

CNC

8037 .T.

Programming manual

Ref. 1310

Soft: V01.4x



FAGOR AUTOMATION



All rights reserved. No part of this documentation may be transmitted, transcribed, stored in a backup device or translated into another language without Fagor Automation's consent. Unauthorized copying or distributing of this software is prohibited.

The information described in this manual may be subject to changes due to technical modifications. Fagor Automation reserves the right to change the contents of this manual without prior notice.

All the trade marks appearing in the manual belong to the corresponding owners. The use of these marks by third parties for their own purpose could violate the rights of the owners.

It is possible that CNC can execute more functions than those described in its associated documentation; however, Fagor Automation does not guarantee the validity of those applications. Therefore, except under the express permission from Fagor Automation, any CNC application that is not described in the documentation must be considered as "impossible". In any case, Fagor Automation shall not be held responsible for any personal injuries or physical damage caused or suffered by the CNC if it is used in any way other than as explained in the related documentation.

The content of this manual and its validity for the product described here has been verified. Even so, involuntary errors are possible, hence no absolute match is guaranteed. However, the contents of this document are regularly checked and updated implementing the necessary corrections in a later edition. We appreciate your suggestions for improvement.

The examples described in this manual are for learning purposes. Before using them in industrial applications, they must be properly adapted making sure that the safety regulations are fully met.

This product uses the following source code, subject to the terms of the GPL license. The applications *busybox* V0.60.2; *dosfstools* V2.9; *linux-ftpd* V0.17; *ppp* V2.4.0; *utelnx* V0.1.1. The library *grx* V2.4.4. The linux kernel V2.4.4. The linux boot *ppcboot* V1.1.3. If you would like to have a CD copy of this source code sent to you, send 10 Euros to Fagor Automation for shipping and handling.

INDEX

About the product	7
Declaration of conformity	9
Version history	11
Safety conditions	13
Warranty terms	17
Material returning terms	19
Additional remarks	21
Fagor documentation	23
 CHAPTER 1	 GENERAL CONCEPTS
1.1 Part programs	26
1.1.1 Considerations regarding the Ethernet connection	28
1.2 DNC connection	29
1.3 Communication protocol via DNC or peripheral device	30
 CHAPTER 2	 CREATING A PROGRAM
2.1 Program structure at the CNC	32
2.1.1 Block header	32
2.1.2 Program block	33
2.1.3 End of block	34
 CHAPTER 3	 AXES AND COORDINATE SYSTEMS
3.1 Axis nomenclature	36
3.2 Plane selection (G16, G17, G18, G19)	37
3.3 Part dimensioning. Millimeters (G71) or inches (G70)	38
3.4 Absolute/incremental programming (G90, G91)	39
3.5 Programming in radius or in diameters (G152, G151)	40
3.6 Coordinate programming	41
3.6.1 Cartesian coordinates	42
3.6.2 Polar coordinates	43
3.6.3 Angle and Cartesian coordinate	45
3.7 Rotary axes	46
3.8 Work zones	47
3.8.1 Definition of the work zones	47
3.8.2 Using the work zones	48
 CHAPTER 4	 REFERENCE SYSTEMS
4.1 Reference points	49
4.2 Machine reference (Home) search (G74)	50
4.3 Programming with respect to machine zero (G53)	51
4.4 Coordinate preset and zero offsets	52
4.4.1 Coordinate preset and S value limitation (G92)	53
4.4.2 Zero offsets (G54..G59 and G159)	54
4.5 Polar origin preset (G93)	56
 CHAPTER 5	 ISO CODE PROGRAMMING
5.1 Preparatory functions	58
5.2 Feedrate F	60
5.2.1 Feedrate in mm/min or inches/min (G94)	61
5.2.2 Feedrate in mm/rev. or inches/rev (G95)	62
5.3 Spindle speed (S)	63
5.3.1 Constant surface speed (G96)	64
5.3.2 Spindle speed in rpm (G97)	65
5.4 Tool number (T) and tool offset (D)	66

5.5	Auxiliary function (M)	68
5.5.1	M00. Program stop	69
5.5.2	M01. Conditional program stop	69
5.5.3	M02. End of program	70
5.5.4	M30. End of program with return to the first block	70
5.5.5	M03 Clockwise spindle rotation	71
5.5.6	M04. Counterclockwise spindle rotation	71
5.5.7	M05. Spindle stop	72
5.5.8	M06. Tool change code	72
5.5.9	M19. Spindle orientation	73
5.5.10	M41, M42, M43, M44. Spindle gear change	74

CHAPTER 6

PATH CONTROL

6.1	Rapid traverse (G00)	76
6.2	Linear interpolation (G01)	77
6.3	Circular interpolation (G02, G03)	78
6.4	Circular interpolation with absolute arc center coordinates (G06)	82
6.5	Arc tangent to previous path (G08)	83
6.6	Arc defined by three points (G09)	84
6.7	Tangential entry at the beginning of a machining operation (G37)	85
6.8	Tangential exit at the end of a machining operator (G38)	86
6.9	Automatic radius blend (G36)	87
6.10	Chamfer (G39)	88
6.11	Threading (G33)	89
6.12	Withdrawal of axes when interrupting a threading operation (G233)	92
6.13	Variable pitch threads (G34)	94
6.14	Move to hardstop (G52)	95
6.15	Feedrate "F" as an inverted function of time (G32)	96

CHAPTER 7

ADDITIONAL PREPARATORY FUNCTIONS

7.1	Interruption of block preparation (G04)	97
7.1.1	G04 K0: Block preparation interruption and coordinate update	99
7.2	Dwell (G04 K)	100
7.3	Working with square (G07) and round (G05,G50) corners	101
7.3.1	G07 (square corner)	101
7.3.2	G05 (round corner)	102
7.3.3	Controlled round corner (G50)	103
7.4	Look-ahead (G51)	104
7.4.1	Advanced look-ahead algorithm (integrating Fagor filters)	106
7.4.2	Look-ahead operation with Fagor filters active	107
7.5	Mirror image (G10, G11, G12, G13, G14)	108
7.6	Scaling factor (G72)	109
7.6.1	Scaling factor applied to all axes.	110
7.6.2	Scaling factor applied to one or more axes	111

CHAPTER 8

TOOL COMPENSATION

8.1	Tool length compensation	113
8.2	Tool radius compensation	114
8.2.1	The location code of the tool (tool type)	115
8.2.2	Working without tool radius compensation	118
8.2.3	Working with tool radius compensation	119
8.2.4	Beginning of tool radius compensation (G41, G42)	120
8.2.5	Sections of tool radius compensation	123
8.2.6	Cancellation of tool radius compensation (G40)	124
8.2.7	Temporary cancellation of tool compensation with G00	128
8.2.8	Change of type of tool radius compensation while machining	130
8.2.9	Tool compensation in any plane	131
8.3	Collision detection (G41 N, G42 N)	132

CHAPTER 9

CANNED CYCLES

9.1	G66. Pattern repeat cycle canned cycle	134
9.1.1	Basic operation	137
9.1.2	Profile programming syntax	139
9.2	G68. X axis roughing canned cycle	140
9.2.1	Basic operation	143
9.2.2	Profile programming syntax	146
9.3	G69. Z axis roughing canned cycle	147
9.3.1	Basic operation	150
9.3.2	Profile programming syntax	153



CNC 8037

SOFT: V01.4x

9.4	G81. Turning canned cycle for straight sections.....	154
9.4.1	Basic operation	156
9.5	G82. Facing canned cycle for straight sections	158
9.5.1	Basic operation	160
9.6	G83. Axial drilling and tapping canned cycle	162
9.6.1	Basic operation	164
9.7	G84. Turning canned cycle for curved sections.....	165
9.7.1	Basic operation	167
9.8	G85. Facing canned cycle for curved sections	169
9.8.1	Basic operation	171
9.9	G86. Longitudinal threading canned cycle.....	173
9.9.1	Basic operation	178
9.10	G87. Face threading canned cycle	179
9.10.1	Basic operation	185
9.11	G88. X axis grooving canned cycle.....	186
9.11.1	Basic operation	187
9.12	G89. Z axis grooving canned cycle.....	188
9.12.1	Basic operation	189

CHAPTER 10 PROBING

10.1	Probing (G75, G76).....	192
------	-------------------------	-----

CHAPTER 11 HIGH-LEVEL LANGUAGE PROGRAMMING

11.1	Lexical description	193
11.2	Variables	195
11.2.1	General purpose parameters or variables.....	196
11.2.2	Variables associated with tools	198
11.2.3	Variables associated with zero offsets	201
11.2.4	Variables associated with machine parameters.....	202
11.2.5	Variables associated with work zones	203
11.2.6	Variables associated with feedrates.....	205
11.2.7	Variables associated with coordinates	207
11.2.8	Variables associated with electronic handwheels	209
11.2.9	Variables associated with feedback	211
11.2.10	Variables associated with the main spindle	212
11.2.11	PLC related variables	215
11.2.12	Variables associated with local parameters	217
11.2.13	Operating-mode related variables.....	218
11.2.14	Other variables.....	221
11.3	Constants.....	225
11.4	Operators	226
11.5	Expressions	228
11.5.1	Arithmetic expressions.....	228
11.5.2	Relational expressions	229

CHAPTER 12 PROGRAM CONTROL INSTRUCTIONS

12.1	Assignment instructions	232
12.2	Display instructions	233
12.3	Enable-disable instructions	234
12.4	Flow control instructions	235
12.5	Subroutine instructions	237
12.6	Interruption-subroutine instructions.....	241
12.7	Program instructions	242
12.8	Screen customizing instructions	245

CHAPTER 13 ANGULAR TRANSFORMATION OF AN INCLINE AXIS

13.1	Turning angular transformation on and off.....	253
13.2	Freezing the angular transformation	254

APPENDIX

A	ISO code programming.....	257
B	Program control instructions	259
C	Summary of internal CNC variables.....	261
D	Key code	267
E	Maintenance	269



CNC 8037

SOFT: V01.4x



CNC 8037

SOFT: V01.4x

ABOUT THE PRODUCT

BASIC CHARACTERISTICS

Monitor	7.5" Color LCD
Block processing time	7 ms
Look-ahead	75 blocks
RAM memory	1 Mb
Flash memory	128 MB
PLC cycle time	3 ms / 1000 instructions
Minimum position loop	4 ms
USB	Standard
RS-232 serial line	Standard
DNC (via RS232)	Standard
Ethernet	Option
5 V or 24 V probe inputs	2
Local digital inputs and outputs.	16 I / 8 O 40 I / 24 O 56 I / 32 O
Feedback inputs for the axes and spindle	4 TTL / 1Vpp inputs
Feedback inputs for handwheels	2 TTL inputs
Analog outputs	4 for axes and spindle
CAN servo drive system for Fagor servo drive connection.	Option
Remote CAN modules, for digital I/O expansion (RIO).	Option



Before start-up, verify that the machine that integrates this CNC meets the 89/392/CEE Directive.



CNC 8037

SOFTWARE OPTIONS

	Model		
	M	T	TC
Number of axes	3	2	2
Number of spindles	1	1	1
Electronic threading	Standard	Standard	Standard
Tool magazine management:	Standard	Standard	Standard
Machining canned cycles	Standard	Standard	Standard
Multiple machining	Standard	-----	-----
Rigid tapping	Standard	Standard	Standard
DNC	Standard	Standard	Standard
Tool radius compensation	Standard	Standard	Standard
Retracing	Standard	-----	-----
Jerk control	Standard	Standard	Standard
Feed forward	Standard	Standard	Standard
Oscilloscope function (Setup assistance)	Standard	Standard	Standard
Roundness test (Setup assistance)	Standard	Standard	Standard

About the product



CNC 8037

DECLARATION OF CONFORMITY

The manufacturer:

Fagor Automation S. Coop.

Barrio de San Andrés Nº 19, C.P. 20500, Mondragón -Guipúzcoa- (SPAIN).

Declares:

Under their responsibility that the product:

8037 CNC

Consisting of the following modules and accessories:

8037-M, 8037-T, 8037-TC

Remote modules RIO

ETHERNET, ETHERNET-CAN-CAN AXES, ETHERNET-CAN AXES

Note. Some additional characters may follow the references mentioned above. They all comply with the directives listed. However, check that that's the case by checking the label of the unit itself.

Referred to by this declaration with following directives:

Low voltage regulations.

EN 60204-1: 2006 Electrical equipment on machines — Part 1. General requirements.

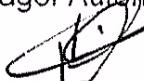
Regulation on electromagnetic compatibility.

EN 61131-2: 2007 PLC — Part 2. Requirements and equipment tests.

As instructed by the European Community Directives 2006/95/EEC on Low Voltage and 2004/108/EC on Electromagnetic Compatibility and its updates.

In Mondragón, March 14th, 2012

Fagor Automation, S. Coop.


Director Gerente
Pedro Ruiz de Aguirre

FAGOR 

CNC 8037

VERSION HISTORY

Here is a list of the features added in each software version and the manuals that describe them.

The version history uses the following abbreviations:

INST	Installation manual
PRG	Programming manual
OPT	Operating manual
OPT-TC	Operating manual for the TC option.

Software V01.42

March 2012

First version.



Version history



CNC 8037

SAFETY CONDITIONS

Read the following safety measures in order to prevent harming people or damage to this product and those products connected to it.

This unit may only be repaired by authorized personnel at Fagor Automation.

Fagor Automation shall not be held responsible of any physical damage or defective unit resulting from not complying with these basic safety regulations.

PRECAUTIONS AGAINST PERSONAL DAMAGE

- Interconnection of modules.
Use the connection cables provided with the unit.
- Use proper Mains AC power cables
To avoid risks, use only the Mains AC cables recommended for this unit.
- Avoid electrical overloads.
In order to avoid electrical discharges and fire hazards, do not apply electrical voltage outside the range selected on the rear panel of the central unit.
- Ground connection.
In order to avoid electrical discharges, connect the ground terminals of all the modules to the main ground terminal. Before connecting the inputs and outputs of this unit, make sure that all the grounding connections are properly made.
- Before powering the unit up, make sure that it is connected to ground.
In order to avoid electrical discharges, make sure that all the grounding connections are properly made.
- Do not work in humid environments.
In order to avoid electrical discharges, always work under 90% of relative humidity (non-condensing) and 45 °C (113° F).
- Do not work in explosive environments.
In order to avoid risks or damages, do no work in explosive environments.

PRECAUTIONS AGAINST PRODUCT DAMAGE

- Working environment.

This unit is ready to be used in industrial environments complying with the directives and regulations effective in the European Community.

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

- Install this unit in the proper place.

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

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

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

- Enclosures.

The manufacturer is responsible of assuring that the enclosure involving the equipment meets all the currently effective directives of the European Community.

- Avoid disturbances coming from the machine tool.

The machine-tool must have all the interference generating elements (relay coils, contactors, motors, etc.) uncoupled.

- DC relay coils. Diode type 1N4000.
- AC relay coils. RC connected as close to the coils as possible with approximate values of $R=220\ \Omega$ / $1\ W$ y $C=0,2\ \mu F$ / $600\ V$.
- AC motors. RC connected between phases, with values of $R=300\ \Omega$ / $6\ W$ y $C=0,47\ \mu F$ / $600\ V$.

- Use the proper power supply.

Use an external regulated 24 Vdc power supply for the inputs and outputs.

