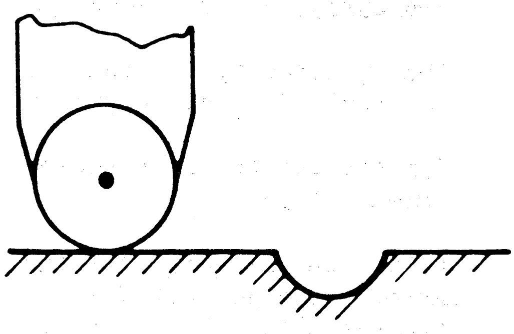
029 Too high a value in a function 4.3 or 3.4. This code is often generated when an F value is first programmed in mm/min. and then switched to mm/rev. without changing the F value.

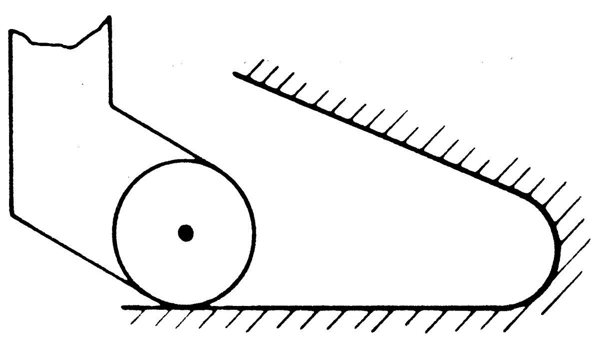
030 A non-existent G has been programmed.

031 Tool radius value too large.



032 Tool radius value too large.





033 A movement greater than 8388 mm or 330.36 inches has been programmed.

Example:

- The X axis is positioned in X - 5 000.

- The block with X 5 000.000 is programmed. The movement is therefore 10 000.000.

- The correct programming would be:

```
G05 X0
    X5 000.000
```

(The programming of G05 prevents the machine from stopping at X=0).

034 S or F has been defined with a greater value than is allowed.

035 There is not enough information for corner rounding, compensation or chamfering.

036 Subroutine repeated.

037 M19 not properly programmed.

038 G72 not properly programmed.

039 More than 15 levels of nesting in calling for subroutines. A jump to itself has been programmed in a block.

040 The circle programmed does not pass through the final point defined (Tolerance 10 microns). The arc defined by G08 or G09 does not exist.

041 Tangential entry. Circle diameter programmed greater than distance between tool starting point and machining commencing point. Also tangential entry with G02 or G03.

042	Tangential exit. Diameter of circle programmed greater than distance between final point and machining exit point. Also tangential exit with G02 or G03.
043	Polar coordinate origin (G93) not properly defined.
044	The function M45 S, live tool's revolution speed, impropery programmed.
045	Error in programming G36,G37,G38 or G39.
046	Polar coordinates not properly defined.
047	A movement zero has been programmed during radius compensation or corner rounding.
048	Beginning or cancellation of radius compensation with G02 or G03.
049	Chamfer not correctly programmed.
050	G96 with a BCD output S in parameter (Ac spindle motor) has been programmed.
051	C axis improperly programmed.
052	Without function.
053	Without function.
054	There is no tape in the cassette reader or the reader head cover is open.
055	Parity error in recording or reading of tape.

056 The CNC will issue this error code:

a) When trying to generate a program using function G76 while the memory is locked.
b) When the program being generated with function G75 P5 is the protected program or program number P99999.
c) When G22 or G23 go after function G76.
d) When more than 70 characters have been written after G76.
e) When G76 (block content) is programmed without G76 P5 or G76 N5 being programmed previously.
f) When in a G76 P5 or G76 N5 type function the 5 digits of the program number are not programmed.
g) When a program number is changed without cancelling the previous one while a program is being generated (G76 P5 or G76 N5).
h) When the programed mentioned in a block G76 P5 exists in memory but is not located in the last position of the memory map.

NOTE: When a program is called for to be edited goes to the last position in the memory map and when it is executed it goes to the first position.

057 Write protected tape.

058 Sluggish tape rotation.

059 Communication error between CNC and tape reader.

060 Circuitry malfunction (Interpolar CPU).

061 Battery malfunction.

064* External emergency activated.

065* The position has been reached without having received the probe's signal. Also, during a probing canned cycle, when the probe's signal is received without the probing movemvent itself being performed (collision).

066* X axis travel limit overrun.

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

067 Without function.

068* Z axis travel limit overrun.

NOTE: Error generated either because machine is beyond limit or because a block has been programmed which would compel machine to go beyond limits.

069 Without function.

070** X axis following error.

071 Synchronized tool's following error.

072** Z axis following error.

073 Without function.

074** S value too high or C axis following error.

075** X axis feedback failure. Connector A1.

076** Synchronized tool feedback failure. Connector A2.

077** Z axis feedback failure. Connector A3.

078** C axis feedback failure. Connector A4.

079** Spindle feedback failure.

080 Without function.

081	Without function.
082	Without function.
083	Withour function.
084	Without function.
085	Without function.
086	Without function.
087**	Error in RAM CMOS memory of interpolating microprocessor.
088**	Error in EPROM of interpolating microprocessor.
089*	The search of the machine-reference point has not been carried out in all the axes. The machine-parameter P98(8) defines its obligatory nature.
090**	Error in PPI 1 (U15).
091**	Error in PPI 2 (U17).
092**	Error in PPI 3 (U10).
093**	Error in TIMER.
094	Parity error in tool table.
095**	Parity error in general parameters.
096**	Parity error in Z axis parameters.
097	Without function.
098**	Parity error in X axis parameters.
099**	Parity error in M decoded table.
100 U41 101 U40	malfunctions in RAM CMOS memory of central microprocessor
102	Without function.
103	Without function.
104	Withour function.
105	More than: 43 characters in a comment, 5 characters to define program number, 4 characters to define block number or extrange characters in memory.
106**	Internal temperature limit overrun.

108** Error in Z axis leadscrew error compensation parameters.

110** Error in X axis leadscrew error compensation parameters.

NOTE: Errors identified by an asterisk (*) deactivate enables and cancel the analog outputs.

Errors identified by two asterisks (**); besides deactivating enables and cancelling analog outputs, they activate the **EMERGENCY** output, setting the CNC to initial conditions.

Errors 094,095**,096**,098**,099**,108**,110** are originated by a **CHECKSUM ERROR** in their relevant area (tool-table, parameters, M-table, leadscrew compensation parameters).

The **CHECKSUM ERROR** usual cause is a battery malfunction, generating a loss of the recorded values. To reset the normal operation.

. Enter again the tool-table, machine-parameter, the decode M functions table, the leadscrew compensation parameters, according to the code displayed by the CNC.

. Keep the CNC on for 4-5 hours to enable the battery to recover the proper voltage level.

WARRANTY

With service contract: According to contract clauses.

Without service contract:

The equipment is under warranty for 15 months from factory delivery date.

This warranty covers both material and labor repair costs at FAGOR.

In case of repair at customer's workshop, any travel expenses are payable by the customer.

This warranty does not cover damages and malfunctions arised from causes not related to normal operation of the equipment, such as blows, poor assembling or handling by untrained personnel, etc.

IMPORTANT NOTE

AURKI S.COOP.LTDA. (in Spain) periodically offers CNC operating and programming courses. They are oriented mainly to those end-users who would like to obtain the maximum benefit from all the features that this CNC offers.

If interested, contact the Communication Department of AURKI S.COOP.LTDA. in Spain by mail or telephone: Phone No. 943 - 799511.