- Grounding of the power supply.

The zero volt point of the external power supply must be connected to the main ground point of the machine.

- Analog inputs and outputs connection.

It is recommended to connect them using shielded cables and connecting their shields (mesh) to the corresponding pin.

- Ambient conditions.

The working temperature must be between $+5\ ^\circ C$ and $+40\ ^\circ C$ ($41^\circ F$ and $104^\circ F$)

The storage temperature must be between $-25\ ^\circ C$ and $+70\ ^\circ C$. ($-13\ ^\circ F$ and $158\ ^\circ F$)

- Central unit enclosure (8037 CNC).

Make sure that the needed gap is kept between the central unit and each wall of the enclosure. Use a DC fan to improve enclosure ventilation.

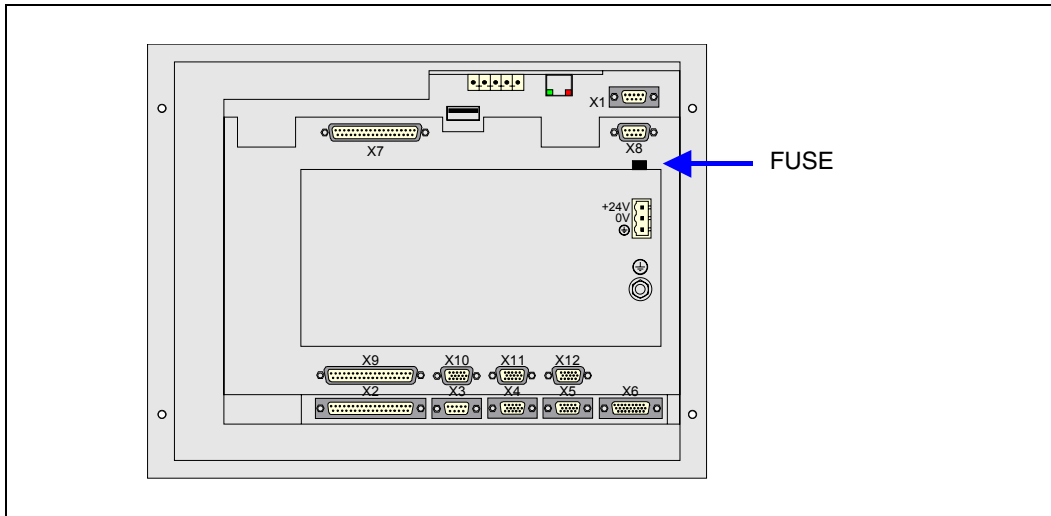
- Power switch.

This power switch must be mounted in such a way that it is easily accessed and at a distance between 0.7 meters (27.5 inches) and 1.7 meters (5.5ft) off the floor.

PROTECTIONS OF THE UNIT ITSELF (8037)

- Central Unit.

It has a 4 A 250V external fast fuse (F).



- Inputs-Outputs.

All the digital inputs and outputs have galvanic isolation via optocouplers between the CNC circuitry and the outside.

Safety conditions

PRECAUTIONS DURING REPAIR



Do not get into the inside of the unit. Only personnel authorized by Fagor Automation may manipulate the inside of this unit.

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

SAFETY SYMBOLS

- Symbols which may appear on the manual.



Symbol for danger or prohibition.

It indicates actions or operations that may cause damage to people or to units.



Warning symbol.

It indicates situations that may be caused by certain operations and the actions to be taken to prevent them.



Obligation symbol.

It indicates actions and operations that must be carried out.



Information symbol.

It indicates notes, warnings and advises.

WARRANTY TERMS

INITIAL WARRANTY

All products manufactured or marketed by FAGOR carry a 12-month warranty for the end user which could be controlled by the our service network by means of the warranty control system established by FAGOR for this purpose.

In order to prevent the possibility of having the time period from the time a product leaves our warehouse until the end user actually receives it run against this 12-month warranty, FAGOR has set up a warranty control system based on having the manufacturer or agent inform FAGOR of the destination, identification and on-machine installation date, by filling out the document accompanying each FAGOR product in the warranty envelope. This system, besides assuring a full year of warranty to the end user, enables our service network to know about FAGOR equipment coming from other countries into their area of responsibility.

The warranty starting date will be the one appearing as the installation date on the above mentioned document. FAGOR offers the manufacturer or agent 12 months to sell and install the product. This means that the warranty starting date may be up to one year after the product has left our warehouse so long as the warranty control sheet has been sent back to us. This translates into the extension of warranty period to two years since the product left our warehouse. If this sheet has not been sent to us, the warranty period ends 15 months from when the product left our warehouse.

This warranty covers all costs of material and labour involved in repairs at FAGOR carried out to correct malfunctions in the equipment. FAGOR undertakes to repair or replace their products within the period from the moment manufacture begins until 8 years after the date on which it disappears from the catalog.

FAGOR has exclusive competence in deciding whether the repair enters within the term defined as the warranty period.

EXCLUDING CLAUSES

Repairs will be carried out on our premises. Therefore, all expenses incurred as a result of trips made by technical personnel to carry out equipment repairs, despite these being within the above-mentioned period of warranty, are not covered by the warranty.

Said warranty will be applied whenever the equipment has been installed in accordance with instructions, has not be mistreated, has not been damaged by accident or by negligence and has not been tampered with by personnel not authorized by FAGOR. If, once servicing or repairs have been made, the cause of the malfunction cannot be attributed to said elements, the customer is obliged to cover the expenses incurred, in accordance with the tariffs in force.

Other warranties, implicit or explicit, are not covered and FAGOR AUTOMATION cannot be held responsible for other damages which may occur.



CNC 8037

WARRANTY ON REPAIRS

In a similar way to the initial warranty, FAGOR offers a warranty on standard repairs according to the following conditions:

PERIOD	12 months.
CONCEPT	Covers parts and labor for repairs (or replacements) at the network's own facilities.
EXCLUDING CLAUSES	The same as those applied regarding the chapter on initial warranty. If the repair is carried out within the warranty period, the warranty extension has no effect.

When the customer does not choose the standard repair and just the faulty material has been replaced, the warranty will cover just the replaced parts or components within 12 months.

For sold parts the warranty is 12 months length.

MAINTENANCE CONTRACTS

The SERVICE CONTRACT is available for the distributor or manufacturer who buys and installs our CNC systems.

MATERIAL RETURNING TERMS

When sending the central nit or the remote modules, pack them in its original package and packaging material. If the original packaging material is not available, pack it as follows:

1. Get a cardboard box whose three inside dimensions are at least 15 cm (6 inches) larger than those of the unit. The cardboard being used to make the box must have a resistance of 170 kg. (375 pounds).
2. Attach a label indicating the owner of the unit, person to contact, type of unit and serial number.
3. In case of failure, also indicate the symptom and a short description.
4. Wrap the unit in a polyethylene roll or similar material to protect it.
5. When sending the central unit, protect especially the screen.
6. Pad the unit inside the cardboard box with polyurethane foam on all sides.
7. Seal the cardboard box with packing tape or industrial staples.

Material returning terms



CNC 8037

ADDITIONAL REMARKS

Mount the CNC away from coolants, chemical products, blows, etc. which could damage it. Before turning the unit on, verify that the ground connections have been properly made.

In case of a malfunction or failure, disconnect it and call the technical service. Do not get into the inside of the unit.

Additional remarks



CNC 8037

FAGOR DOCUMENTATION

OEM manual

It is directed to the machine builder or person in charge of installing and starting-up the CNC.

USER-M manual

Directed to the end user.

It describes how to operate and program in M mode.

USER-T manual

Directed to the end user.

It describes how to operate and program in T mode.

TC Manual

Directed to the end user.

It describes how to operate and program in TC mode.

It contains a self-teaching manual.



CNC 8037

The CNC may be programmed at the machine (from the front panel) and from a peripheral (computer). Memory available to the user for carrying out the part programs is 1 Mbyte.

The part programs and the values in the tables which the CNC has can be entered from the front panel, from a pc (DNC) or from a peripheral.

Entering programs and tables from the front panel.

Once the editing mode or desired table has been selected, the CNC allows you to enter data from the keyboard.

Entering programs and tables from a Computer (DNC) or peripheral device.

The CNC allows data to be exchanged with a computer or peripheral device, using the RS232C serial line.

If this is controlled from the CNC, it is necessary to preset the corresponding table or part program directory (utilities) you want to communicate with.

Depending on the type of communication required, the serial port machine parameter "PROTOCOL" should be set.

"PROTOCOL" = 0 If the communication is with a peripheral device.

"PROTOCOL" = 1 If the communication is via DNC.

1.1 Part programs

The operating manual describes the different operating modes. Refer to that manual for further information.

Editing a part-program

To create a part-program, access the –Edit– mode.

The new part-program edited is stored in the CNC's RAM memory. A copy of the part-programs may be stored in the hard disk (KeyCF) at a PC connected through the serial line or in the USB disk.

To transmit a program to a PC through the serial, proceed as follows:

1. Execute the "WinDNC.exe" application program at the PC.
2. Activate DNC communications at the CNC.
3. Select the work directory at the CNC. It is selected from the –Utilities– mode, option Directory \Serial L \Change directory

In –Edit– mode, it is possible to modify part-programs residing in the CNC's RAM memory. To modify a program stored in the hard disk (KeyCF), in a PC or in the USB disk, it must be previously copied into RAM memory.

Executing and editing a part-program

Part-programs stored anywhere may be executed or simulated. Simulation is carried out in the –Simulation– mode, whereas the execution is done in the –Automatic– mode

When executing or simulating a part-program, bear in mind the following points:

- Only subroutines stored in the CNC's RAM memory can be executed. Therefore, to execute a subroutine stored in the hard disk (KeyCF), in a PC or in the USB disk, it must be first copied into the CNC's RAM memory.
- The GOTO and RPT instructions cannot be used in programs that are executed from a PC connected through the serial line.
- From a program in execution, it is possible to execute another program located in RAM memory, in the hard disk (KeyCF) or in a PC using the EXEC instruction.

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

–Utilities– operating mode.

The –Utilities– mode, lets display the part-program directory of all the devices, make copies, delete, rename and even set the protections for any of them.

Operations that may be carried out with part-programs.

	RAM memory	Hard disk	DNC
See the program directory of ...	Yes	Yes	Yes
See the subroutine directory of ...	Yes	No	No
Create the work directory from ...	No	No	No
Change the work directory from ...	No	No	Yes
Edit a program from ...	Yes	Yes	No
Modify a program from ...	Yes	Yes	No
Delete a program from ...	Yes	Yes	Yes
Copy from/to RAM memory to/from ...	Yes	Yes	Yes
Copy from/to HD to/from ...	Yes	Yes	Yes
Copy from/to DNC to/from ...	Yes	Yes	Yes
Rename a program from ...	Yes	Yes	No
Change the comment of a program from ...	Yes	Yes	No
Change the protections of a program from ...	Yes	Yes	No
Execute a part-program from ...	Yes	Yes	Yes
Execute a user program from ...	Yes	Yes	No
Execute a PLC program from ...	Yes	No	No
Execute programs with GOTO or RPT instructions from ...	Yes	Yes	No
Execute subroutines residing in ...	Yes	No	No
Execute programs with the EXEC instruction, in RAM from ...	Yes	Yes	Yes
Execute programs with the EXEC instruction, in HD from ...	Yes	Yes	Yes
Execute programs with the EXEC instruction, in DNC from ...	Yes	Yes	No
Open programs with the OPEN instruction, in RAM from ...	Yes	Yes	Yes
Open programs with the OPEN instruction, in HD from ...	Yes	Yes	Yes
Open programs with the OPEN instruction, in DNC from ...	Yes	Yes	No
Via Ethernet:			
See from a PC the program directory of ...	No	Yes	No
See from a PC the subroutine directory of ...	No	No	No
See from a PC, a directory in ...	No	No	No

(*) If it is not in RAM memory, it generates the executable code in RAM and it executes it.

Ethernet

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

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

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

1.

GENERAL CONCEPTS Part programs



CNC 8037

· T· MODEL
SOFT: V01.4x

1.1.1 Considerations regarding the Ethernet connection

When configuring the CNC as another node in the computer network, the programs stored in the hard disk (KeyCF) may be edited and modified from any PC.

Instructions for setting up a PC to access CNC directories

To set up the PC to access the CNC directories, we recommend to proceed as follows.

1. Open the "Windows Explorer"
2. On the "Tools" menu, select the "Connect to Network Drives" option.
3. Select the drive, for example "D".
4. Indicate the path. The path will be the CNC name followed by the name of the shared directory.
For example: \\FAGORCNC\CNCHD
5. When selecting the option: "Connect again when initiating the session", the selected CNC will appear on each power-up as another path of the "Windows Explorer" without having to define it again.

Data format

This connection is established through Ethernet and, therefore, the CNC does not control the syntax of the programs while they are received or modified. However, whenever accessing the program directory of the Hard Disk (HD), the following verification takes place:

File name.

The file number must always have 6 digits and the extension PIM (for milling) or PIT (for lathe).

Examples: 001204.PIM 000100.PIM 123456.PIT 020150.PIT

If the file has been given the wrong name, for example: 1204.PIM or 100.PIT, the CNC will not change it, but it will display it with the comment "*****". The file name cannot be modified at the CNC; it must be edited from the PC to correct the error.

File size.

If the file is empty (size = 0) the CNC will display it with the comment "*****".

The file can be edited or deleted either from the CNC or from the PC.

First line of the program.

The first line of the program must have the % character, the comment associated with the file (up to 20 characters) and between the two commas (,) the program attributes O (OEM), H (hidden), M (modifiable), X (executable).

Examples: %Comment ,MX,
% ,OMX,

If the first line does not exist, the CNC will display the program with an empty comment and with the modifiable (M) and executable (X) attributes.

When the format of the first line is wrong, the CNC does not modify it, but it displays it with the comment "*****". The file can be edited or deleted either from the CNC or from the PC.

The format is incorrect when the comment has more than 20 characters, a comma (,) is missing to group the attributes or there is a strange character in the attributes.

1.

GENERAL CONCEPTS
Part programs



CNC 8037

·T· MODEL
SOFT: V01.4x

1.2 DNC connection

The CNC offers as optional feature the possibility of working in DNC (Distributed Numerical Control), enabling communication between the CNC and a computer to carry out the following functions:

- Directory and delete commands.
- Transfer of programs and tables between the CNC and a computer.
- Remote control of the machine.
- The ability to supervise the status of advanced DNC systems.

1.

GENERAL CONCEPTS
DNC connection



CNC 8037

·T· MODEL
SOFT: V01.4x

1.3 Communication protocol via DNC or peripheral device

This type of communication enables program-and-table transfer commands, plus the organization of CNC directories such as the computer directory, for copying/deleting programs, etc. to be done either from the CNC or the computer.

When you want to transfer files, it is necessary to follow this protocol:

- The "%" symbol will be used to start the file, followed by the program comment (optional), of up to 20 characters.

Then, and separated by a comma ",", comes the protection of each file, read, write, etc. These protections are optional and need not be programmed.

To end the file header, RT (RETURN) or LF (LINE FEED) characters should be sent separated by a comma (",").

Example: %Fagor Automation, MX, RT

- Following the header, the file blocks should be programmed. These will all be programmed according to the programming rules indicated in this manual. After each block, to separate it from the others, the RT (RETURN) or LF (LINE FEED) characters should be used.

Example: N20 G90 G01 X100 Y200 F2000 LF
 (RPT N10, N20) N3 LF

If communication is made with a peripheral device, you will need to send the 'end of file' command. This command is selected via the machine parameter for the serial port: "EOFCHR", and can be one of the following characters :

ESC	ESCAPE
EOT	END OF TRANSMISSION
SUB	SUBSTITUTE
EXT	END OF TRANSMISSION

1.

GENERAL CONCEPTS

Communication protocol via DNC or peripheral device



CNC 8037

·T· MODEL
SOFT: V01.4x

CREATING A PROGRAM

2

A CNC program consists of a series of blocks or instructions. These blocks or instructions are made of words composed of capital letters and numerical format.

The CNC's numerical format consists of :

- The signs . (decimal points), + (plus), - (minus).
- Digits 0 1 2 3 4 5 6 7 8 9.

Programming allows spaces between letters, numbers and symbols, in addition to ignoring the numerical format if it has zero value, or a symbol if it is positive.

The numeric format of a word may be replaced by an arithmetic parameter when programming. Later on, during execution, the CNC will replace the arithmetic parameter by its value. For example, if XP3 has been programmed, during execution the CNC will replace P3 by its numerical value, obtaining results such as X20, X20.567, X-0.003, etc.

2.1 Program structure at the CNC

All the blocks which make up the program have the following structure:

Block header + program block + end of block

2.

CREATING A PROGRAM

Program structure at the CNC

2.1.1 Block header

The block header is optional, and may consist of one or more block skip conditions and by the block number or label. Both must be programmed in this order.

Block skip condition. *"/", "/1", "/2", "/3".*

These three block skip conditions, given that "/" and "/1" are the same, they are governed by the marks BLKSKIP1, BLKSKIP2 and BLKSKIP3 of the PLC. If any of these marks is active, the CNC will not execute the block or blocks in which it has been programmed; the execution takes place in the following block.

Up to 3 skip conditions can be programmed in one block; they will be evaluated one by one, respecting the order in which they have been programmed.

The control reads 200 blocks ahead of the one being executed in order to calculate in advance the path to be run. The condition for block skip will be analyzed at the time when the block is read i.e. 200 blocks before execution.

If the block skip needs to be analyzed at the time of execution, it is necessary to interrupt the block preparation, by programming G4 in the previous block.

Label or block number. *N(0-99999999).*

This is used to identify the block, and is only used when block references or jumps are made. They are represented by the letter N followed by up to 8 digits (0-99999999).

No particular order is required and the numbers need not be sequential. If two or more blocks with the same label number are present in the same program, the CNC will always give priority to the first number.

Although it is not necessary to program it, by using a softkey the CNC allows the automatic programming of labels. The programmer can select the initial number and the step between labels.

Restrictions:

- Displaying the number of the active block in the top window of the screen:
 - When executing a program in ISO mode, when the label number is higher than 9999, it displays N****.
 - On the "DISPLAY / SUBROUTINES" window, when displaying an RPT that has a label higher than 9999, it displays it with ****.
- Canned cycles G66, G68 and G69 can only be edited using 4-digit labels.



CNC 8037

·T· MODEL
SOFT: V01.4x

2.1.2 Program block

This is written with commands in ISO and high level languages. To prepare a program, blocks written in both languages will be used, although each one should be edited with commands in just one language.

ISO language

This language is specially designed to control axis movement, as it gives information and movement conditions, in addition to data on feedrate. It offers the following types of functions.

- Preparatory functions for movement, used to determine geometry and working conditions, such as linear and circular interpolations, threading, etc.
- Control functions for axis feedrate and spindle speeds.
- Tool control functions.
- Complementary functions, with technological instructions.

High level language

This enables access to general purpose variables and to system tables and variables.

It gives the user a number of control sentences which are similar to the terminology used in other languages, such as IF, GOTO, CALL, etc. It also allows the use of any type of expression (arithmetic, referential, or logical).

It also has instructions for the construction of loops, plus subroutines with local variables. A local variable is one that is only recognized by the subroutine in which it has been defined.

It is also possible to create libraries, grouping subroutines with useful and tested functions, which can be accessed from any program.

2.

CREATING A PROGRAM

Program structure at the CNC

2.1.3 End of block

The end of block is optional and may consist of the indication of number of repetitions of the block and of the block comment. Both must be programmed in this order.

Number of block repetitions. N(0-9999)

This indicates the number of times the block will be executed. The number of repetitions is represented by the letter N followed by up to 4 digits (0-9999). The active machining operation does not take place if N0 is programmed; only the movement programmed within the block takes place.

Movement blocks can only be repeated which, at the time of their execution, are under the influence of a modal subroutine. In these cases, the CNC executes the programmed move and the active machining operation (canned cycle or modal subroutine) the indicated number of times.

Block comment

The CNC allows you to incorporate any kind of information into all blocks in the form of a comment. The comment is programmed at the end of the block, and should begin with the character ";" (semicolon).

If a block begins with ";" all its contents will be considered as a comment, and it will not be executed.

Empty blocks are not permitted. They should contain at least one comment.

2.

CREATING A PROGRAM

Program structure at the CNC



CNC 8037

·T· MODEL
SOFT: V01.4x

AXES AND COORDINATE SYSTEMS

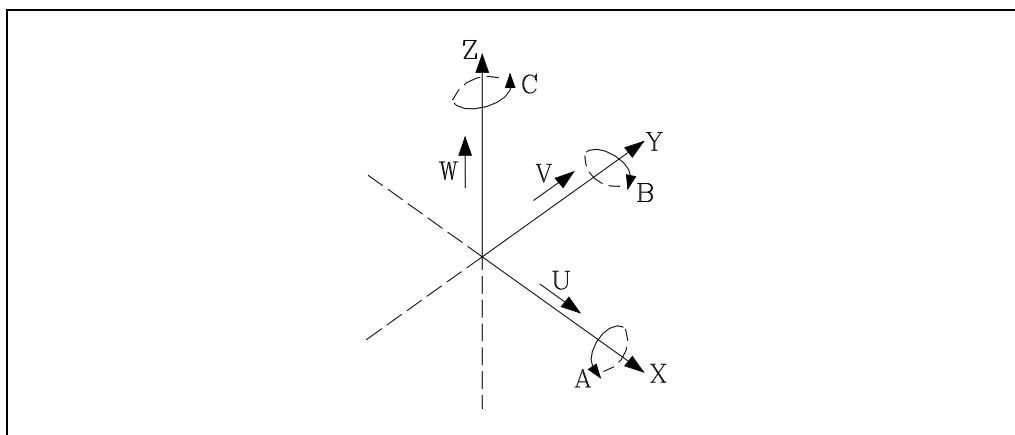
3

Given that the purpose of the CNC is to control the movement and positioning of axes, it is necessary to determine the position of the point to be reached through its coordinates.

The CNC allows you to use absolute, relative or incremental coordinates throughout the same program.

3.1 Axis nomenclature

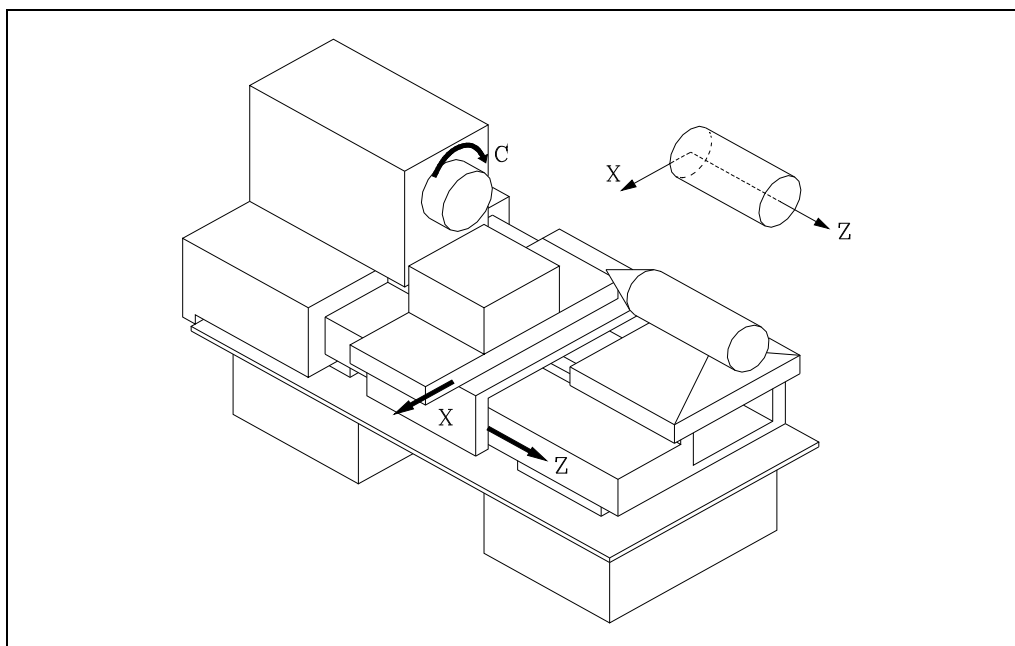
The axes are named according to DIN 66217.



Characteristics of the system of axes:

- X and Y main movements on the main work plane of the machine.
- Z parallel to the main axis of the machine, perpendicular to the main XY plane.
- U, V, W auxiliary axes parallel to X, Y, Z respectively.
- A B C Rotary axes on each axis X, Y, Z.

The drawing below shows an example of the nomenclature of the axes on a parallel lathe.



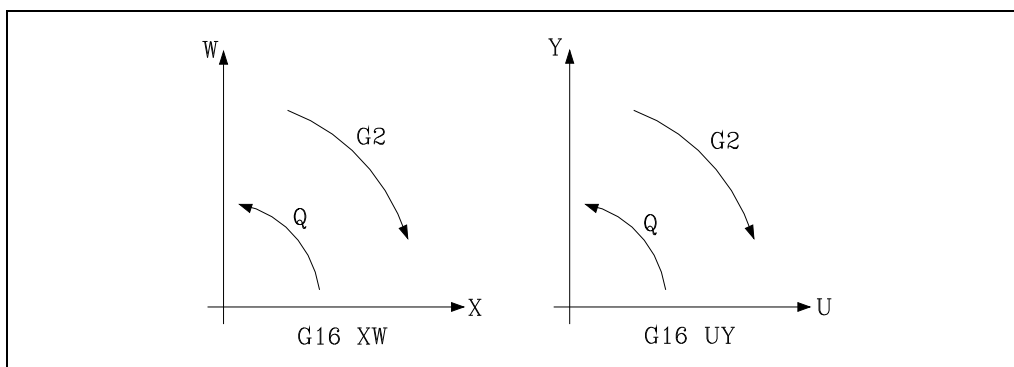
3.2 Plane selection (G16, G17, G18, G19)

Plane selection should be made when the following are carried out :

- Circular interpolations.
- Controlled corner rounding.
- Tangential entry and exit.
- Chamfer.
- Coordinate programming in Polar coordinates.
- Pattern rotation.
- Tool radius Compensation.
- Tool length compensation.

The "G" functions which enable selection of work planes are as follows :

G16 axis1 axis2. Enables selection of the desired work plane, plus the direction of G02 G03 (circular interpolation), axis1 being programmed as the abscissa axis and axis2 as the ordinate axis.

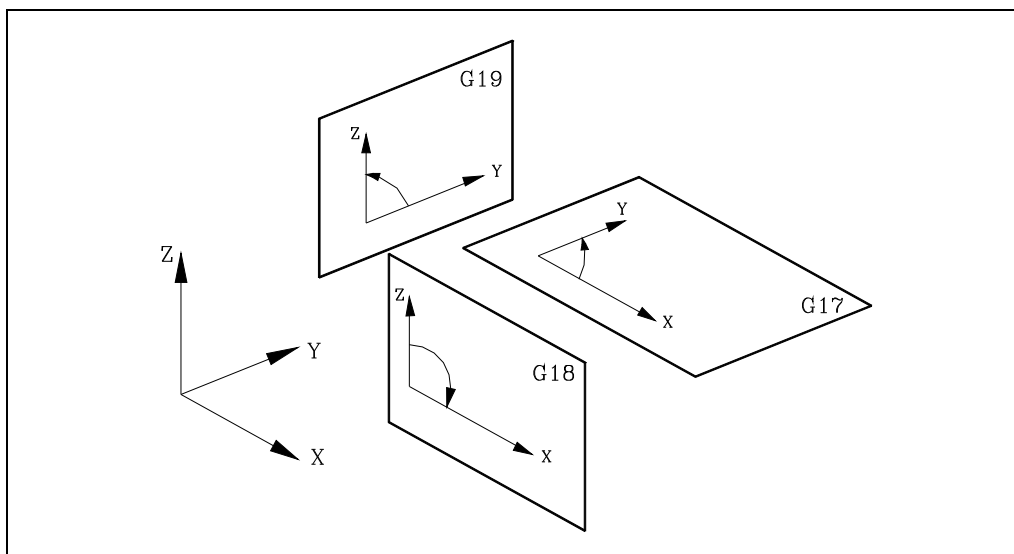


G17. Selects the XY plane.

G18. Selects the ZX plane.

G19. Selects the YZ plane.

The G16, G17, G18 and G19 functions are modal and incompatible among themselves. The G16 function should be programmed on its own within a block.



On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC will assume that the plane defined by the general machine parameter as "IPLANE" is the work plane.

3.

AXES AND COORDINATE SYSTEMS
Plane selection (G16, G17, G18, G19)

FAGOR

CNC 8037

·T· MODEL
SOFT: V01.4x

3.3 Part dimensioning. Millimeters (G71) or inches (G70)

The CNC allows you to enter units of measurement with the programming, either in millimeters or inches.

It has a general machine parameter "INCHES" to define the unit of measurement of the CNC.

However, these units of measurement can be changed at any time in the program. Two functions are supplied for this purpose :

- G70. Programming in inches.
- G71. Programming in millimeters.

Depending on whether G70 or G71 has been programmed, the CNC assumes the corresponding set of units for all the blocks programmed from that moment on.

The G70 and G71 functions are modal and are incompatible.

The CNC allows you to program figures from 0.00001 to 99999.9999 with or without sign, working in millimeters (G71), called format +/-5.4, or either from 0.00001 to 3937.00787 with or without sign if the programming is done in inches (G70), called format +/-4.5.

However, and to simplify the instructions, we can say that the CNC admits +/- 5.5 format, thereby admitting +/- 5.4 in millimeters and +/- 4.5 in inches.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC will assume that the system of units of measurement is the one defined by the general machine parameter "INCHES".

3.

AXES AND COORDINATE SYSTEMS
Part dimensioning. Millimeters (G71) or inches (G70)



CNC 8037

·T· MODEL
SOFT: V01.4x

3.4 Absolute/incremental programming (G90, G91)

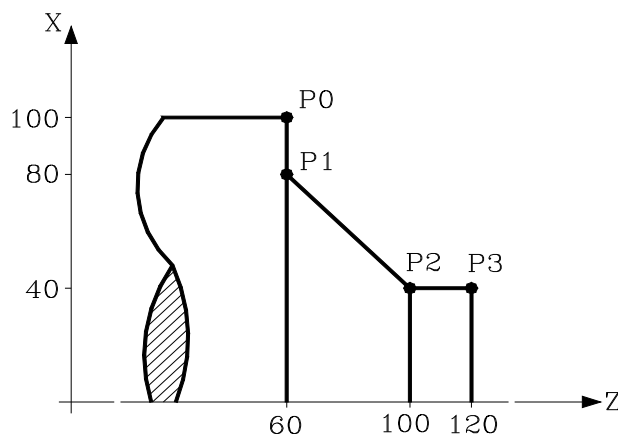
The CNC allows the programming of the coordinates of one point either with absolute G90 or incremental G91 values.

When working with absolute coordinates (G90), the point coordinates refer to a point of origin of established coordinates, often the part zero (datum).

When working in incremental coordinates (G91), the numerical value programmed corresponds to the movement information for the distance to be traveled from the point where the tool is situated at that time. The sign in front shows the direction of movement.

The G90/G91 functions are modal and incompatible with each other.

Example programming the X axis in diameter.



Absolute coordinates:

G90	X200	Z60	; Point P0
	X160	Z60	; Point P1
	X80	Z100	; Point P2
	X80	Z120	; Point P3

Incremental coordinates:

G90	X200	Z60	; Point P0
G91	X-40		; Point P1
	X-80	Z40	; Point P2
	Z20		; Point P3

On power-up, after executing M02, M30 or after an EMERGENCY or RESET, the CNC will assume G90 or G91 according to the definition by the general machine parameter "ISYSTEM".

3.

AXES AND COORDINATE SYSTEMS
Absolute/incremental programming (G90, G91)

FAGOR 

CNC 8037

· T· MODEL
SOFT: V01.4x

3.5 Programming in radius or in diameters (G152, G151)

The X axis coordinates may be programmed in radius or diameter. The following functions may be used for that.

- G151. Programming the X axis in diameter.
- G152. Programming the X axis in radius.

It is carried out by interpolating the main spindle (that is turning) with the Z axis. After the execution of one of these functions, the CNC assumes the relevant programming mode for the blocks programmed afterwards.

The change of units is also taken into account in the following cases.

- Displaying the real X value of in the part's coordinate system.
- Reading of the variable PPOSX (programmed coordinate).

Functions G151 and G152 are modal and incompatible with each other.

On power-up, after executing an M02, M30 or after an emergency or reset, the CNC assumes G151 or G152 depending on the setting of X axis machine parameter "DFORMAT".

3.

AXES AND COORDINATE SYSTEMS

Programming in radius or in diameters (G152, G151)



CNC 8037

·T· MODEL
SOFT: V01.4x

3.6 Coordinate programming

The CNC allows the selection of up to 2 of the 9 possible axes X, Y, Z, U, V, W, A, B, C.

Each of these may be linear, linear to position only, normal rotary, rotary to position only or rotary with hirth toothing (positioning in complete degrees), according to the specification in the machine parameter of each "AXISTYPE" axis.

With the aim of always selecting the most suitable coordinate programming system, the CNC has the following types :

- Cartesian coordinates
- Polar coordinates
- Angle and Cartesian coordinate

3.

AXES AND COORDINATE SYSTEMS
Coordinate programming



CNC 8037

·T· MODEL
SOFT: V01.4x

3.6.1 Cartesian coordinates

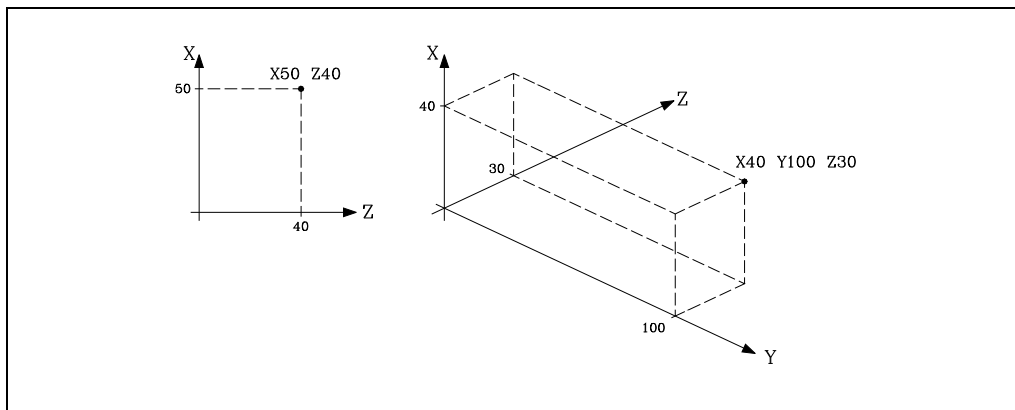
The Cartesian Coordinate System is defined by two axes on the plane, and by three or more axes in space.

The origin of all these, which in the case of the axes X Y Z coincides with the point of intersection, is called Cartesian Origin or Zero Point of the Coordinate System.

The position of the different points of the machine is expressed in terms of the coordinates of the axes, with two, three, four, or five coordinates.

The coordinates of the axes are programmed via the letter of the axis (X, Y, Z, U, V, W, A, B, C, always in this order) followed by the coordinate value.

The values of the coordinates are absolute or incremental, depending on whether it is working in G90 or G91, and its programming format is ± 5.5 .



3.

AXES AND COORDINATE SYSTEMS

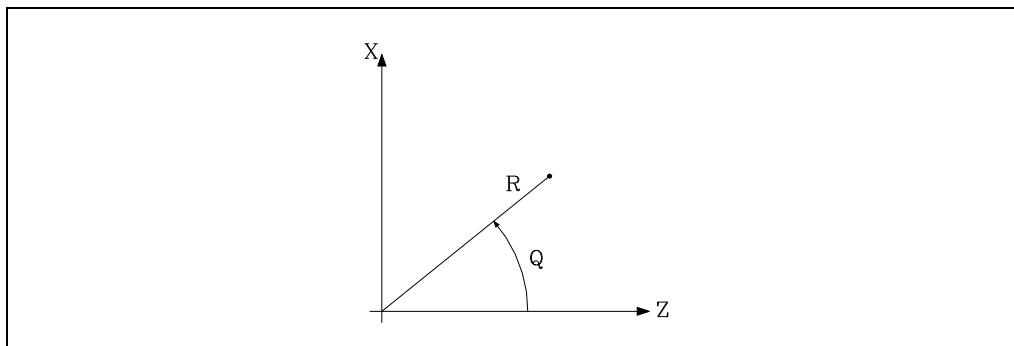
Coordinate programming

3.6.2 Polar coordinates

In the event of the presence of circular elements or angular dimensions, the coordinates of the different points on the plane (2 axes at the same time), it may be easier to express them in polar coordinates.

The reference point is called Polar Origin, and this will be the origin of the Polar Coordinate System.

A point on this system would be defined by :



- The RADIUS (R), the distance between the polar origin and the point.
- The ANGLE (Q), formed by the abscissa axis and the line which joins the polar origin with the point. (In degrees).

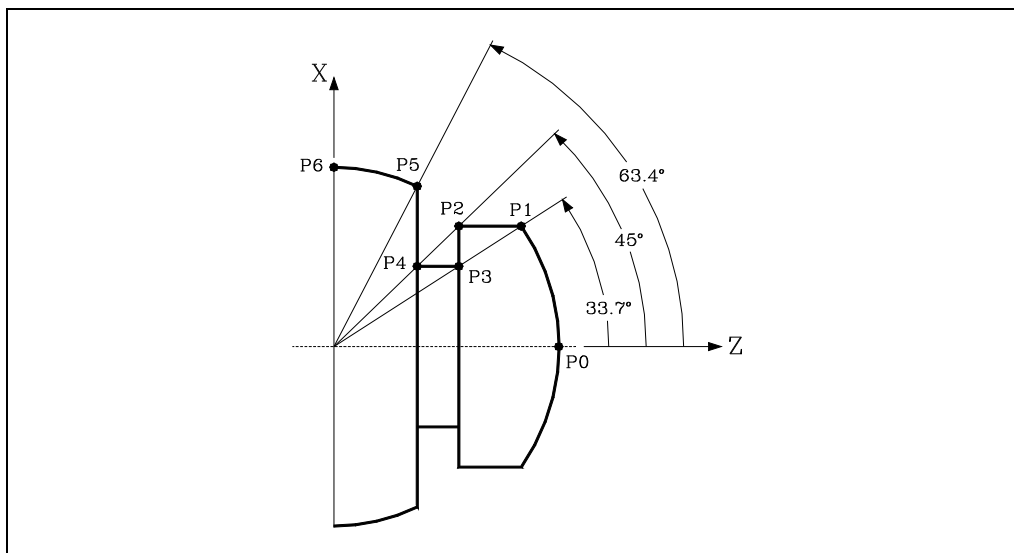
The values R and Q are absolute or incremental depending on whether you are working with G90 or G91, and their programming format will be $R5.5\ Q\pm5.5$. The radius value must always be positive.

The values R and Q are incremental and their programming format will be $R\pm5.5\ Q\pm5.5$.

The R values may be negative when programming in incremental coordinates; but the resulting value assigned to the radius must always be positive.

When programming a "Q" value greater than 360° , the module will be assumed after dividing it by 360. Thus, $Q420$ is the same as $Q60$ and $Q-420$ is the same as $Q-60$.

Programming example assuming that the Polar Origin is located at the Coordinate Origin.



3.

AXES AND COORDINATE SYSTEMS
Coordinate programming

FAGOR 

CNC 8037

· T· MODEL
SOFT: V01.4x

3.

AXES AND COORDINATE SYSTEMS
Coordinate programming

Absolute coordinates:

G90	R430	Q0	; Point P0
G03		Q33.7	; Point P1, in an arc (G03)
G01	R340	Q45	; Point P2, in a straight line (G01)
G01	R290	Q33.7	; Point P3, in a straight line (G01)
G01	R230	Q45	; Point P4, in a straight line (G01)
G01	R360	Q63.4	; Point P5, in a straight line (G01)
G03		Q90	; Point P6, in an arc (G03).

Incremental coordinates:

G90	R430	Q0	; Point 0
G91 G03		Q33.7	; Point P1, in an arc (G03)
G01	R-90	Q11.3	; Point P2, in a straight line (G01)
G01	R-50	Q-11.3	; Point P3, in a straight line (G01)
G01	R-60	Q11.3	; Point P4, in a straight line (G01)
G01	R130	Q18.4	; Point P5, in a straight line (G01)
G03		Q26.6	; Point P6, in an arc (G03).

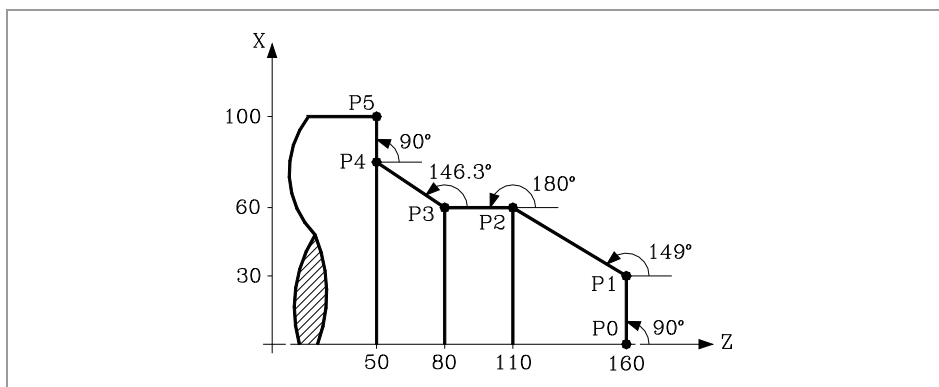
The polar origin, apart from being able to be preset using function G93 (described later) can be modified in the following cases :

- On power-up, after executing M02, M30 EMERGENCY or RESET, the CNC will assume, as the polar origin, the coordinate origin of the work plane defined by the general machine parameter "IPLANE".
- Every time the work plane is changed (G16,G17,G18 or G19), the CNC assumes the coordinate origin of the new work plane selected as the polar origin.
- When executing a circular interpolation (G02 or G03), and if the general machine parameter "PORGMOVE" has a value of 1, the center of the arc will become the new polar origin.

3.6.3 Angle and Cartesian coordinate

A point on the main plane can be defined via one of its Cartesian coordinates, and the exit angle of the previous path.

Example programming the X axis in radius.



X0	Z160	; Point P0
Q90	X30	; Point P1
Q149	Z110	; Point P2
Q180	Z80	; Point P3
Q146.3	Z50	; Point P4
Q90	X100	; Point P5

If you wish to represent a point in space, the remaining coordinates can be programmed in Cartesian coordinates.

3.

AXES AND COORDINATE SYSTEMS
Coordinate programming

FAGOR 

CNC 8037

·T· MODEL
SOFT: V01.4x

3.7 Rotary axes

The types of rotary axes available are:

Normal rotary axis.

Positioning-only rotary axis.

Rotary HIRTH axis.

Each one of them can be divided into:

Rollover When it is displayed between 0° and 360° .

Non Rollover When it may be displayed between -99999° and 99999° .

They are all programmed in degrees. Therefore, their readings are not affected by the inch/mm conversion.

Normal rotary axes

They can interpolate with linear axes.

Movement: In G00 and G01.

Rollover axis programming:

G90 The sign indicates the turning direction and the target position (between 0 and 359.9999).

G91 The sign indicates the turning direction. If the programmed movement exceeds 360° , the axis will rotate more than one turn before positioning at the desired point.

Non-rollover axis programming.

In G90 and G91 like a linear axis.

Positioning-only rotary axis

They cannot be interpolated with linear axes.

Movement: Always in G00 and they do not admit tool radius compensation (G41, G42).

Rollover axis programming:

G90 Always positive and in the shortest direction. End coordinate between 0 and 359.9999.

G91 The sign indicates the turning direction. If the programmed movement exceeds 360° , the axis will rotate more than one turn before positioning at the desired point.

Non-rollover axis programming.

In G90 and G91 like a linear axis.

Rotary Hirth axis

They work like the positioning-only axis except that they do not admit decimal position values (coordinates).

More than one hirth axis can be used, but they can only be moved one at a time.

3.

AXES AND COORDINATE SYSTEMS
Rotary axes



CNC 8037

·T· MODEL
SOFT: V01.4x

3.8 Work zones

The CNC provides four work zones or areas, and also limits the tool movement in each of these.

3.8.1 Definition of the work zones

Within each work zone, the CNC allows you to limit the movement of the tool on each axis, with upper and lower limits being defined in each axis.

G20: Defines the lower limits in the desired zone.

G21: Defines the upper limits in the desired zone.

The format to program these functions is:

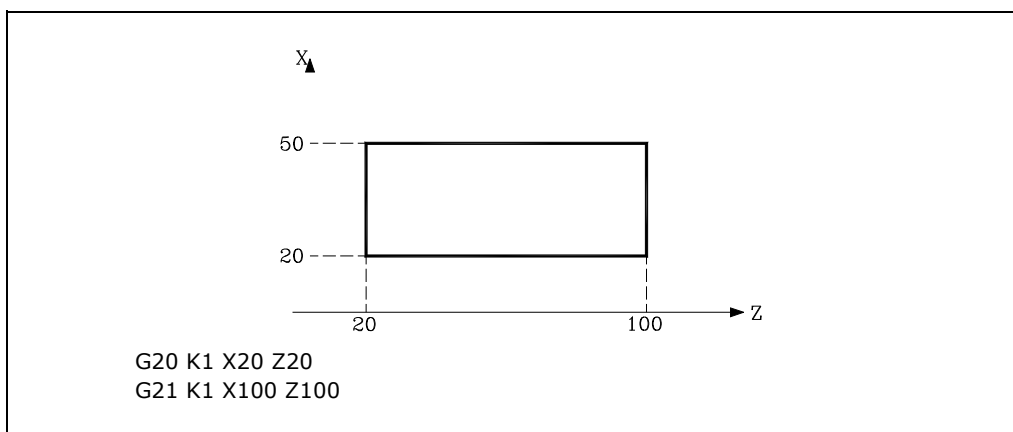
G20 K X...C±5.5

G21 K X...C±5.5

Where:

- | | |
|-------|---|
| K | Indicates the work zone you wish to define (1, 2, 3 or 4). |
| X...C | Indicates the coordinates (upper or lower) with which you wish to limit the axes. These coordinates are given in radius and must be programmed with reference to machine zero (home). For safety, the axis stops 0.1mm before the programmed limit. |

It is not necessary to program all the axes, so only defined axes will be limited.

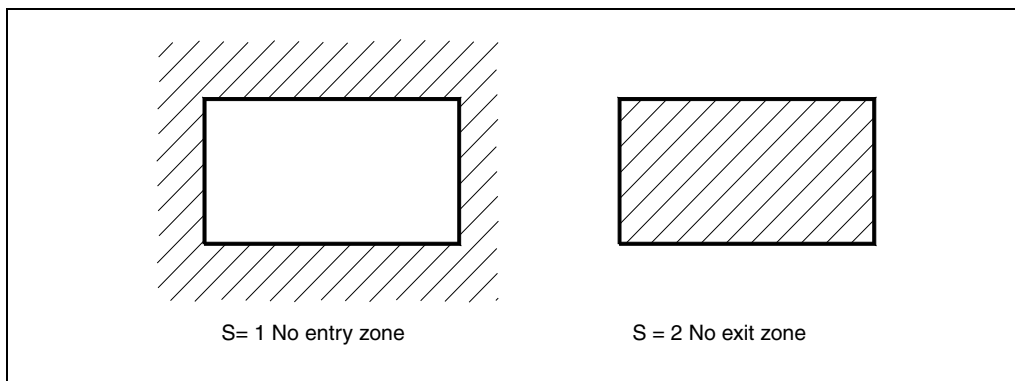


3.

AXES AND COORDINATE SYSTEMS
Work zones

3.8.2 Using the work zones

Within each work zone, the CNC allows you to restrict the movement of the tool, either prohibiting its exit from the programmed zone (no exit zone) or its entry into the programmed zone (no entry zone).



The CNC will take the dimensions of the tool into account at all times (tool offset table) to avoid it exceeding the programmed limits.

The presetting of work zones is done via Function G22, the programming format being:

G22 K S

Where:

K Indicates the work zone you wish to define (1, 2, 3 or 4).

S Indicates the enabling/disabling of the work zone.

S=0 disabled.

S=1 enabled as a no-entry zone.

S=2 enabled as a no-exit zone.

On power-up, the CNC will disable all work zones. However, upper and lower limits for these zones will not undergo any variation, and they can be re-enabled through the G22 function.

3.

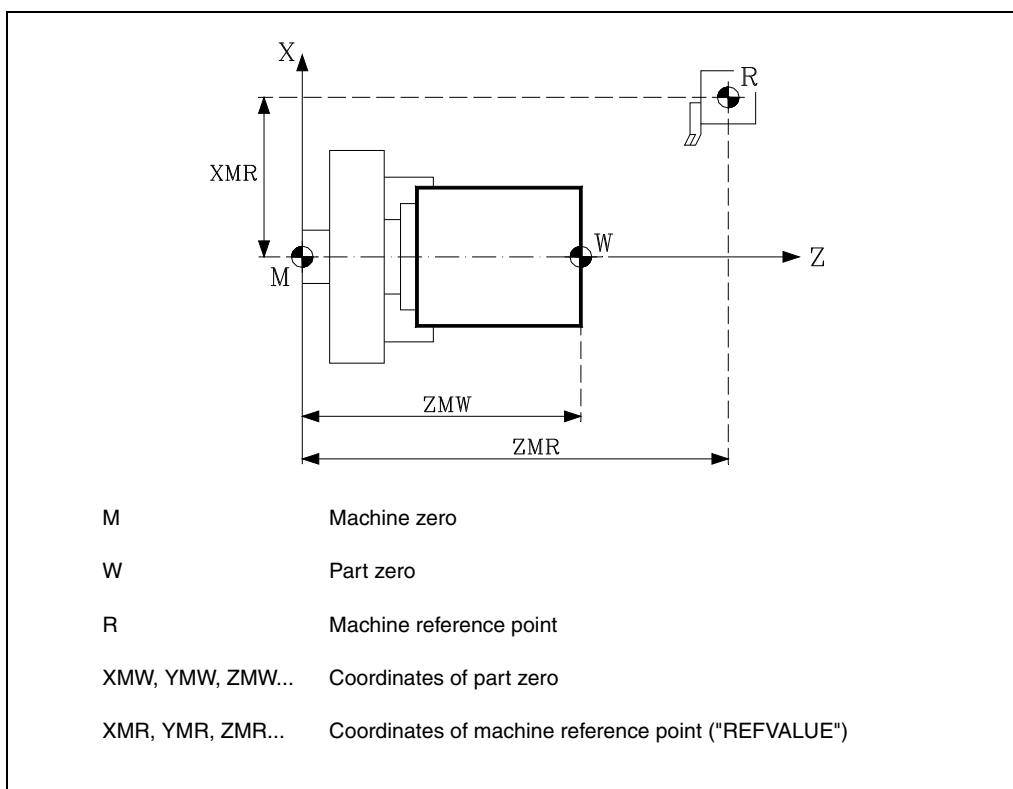
REFERENCE SYSTEMS

4

4.1 Reference points

A CNC machine needs the following origin and reference points defined :

- Machine Reference Zero or home. This is set by the manufacturer as the origin of the coordinate system of the machine.
- Part zero or point of origin of the part. This is the origin point that is set for programming the measurements of the part. It can be freely selected by the programmer, and its machine reference zero can be set by the zero offset.
- Machine Reference point. This is a point on the machine established by the manufacturer around which the synchronization of the system is done. The control positions the axis on this point, instead of moving it as far as the Machine Reference Zero, taking, at this point, the reference coordinates which are defined via the axis machine parameter "REFVALUE".



4.2 Machine reference (Home) search (G74)

The CNC allows you to program the machine reference search in two ways :

- Machine reference (home) search of one or more axes in a particular order.

G74 is programmed followed by the axes in which you want to carry out the reference search. For example: G74 X Z.

The CNC begins the movement of all the selected axes which have a machine reference switch (machine axis parameter "DECINPUT") and in the direction indicated by the axis machine parameter "REFDIREC".

This movement is carried out at the feedrate indicated by the axis machine parameter "REFEED1" for each axis until the home switch is hit.

Next, the home search (marker pulse or home) will be carried out in the programmed order.

This second movement will be carried out one axis at a time, at the feedrate indicated in the axis machine parameter "REFEED2" until the machine reference point is reached (i.e. the marker pulse is found).

- Home search using the associated subroutine.

The G74 function will be programmed alone in the block, and the CNC will automatically execute the subroutine whose number appears in the general machine parameter "REFPSUB". In this subroutine it is possible to program the machine reference searches required, and also in the required order.

In a block in which G74 has been programmed, no other preparatory function may appear.

If the machine reference search is done in JOG mode, the part zero selected is lost. The coordinates of the reference point indicated in the machine axis parameter "REFVALUE" is displayed. In all other cases, the active part zero will be maintained and the CNC will display the position values with respect to that part zero.

If the G74 command is executed in MDI, the display of coordinates depends on the mode in which it is executed : Jog, Execution, or Simulation.

4.

REFERENCE SYSTEMS

Machine reference (Home) search (G74)



CNC 8037

•T• MODEL
SOFT: V01.4x

4.3 Programming with respect to machine zero (G53)

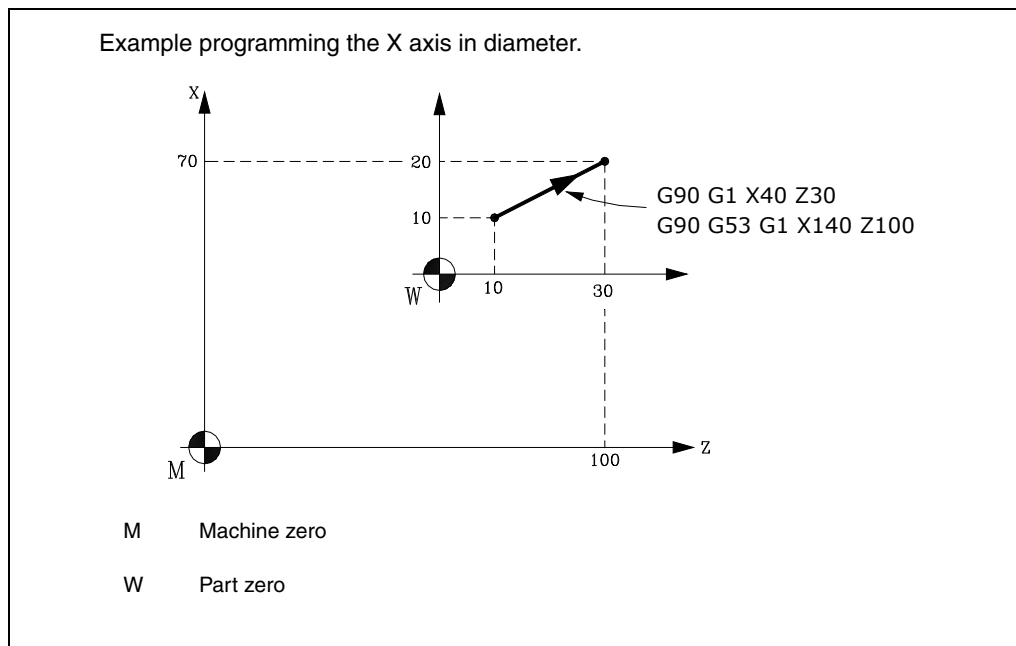
Function G53 can be added to any block that has path control functions.

It is only used when the programming of block coordinates relating to machine zero is required. These coordinates should be expressed in millimeters or inches, depending on how the general machine parameter "INCHES" is defined.

By programming G53 alone (without motion information) the current active zero offset is canceled regardless of whether it was originated by a G54-G59 or a G92 preset. This origin preset is described next.

Function G53 is not modal, so it should be programmed every time you wish to indicate the coordinates referred to machine zero.

This function temporarily cancels radius and tool length compensation.



4.

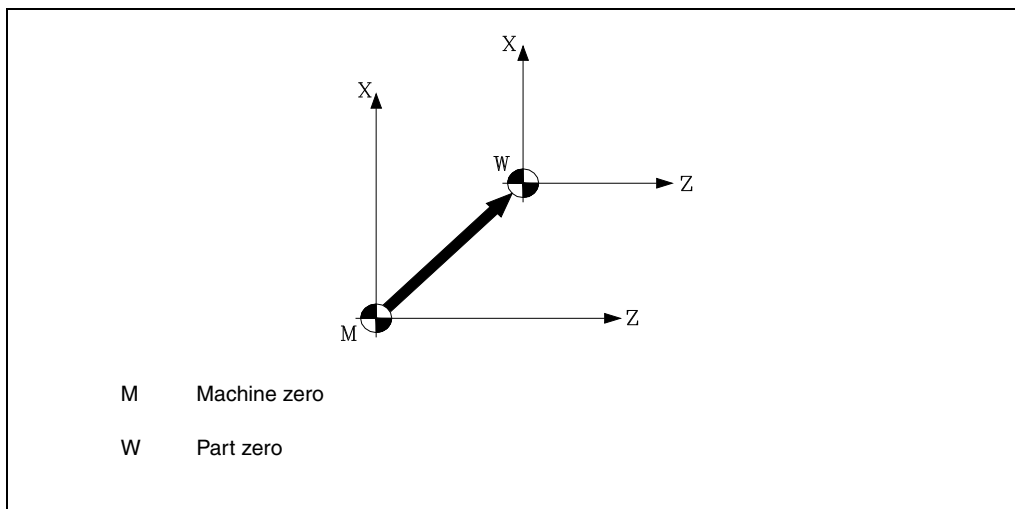
REFERENCE SYSTEMS

Programming with respect to machine zero (G53)

4.4 Coordinate preset and zero offsets

The CNC allows you to carry out zero offsets with the aim of using coordinates related to the plane of the part, without having to modify the coordinates of the different points of the part at the time of programming.

The zero offset is defined as the distance between the part zero (point of origin of the part) and the machine zero (point of origin of the machine).

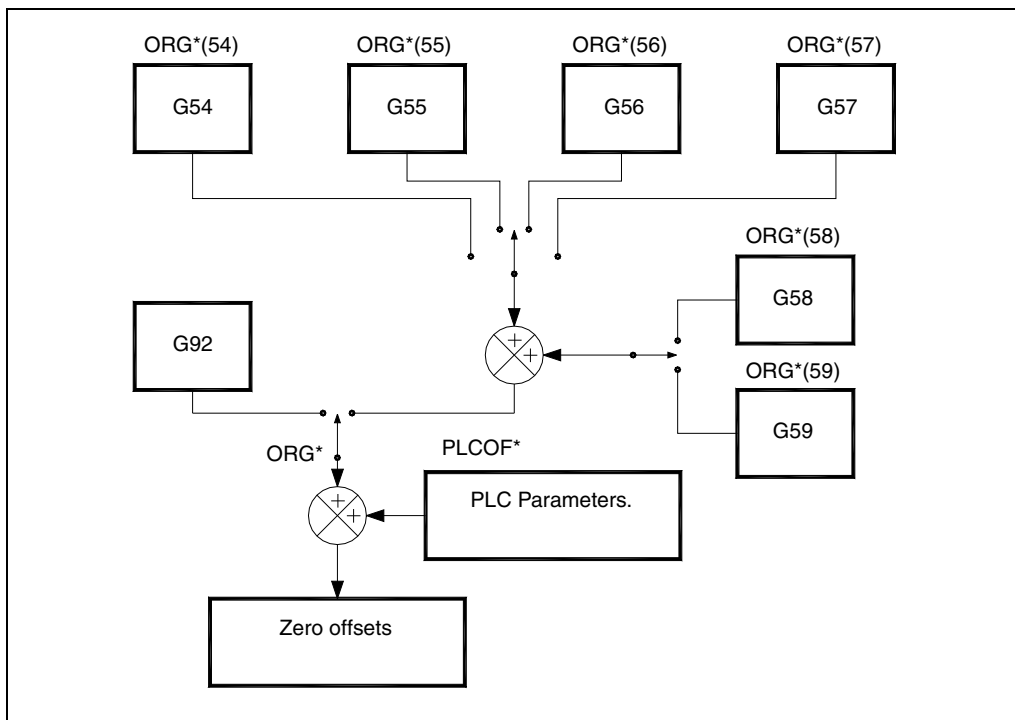


This zero offset can be carried out in one of two ways :

- Via Function G92 (coordinate preset). The CNC accepts the coordinates of the programmed axes after G92 as new axis values.
- Via the use of zero offsets (G54 ... G59, G159N1 ... G159N20); the CNC accepts as a new part zero the point located relative to machine zero at the distance indicated by the selected table(s).

Both functions are modal and incompatible, so if one is selected the other is disabled.

There is, moreover, another zero offset which is governed by the PLC. This offset is always added to the zero offset selected and is used (among other things) to correct deviations produced as a result of expansion, etc.



4.4.1 Coordinate preset and S value limitation (G92)

Via Function G92 one can select any value in the axes of the CNC, in addition to limiting the spindle speed.

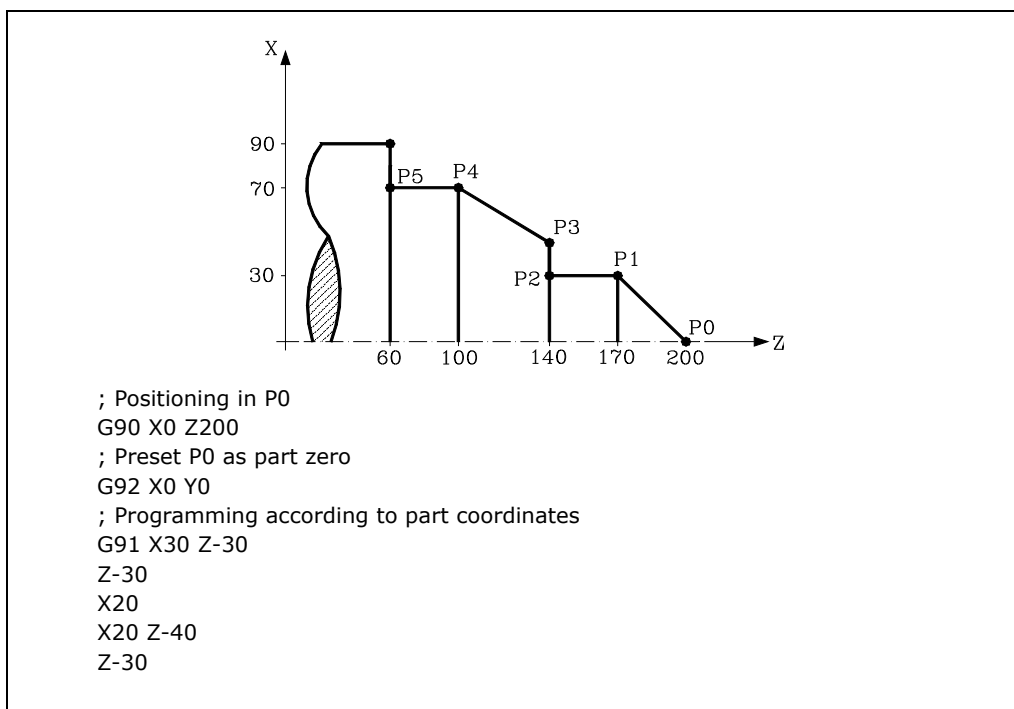
- Coordinate preset.

When carrying out a zero offset via Function G92, the CNC assumes the coordinates of the axes programmed after G92 as new axis values.

No other function can be programmed in the block where G92 is defined, the programming format being :

G92 X...C ± 5.5

The values assigned to the axes are programmed in either radius or diameter depending on the setting of axis machine parameter "DFORMAT".



- Spindle speed limitation

Programming "G92 S5.4", the CNC limits the spindle speed to the value set by S5.4.

This means that the CNC will not accept, from that block on, the programming of S values higher than the maximum defined. Neither is it possible to exceed this maximum value from the keyboard on the front panel.

4.

REFERENCE SYSTEMS

Coordinate preset and zero offsets

4.4.2 Zero offsets (G54..G59 and G159)

The CNC has a table of zero offsets, in which several zero offsets can be selected. The aim is to generate certain part zeros independently of the part zero active at the time. The table values are given in radius.

Access to the table can be obtained from the front panel of the CNC (as explained in the Operating Manual), or via the program using high-level language commands.

There are two kinds of zero offsets :

- Absolute zero offsets (G54 ... G57, G159N1 ... G159N20), which must be referred to machine zero.
- Incremental zero offsets (G58,G59).

Functions G54, G55, G56, G57, G58 & G59 must be programmed alone in the block, and work in the following way:

When one of the G54, G55, G56, G57 functions is executed, the CNC applies the zero offset programmed with respect to machine zero, canceling the possible active zero offsets.

If one of the incremental offsets G58 or G59 is executed, the CNC adds its values to the absolute zero offset active at the time. Previously canceling the additive offset which might be active.

You can see (in the following example) the zero offsets which are applied when the program is executed.

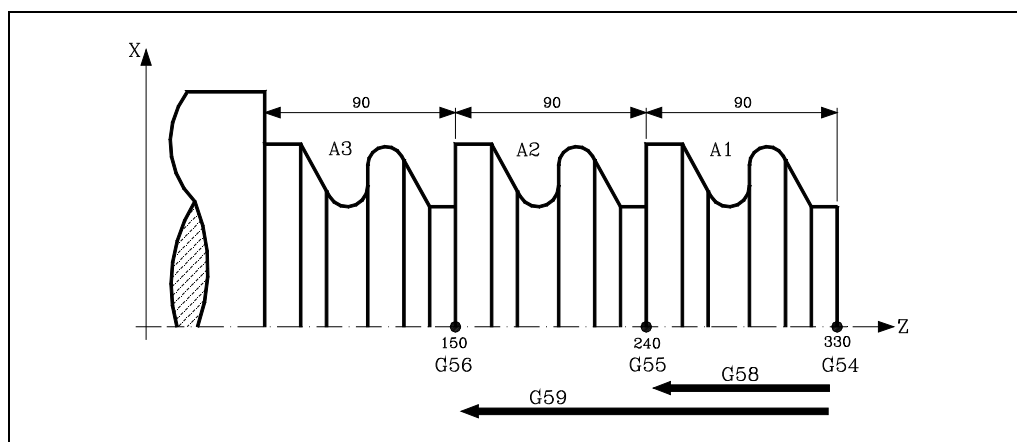
G54	Applies zero offsets G54	==> G54
G58	Adds zero offset G58	==> G54+G58
G59	Cancels G58 and adds G59	==> G54+G59
G55	Cancels whatever and applies G55	==> G55

Once a Zero Offset has been selected, it will remain active until another one is selected or until a home search is carried out (G74) in JOG mode. This zero offset will remain active even after powering the CNC off and back on.

This kind of zero offsets established by program is very useful for repeated machining operations at different machine positions.

Example: The zero offset table is initialized with the following values:

G54:	X0	Z330
G55:	X0	Z240
G56:	X0	Z150
G58:	X0	Z-900
G59:	X0	Z-180



Using absolute zero offsets:

G54	; Applies G54 offset
Profile execution	; Executes profile A1
G55	; Applies G55 offset
Profile execution	; Executes profile A2
G56	; Applies G56 offset
Profile execution	; Executes profile A3

Using incremental zero offsets:

G54	; Applies G54 offset
Profile execution	; Executes profile A1
G58	; Applies offsets G54+G58
Profile execution	; Executes profile A2
G59	; Applies offsets G54+G59
Profile execution	; Executes profile A3

Function G159

To apply any zero offset defined in the table.

The first six zero offsets are the same as programming G54 through G59, except that the values of G58 and G59 are absolute. This is because function G159 cancels functions G54 through G57 and, consequently, there is no active zero offset to add the G58 or G59 to.

Function G159 is programmed as follows:

G159 Nn Where n is a number from 1 to 20 that indicates the number of the zero offset being applied.

Function G159 is modal, it is programmed alone in the block and is incompatible with functions G53, G54, G55, G56, G57, G58, G59 and G92.

On power-up, the CNC assumes the zero offset that was active when the CNC was turned off. On the other hand, the zero offset is neither affected by functions M02 and M30 nor by RESET.

This function is displayed in the history like G159Nn where the n is the active zero offset.

Examples:

G159 N1	It applies the first zero offset. It is the same as programming G54.
G159 N6	It applies the second zero offset. It is the same as programming G59, but it is applied in absolute.
G159 N20	It applies the 20th zero offset.

4.

REFERENCE SYSTEMS

Coordinate preset and zero offsets

4.5 Polar origin preset (G93)

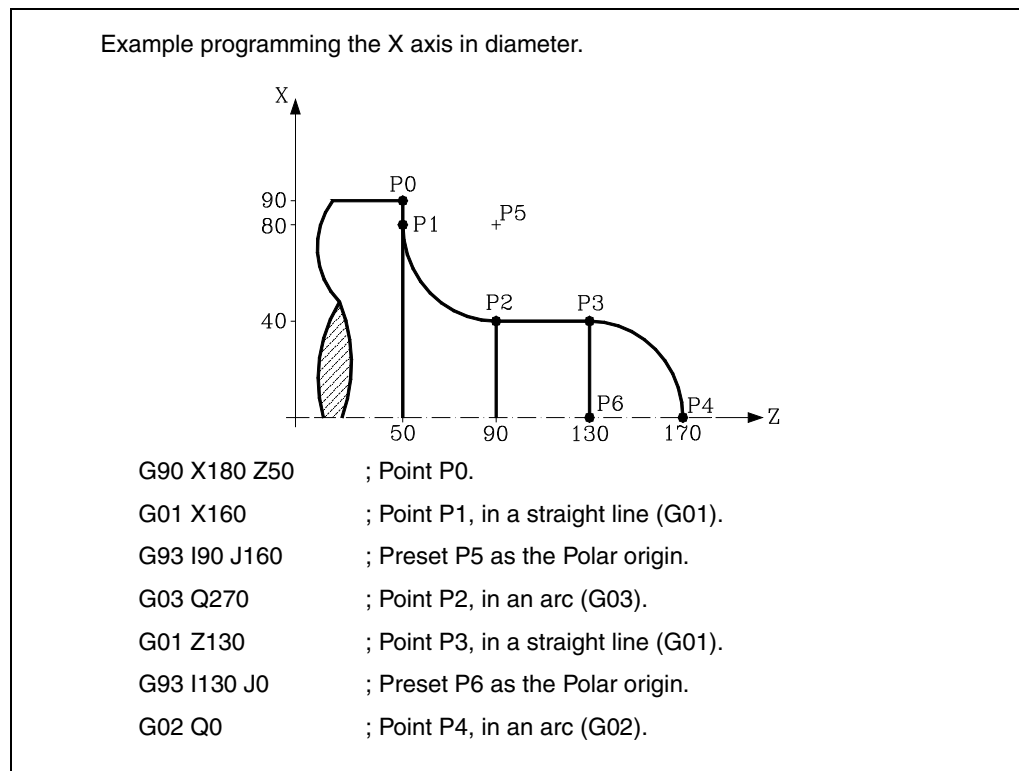
Function G93 allows you to preset any point from the work plane as a new origin of polar coordinates.

This function must be programmed alone in the block, its programming format being :

G93 I±5.5 J±5.5

Parameters I & J respectively define the abscissa and ordinate axes, of the new origin of polar coordinates referred to part zero.

The values assigned to those parameters are programmed in either radius or diameter depending on the setting of axis machine parameter "DFORMAT".



If G93 is only programmed in a block, the point where the machine is at that moment becomes the polar origin.

On power-up; or after executing M02, M30; or after an EMERGENCY or RESET; the CNC assumes the currently active part zero as polar origin.

When selecting a new work plane (G16, G17, G18, G19), the CNC assumes as polar origin the part zero of that plane.



The CNC does not modify the polar origin when defining a new part zero; but it modifies the values of the variables: "PORGF" y "PORGS".

If, while selecting the general machine parameter "PORGMOVE" a circular interpolation is programmed (G02 or G03), the CNC assumes the center of the arc as the new polar origin.

A block programmed in ISO language can consist of:

- Preparatory (G) functions
- Axis coordinates (X...C)
- Feedrate (F)
- Spindle speed (S)
- Tool number (T)
- Tool offset number (D)
- Auxiliary functions (M)

This order should be maintained within each block, although it is not necessary for every block to contain the information.

The CNC allows you to program figures from 0.00001 to 99999.9999 with or without sign, working in millimeters (G71), called format +/-5.4, or either from 0.00001 to 3937.00787 with or without sign if the programming is done in inches (G70), called format +/-4.5.

However, and to simplify the instructions, we can say that the CNC admits +/- 5.5 format, thereby admitting +/- 5.4 in millimeters and +/- 4.5 in inches.

Any function with parameters can also be programmed in a block, apart from the number of the label or block. Thus, when the block is executed the CNC substitutes the arithmetic parameter for its value at that time.

5.1 Preparatory functions

Preparatory functions are programmed using the letter G followed by up to 3 digits (G0 - G319).

They are always programmed at the beginning of the body of the block and are useful in determining the geometry and working condition of the CNC.

Table of G functions used in the CNC.

Function	M	D	V	Meaning	Section
G00	*	?	*	Rapid traverse	6.1
G01	*	?	*	Linear interpolation	6.2
G02	*		*	Clockwise circular interpolation	6.3
G03	*		*	Counterclockwise circular interpolation	6.3
G04				Dwell/interruption of block preparation	7.1 / 7.2
G05	*	?	*	Round corner	7.3.2
G06			*	Circle center in absolute coordinates	6.4
G07	*	?		Square corner	7.3.1
G08			*	Arc tangent to previous path.	6.5
G09			*	Arc defined by three points	6.6
G10	*	*		Mirror image cancellation	7.5
G11	*		*	Mirror image on X axis	7.5
G12	*		*	Mirror image on Y axis	7.5
G13	*		*	Mirror image on Z axis	7.5
G14	*		*	Mirror image in the programmed directions	7.5
G16	*		*	Main plane selection by two addresses	3.2
G17	*	?	*	Main plane X-Y and longitudinal Z	3.2
G18	*	?	*	Main plane Z-X and longitudinal Y	3.2
G19	*		*	Main plane Y-Z and longitudinal X	3.2
G20				Definition of lower work zone limits	3.8.1
G21				Definition of upper work zone limits.	3.8.1
G22			*	Enable/disable work zones.	3.8.2
G32	*		*	Feedrate "F" as an inverted function of time.	6.15
G33	*		*	Electronic threading	6.11
G34				Variable-pitch threading	6.13
G36			*	Controlled corner rounding	6.9
G37			*	Tangential entry	6.7
G38			*	Tangential exit	6.8
G39			*	Chamfer	6.10
G40	*	*		Cancellation of tool radius compensation	8.2.6
G41	*		*	Left-hand tool radius compensation	8.2.3
G41 N	*		*	Collision detection	8.3
G42	*		*	Right-hand tool radius compensation	8.2.3
G42 N	*		*	Collision detection	8.3
G50	*		*	Controlled corner rounding	7.3.3
G51	*		*	Look-Ahead	7.4
G52			*	Movement until making contact	6.14
G53			*	Programming with respect to machine zero	4.3
G54	*		*	Absolute zero offset 1	4.4.2
G55	*		*	Absolute zero offset 2	4.4.2
G56	*		*	Absolute zero offset 3	4.4.2
G57	*		*	Absolute zero offset 4	4.4.2
G58	*		*	Additive zero offset 1	4.4.2
G59	*		*	Additive zero offset 2	4.4.2
G66			*	Pattern repeat cycle canned cycle	9.1
G68			*	X axis roughing canned cycle	9.2
G69			*	Z axis roughing canned cycle	9.3

5.

ISO CODE PROGRAMMING
Preparatory functions



CNC 8037

.T- MODEL
SOFT: V01.4x

Function	M	D	V	Meaning	Section
G70	*	?	*	Programming in inches	3.3
G71	*	?		Programming in millimeters	3.3
G72	*		*	General and specific scaling factor	7.6
G74			*	Home search	4.2
G75			*	Probing move until touching	10.1
G76			*	Probing move while touching	10.1
G81			*	Turning canned cycle for straight sections	9.4
G82			*	Facing canned cycle for straight sections	9.5
G83			*	Drilling canned cycle	9.6
G84			*	Turning canned cycle for curved sections	9.7
G85			*	Facing canned cycle for curved sections	9.8
G86			*	Longitudinal threading canned cycle	9.9
G87			*	Face threading canned cycle	9.10
G88			*	X axis grooving canned cycle	9.11
G89			*	Z axis grooving canned cycle	9.12
G90	*	?		Absolute programming:	3.4
G91	*	?	*	Incremental programming	3.4
G92				Coordinate preset / spindle speed limit	4.4.1
G93				Polar origin preset	4.5
G94	*	?		Feedrate in millimeters (inches) per minute	5.2.1
G95	*	?	*	Feedrate in millimeters (inches) per revolution.	5.2.2
G96	*		*	Constant surface speed	5.3.1
G97	*		*	Spindle speed in RPM	5.3.2
G151	*	?		Programming the X axis in diameter.	3.5
G152	*	?		Programming the X axis in radius.	3.5
G159	*			Absolute zero offsets	4.4.2
G233	*			Withdrawal of axes when interrupting a threading operation.	6.12

M means modal, i.e. the G function, once programmed, remains active until another incompatible G function is programmed or until an M02, M30, EMERGENCY or RESET is executed or the CNC is turned off and back on.

D means BY DEFAULT, i.e. they will be assumed by the CNC when it is powered on, after executing M02, M30 or after EMERGENCY or RESET.

In those cases indicated by ?, it should be understood that the DEFAULT of these G functions depends on the setting of the general machine parameters of the CNC.

The letter V means that the G code is displayed next to the current machining conditions in the execution and simulation modes.

5.

ISO CODE PROGRAMMING

Preparatory functions



CNC 8037

·T· MODEL
SOFT: V01.4x

5.2 Feedrate F

The machining feedrate can be selected from the program. It remains active until another feedrate is programmed. It is represented by the letter F and Depending on whether it is working in G94 or G95, it is programmed in mm/minute (inches/minute) or in mm/revolution (inches/revolution).

Its programming format is 5.5; in other words, 5.4 when programmed in mm and 4.5 when programmed in inches.

The maximum operating feedrate of the machine, limited on each axis by the axis machine parameter "MAXFEED", may be programmed via code F0, or by giving F the corresponding value.

The programmed feedrate F is effective working in linear (G01) or circular (G02, G03) interpolation. If function F is not programmed, the CNC assumes the feedrate to be F0. When working in rapid travel (G00), the machine will move at the rapid feedrate indicated by the axis machine parameter "G00FEED", apart from the F programmed.

The programmed feedrate F may be varied between 0% and 255% via the PLC, or by DNC, or between 0% and 120% via the switch located on the Operator Panel of the CNC.

The CNC, however, is equipped with the general machine parameter "MAXFOVR" to limit maximum feedrate variation.

If you are working in rapid travel (G00), rapid feedrate will be fixed at 100%, alternatively it can be varied between 0% and 100%, depending on how the machine parameter "RAPIDOVR" is set.

When functions G33 (electronic threading), G34 (variable-pitch threading), G86 (longitudinal threading canned cycle) or G87 (face threading canned cycle) are executed the feedrate cannot be modified; it works at 100% of programmed F.

5.**ISO CODE PROGRAMMING**

Feedrate F

**CNC 8037**.T. MODEL
SOFT: V01.4x

5.2.1 Feedrate in mm/min or inches/min (G94)

From the moment the code G94 is programmed, the control takes that the feedrates programmed through F5.5 are in mm/min or inches/mm.

If the moving axis is rotary, the CNC interprets that the programmed feedrate is in degrees/minute.

If an interpolation is made between a rotary and a linear axis, the programmed feedrate is taken in mm/min or inches/min, and the movement of the rotary axis (programmed in degrees) will be considered programmed in millimeters or inches.

The relationship between the feedrate of the axis component and the programmed feedrate "F" is the same as that between the movement of the axis and the resulting programmed movement.

$$\text{Feedrate component} = \frac{\text{Feedrate F} \times \text{Movement of axis}}{\text{Resulting programmed movement}}$$

Example:

On a machine that has linear X and Z axes and rotary C axis, all located at point X0 Z0 C0, the following movement is programmed:

G1 G90 X100 Z20 C270 F10000

You get:

$$F_x = \frac{F \cdot \Delta x}{\sqrt{(\Delta x)^2 + (\Delta z)^2 + (\Delta c)^2}} = \frac{10000 \times 100}{\sqrt{100^2 + 20^2 + 270^2}} = 3464,7946$$

$$F_z = \frac{F \cdot \Delta z}{\sqrt{(\Delta x)^2 + (\Delta z)^2 + (\Delta c)^2}} = \frac{10000 \times 20}{\sqrt{100^2 + 20^2 + 270^2}} = 692,9589$$

$$F_c = \frac{F \cdot \Delta c}{\sqrt{(\Delta x)^2 + (\Delta z)^2 + (\Delta c)^2}} = \frac{10000 \times 270}{\sqrt{100^2 + 20^2 + 270^2}} = 9354,9455$$

Function G94 is modal i.e. once programmed it stays active until G95 is programmed.

On power-up, after executing M02, M30 or following EMERGENCY or RESET, the CNC assumes function G94 or G95 according to how the general machine parameter "IFEED" is set.

5.

ISO CODE PROGRAMMING

Feedrate F

FAGOR 

CNC 8037

· T· MODEL
SOFT: V01.4x

5.2.2 Feedrate in mm/rev.or inches/rev (G95)

From the moment when the code G95 is programmed, the control assumes that the feedrates programmed through F5.5 are in mm/rev or inches/mm.

This function does not affect the rapid moves (G00) which will be made in mm/min or inch/min. By the same token, it will not be applied to moves made in the JOG mode, during tool inspection, etc.

Function G95 is modal i.e. once programmed it stays active until G94 is programmed.

On power-up, after executing M02, M30 or following EMERGENCY or RESET, the CNC assumes function G94 or G95 according to how the general machine parameter "IFEED" is set.

Jogging in G95

If the axis to be jogged does not belong to the active plane, the movement is carried out in mm/minute; thus, it is not necessary to program an S at the spindle.

Also, if any axis of the plane is the Y axis, it is not necessary either to program the S for jog movements in any axis, regardless of whether it belongs to the plane or not.

This is especially interesting for auxiliary axes, center rests and tailstocks, because, in those cases, the S has no effect.

5.

ISO CODE PROGRAMMING

Feedrate F



CNC 8037

·T· MODEL
SOFT: V01.4x

5.3 Spindle speed (S)

The turning speed of the spindle is programmed directly in rpm via code S5.4 (G97) or in m/min (ft/min when working in inches) when in constant surface speed mode (G96).

The maximum value is limited by spindle machine parameters "MAXGEAR1", MAXGEAR2, MAXGEAR 3 and MAXGEAR4", in each case depending on the spindle range selected.

It is also possible to limit this maximum value from the program by using function G92 S5.4.

The programmed turning speed S may be varied from the PLC, DNC, or by the SPINDLE keys "+" and "-" on the Operator Panel of the CNC.

This speed variation is made between the maximum and minimum values established by spindle machine parameters "MINSOVR" and "MAXSOVR".

The incremental pitch associated with the SPINDLE keys "+" and "-" on the CNC Operator Panel in order to vary the programmed S value is fixed by the spindle machine parameter "SOVRSTEP".

When functions G33 (electronic threading), G34 (variable-pitch threading), G86 (longitudinal threading canned cycle) or G87 (face threading canned cycle) are executed the programmed speed cannot be modified; it works at 100% of programmed S.

5.**ISO CODE PROGRAMMING**

Spindle speed (S)

5.3.1 Constant surface speed (G96)

When programming G96, the CNC assumes that the spindle speed programmed with S5.4 is in m/min or ft/min and the lathe starts working in constant surface speed mode.

It is recommended to program the spindle speed S5.4 in the same block where G96 is programmed; the corresponding spindle gear (M41, M42, M43, M44) must be selected in the same block or in a previous one.

If no spindle speed is programmed in the block containing the G96, the CNC assumes as spindle speed the last one used in constant surface speed mode.

If no spindle speed is programmed and there is no previous one or the corresponding spindle gear is not selected, the CNC will show the corresponding error message.

Function G96 is modal i.e. once programmed it stays active until G97 is programmed.

On power-up, after executing M02, M30 or following EMERGENCY or RESET, the CNC assumes function G97.

5.**ISO CODE PROGRAMMING**

Spindle speed (S)

**CNC 8037**·T· MODEL
SOFT: V01.4x

5.3.2 Spindle speed in rpm (G97)

When programming G97, the CNC assumes that the spindle speed programmed with S5.4 is in rpm.

If no spindle speed S5.4 is programmed in the block containing the G97, the CNC assumes as programmed speed at which the spindle was turning at the time.

Function G97 is modal i.e. once programmed it stays active until G96 is programmed.

On power-up, after executing M02, M30 or following EMERGENCY or RESET, the CNC assumes function G97.

5.

ISO CODE PROGRAMMING

Spindle speed (S)

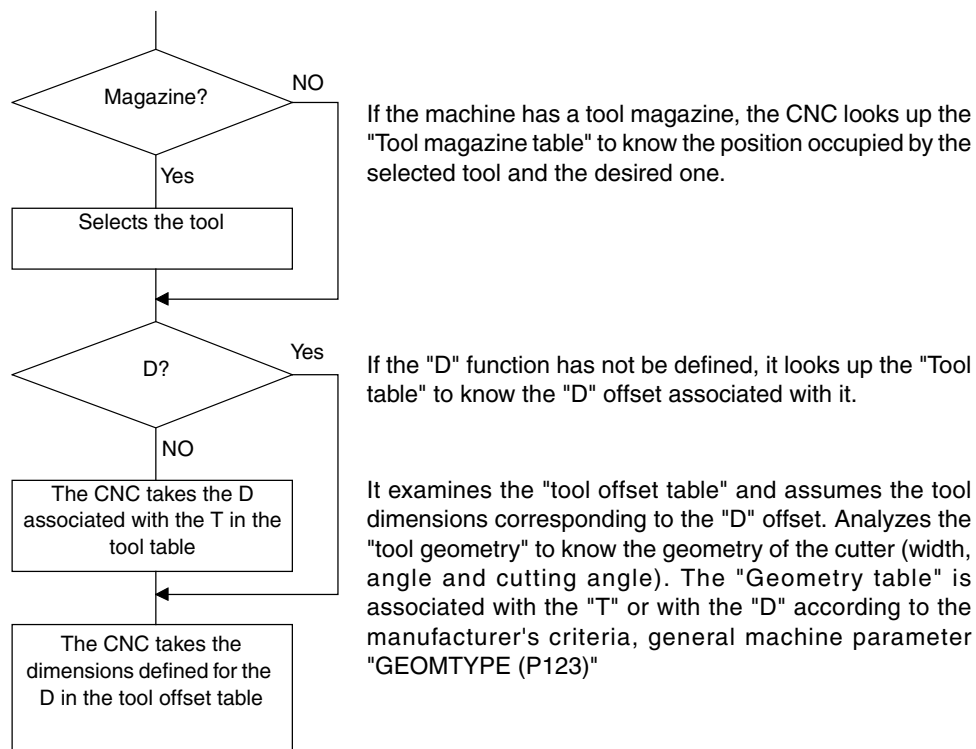


CNC 8037

·T· MODEL
SOFT: V01.4x

5.4 Tool number (T) and tool offset (D)

With the "T" function, it is possible to select the tool and with the "D" function it is possible to select the offset associated with it. When defining both parameters, the programming order is T D. For example: T6 D17



To access, check and define these tables, refer to the operating manual.

How to use the T and D functions

- The "T" and "D" functions may be programmed alone or together as shown in the following example:

T5 D18 Selects tool 5 and assumes the dimensions of tool offset 18

D22 Tool 5 stays selected and it assumes the dimensions of tool offset 22.

T3 Selects tool 3 and assumes the dimensions of the offset associated with that tool.

- When using a tool holding turret, more tools are normally used than the available turret positions. Thus, the same turret position must be used by several tools.

In these cases, the "T" and "D" functions must be programmed.

The "T" function refers to the turret position and the "D" function to the dimensions of the tool located in that position.

Thus, for example, programming T5 D23 means selecting the turret position 5 and assuming the geometry and dimensions of tool offset 23.

- When having a tool holding arm with two cutters, the "T" and "D" functions must also be programmed.

The "T" function refers to the arm and the "D" function to the dimensions of the cutter. Thus, one may program T1 D1 or T1 D2 depending on which of the two cutters are to be used.

Tool length and radius compensation.

The CNC looks up the "tool offset table" and assumes the tool dimensions corresponding to the active "D" offset.

Tool length compensation is applied all the time, whereas tool radius compensation must be selected by the user by means of functions G40, G41 and G42.

If there is no tool selected or D0 is defined, neither tool length nor radius compensation is applied.

For further detail refer to See chapter ["8 Tool Compensation"](#).

5.

ISO CODE PROGRAMMING

Tool number (T) and tool offset (D)



CNC 8037

·T· MODEL
SOFT: V01.4x

5.5 Auxiliary function (M)

The miscellaneous functions are programmed by means of the M4 Code, it being possible to program up to 7 functions in the same block.

When more than one function has been programmed in one block, the CNC executes these correlatively to the order in which they have been programmed.

The CNC is provided with an M functions table with "NMISCFUN" (general machine parameter) components, specifying for each element:

- The number (0-9999) of the defined miscellaneous M function.
- The number of the subroutine which is required to associate to this miscellaneous function.
- An indicator which determines if the M function is executed before or after the movement block in which it is programmed.
- An indicator which determines if the execution of the M function interrupts block preparation or not.
- An indicator which determines if the M function is executed or not, after the execution of the associated subroutine.
- An indicator which determines if the CNC must wait for the signal AUX END or not (Executed M signal, coming from the PLC), to continue the execution of the program.

If, when executing the M miscellaneous function, this is not defined in the M functions table, the programmed function will be executed at the beginning of the block and the CNC will wait for the AUX END to continue the execution of the program.

Some of the miscellaneous functions are assigned an internal meaning in the CNC.

If, while executing the associated subroutine of an "M" miscellaneous function, there is a block containing the same "M", this will be executed but not the associated subroutine.



All the miscellaneous "M" functions which have an associated subroutine must be programmed alone in a block.

In the case of functions M41 through M44 with associated subroutine, the S that generates the gear change must be programmed alone in the block. Otherwise, the CNC will display error 1031.

5.5.1 M00. Program stop

When the CNC reads code M00 in a block, it interrupts the program. To start up again, press CYCLE START.

We recommend that you set this function in the table of M functions, in such a way that it is executed at the end of the block in which it is programmed.

5.5.2 M01. Conditional program stop

This is identical to M00, except that the CNC only takes notice of it if the signal M01 STOP from the PLC is active (high logic level).

5.

ISO CODE PROGRAMMING
Auxiliary function (M)

5.5.3 M02. End of program

This code indicates the end of program and carries out a "General Reset" function of the CNC (returning it to original state). It also carries out the M05 function.

We recommend that you set this function in the table of M functions, in such a way that it is executed at the end of the block in which it is programmed.

5.

ISO CODE PROGRAMMING
Auxiliary function (M)

5.5.4 M30. End of program with return to the first block

Identical to M02 except that the CNC returns to the first block of the program.



CNC 8037

·T· MODEL
SOFT: V01.4x

5.5.5 M03 Clockwise spindle rotation

This code represents clockwise spindle start.

We recommend that you set this function in the table of M functions, so that it is executed at the beginning of the block in which it is programmed.

5.5.6 M04. Counterclockwise spindle rotation

This code represents counterclockwise spindle start. We recommend that you set this function in the table of M functions, so that it is executed at the beginning of the block in which it is programmed.

5.

ISO CODE PROGRAMMING
Auxiliary function (M)

5.5.7 M05. Spindle stop

We recommend that you set this function in the table of M functions, in such a way that it is executed at the end of the block in which it is programmed.

5.

ISO CODE PROGRAMMING
Auxiliary function (M)

5.5.8 M06. Tool change code

If the general machine parameter "TOFFM06" (indicating that it is a machining center) is active, the CNC sends instructions to the tool changer and updates the table corresponding to the tool magazine.

It is recommended to set this function in the table of M functions, so that the subroutine corresponding to the tool changer installed in the machine is executed.



CNC 8037

·T· MODEL
SOFT: V01.4x

5.5.9 M19. Spindle orientation

With this CNC it is possible to work with the spindle in open loop (M3, M4) and with the spindle in closed loop (M19).

In order to work in closed loop, it is necessary to have a rotary encoder installed on the spindle of the machine.

To switch from open loop to closed loop, execute function M19 or M19 S \pm 5.5. The CNC will act as follows:

- If the spindle has a home switch, the CNC modifies the spindle speed until it reaches the one set by spindle machine parameter "REFEED1".

It then searches for actual marker pulse (I0) of the spindle encoder at the turning speed set by spindle machine parameter REFEED2.

And, finally, it positions the spindle at the programmed S \pm 5.5 point.

- If the spindle does not have a home switch, it searches the encoder marker pulse at the turning speed set by spindle machine parameter REFEED2.

And, then, it positions the spindle at the programmed S \pm 5.5 point.

If only M19 is executed, the spindle is oriented after having "found" the home switch (I0).

To, now, orient the spindle to another position, program M19 S \pm 5.5, the CNC will not perform the home search since it is already in closed loop and it will orient the spindle to the indicated position. (S \pm 5.5).

The S \pm 5.5 code indicates the spindle position, in degrees, from the spindle reference point (marker pulse).

The sign indicates the counting direction and the 5.5 value is always considered to be absolute coordinates regardless of the type of units currently selected.

Example:

S1000 M3

Spindle in open loop.

M19 S100

The spindle switches to closed loop. Home search and positioning (orientation) at 100°.

M19 S -30

The spindle orients to -30°, passing through 0°.

M19 S400

The spindle turns a whole revolution and positions at 40°.



During the M19 process the screen will display the warning: "M19 in execution"

5.

ISO CODE PROGRAMMING

Auxiliary function (M)

FAGOR

CNC 8037

· T· MODEL
SOFT: V01.4x

5.5.10 M41, M42, M43, M44. Spindle gear change

The CNC offers 4 spindle speed ranges M41, M42, M43 and M44 with maximum speed limits set by the spindle machine parameters "MAXGEAR1", MAXGEAR2", "MAXGEAR3" and "MAXGEAR4".

If machine parameter "AUTOGEAR" is set so the CNC executes the range change automatically, M41 thru M44 will be sent out automatically by the CNC without having to be programmed.

If this machine parameter is set for non-automatic gear change, M41 thru M44 will have to be programmed every time a gear change is required. Bear in mind that the maximum velocity command value assigned to machine parameter "MAXVOLT" corresponds to the maximum speed indicated for each one of the speed ranges (machine parameters "MAXGEAR1" thru "MAXGEAR4").

Regardless of whether the gear change is automatic or not, functions M41 through M44 may have an associated subroutine. If the function M41 through M44 is programmed and then an S corresponding to that gear, it does not generate the automatic gear change and it does not execute the associated subroutine.

5.

ISO CODE PROGRAMMING
Auxiliary function (M)



CNC 8037

·T· MODEL
SOFT: V01.4x

The CNC allows you to program movements on one axis only or several at the same time.

Only those axes which intervene in the required movement are programmed. The programming order of the axes is as follows :

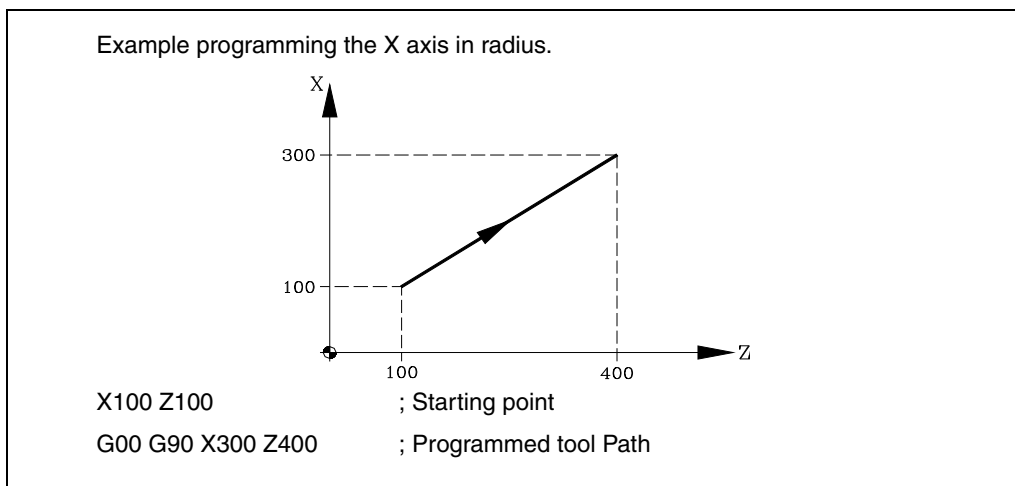
X, Y, Z, U, V, W, A, B, C

The coordinates of each axis are programmed in either radius or diameter depending on the setting of axis machine parameter "DFORMAT".

6.1 Rapid traverse (G00)

The movements programmed after G00 are executed at the rapid feedrate indicated in the axis machine parameter "G00FEED".

Independently of the number of axis which move, the resulting path is always a straight line between the starting point and the final point.



It is possible, via the general machine parameter "RAPIDOVR", to establish if the feedrate override % switch (when working in G00) operates from 0% to 100%, or whether it stays constant at 100%.

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

G00 is modal and incompatible with G01, G02, G03, G33 G34 and G75. Function G00 can be programmed as G or G0.

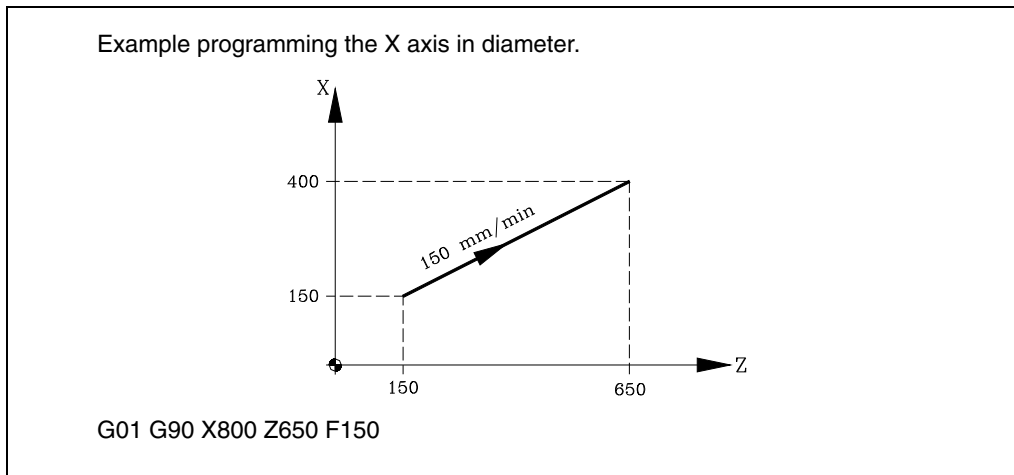
On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G00 or G01, depending on how general machine parameter "IMOVE" has been set.

6.2 Linear interpolation (G01)

The movements programmed after G01 are executed according to a straight line and at the programmed feedrate "F".

When two or three axes move simultaneously the resulting path is a straight line between the starting point and the final point.

The machine moves according to this path to the programmed feedrate "F". The CNC calculates the feedrates of each axis so that the resulting path is the "F" value programmed.



The programmed feedrate "F" may vary between 0% and 120% via the switch located on the Control Panel of the CNC, or by selecting between 0% and 255% from the PLC, or via the DNC or the program.

The CNC, however, is equipped with the general machine parameter "MAXFOVR" to limit maximum feedrate variation.

With this CNC, it is possible to program a positioning-only axis in a linear interpolation block. The CNC will calculate the feedrate for this positioning-only axis so it reaches the target coordinate at the same time as the interpolating axes.

Function G01 is modal and incompatible with G00, G02, G03, G33 and G34. Function G01 can be programmed as G1.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G00 or G01, depending on how general machine parameter "IMOVE" has been set.

6.

PATH CONTROL
Linear interpolation (G01)

6.3 Circular interpolation (G02, G03)

There are two ways of carrying out circular interpolation:

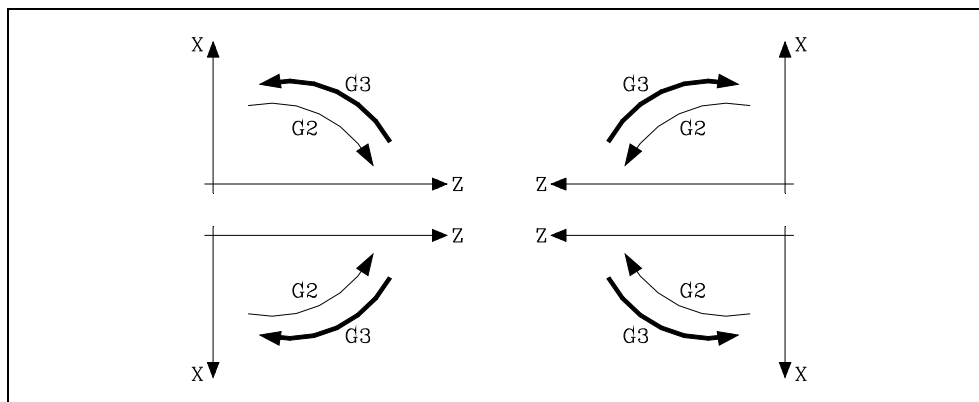
G02: Clockwise circular interpolation

G03: Counterclockwise circular interpolation

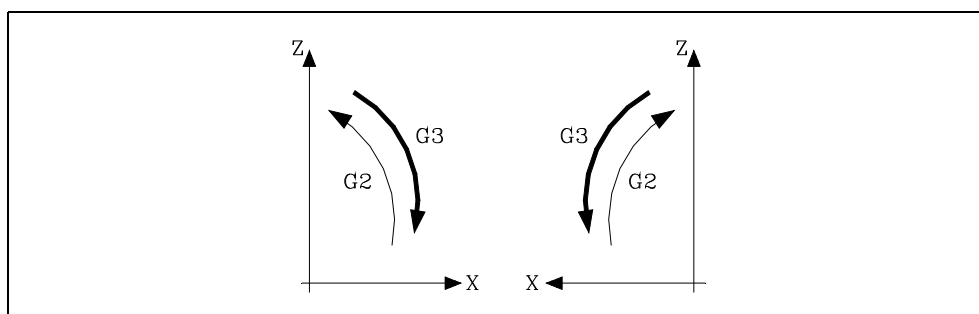
Movements programmed after G02 and G03 are executed in the form of a circular path and at the programmed feedrate "F".

The following example shows the direction of G02 and G03 on different machines. Observe how the relative tool position is maintained with respect to the axes.

- Horizontal lathes:



- Vertical lathes:

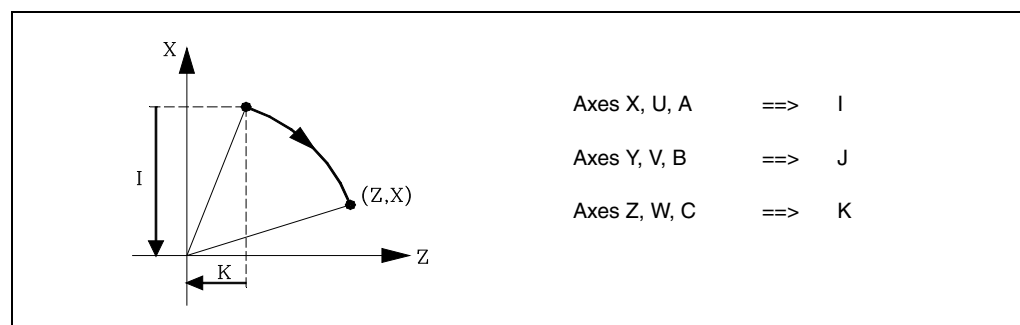


Circular interpolation can only be executed on a plane. The form of definition of circular interpolation is as follows :

Cartesian coordinates

The coordinates of the endpoint of the arc and the position of the center with respect to the starting point are defined according to the axes of the work plane.

The center coordinates are defined in radius by the letters I, J, or K, each one of these being associated to the axes as follows: When not defining the center coordinates, the CNC assumes that their value is zero.



6.

PATH CONTROL
Circular interpolation (G02, G03)



CNC 8037

·T· MODEL
SOFT: V01.4x

Programming format:

Plane XY:	G02(G03)	X±5.5	Y±5.5	I±6.5	J±6.5
Plane ZX:	G02(G03)	X±5.5	Z±5.5	I±6.5	K±6.5
Plane YZ:	G02(G03)	Y±5.5	Z±5.5	J±6.5	K±6.5

The programming order of the axes is always maintained regardless of the plane selected,, as are the respective center coordinates.

Plane AY:	G02(G03)	Y±5.5	A±5.5	J±6.5	I±6.5
Plane XU:	G02(G03)	X±5.5	U±5.5	I±6.5	I±6.5

Polar coordinates

It is necessary to define the angle to be traveled Q and the distance from the starting point to the center (optional), according to the axes of the work plane.

The center coordinates are defined in radius by the letters I, J, or K, each one of these being associated to the axes as follows:

Axes X, U, A	==>	I
Axes Y, V, B	==>	J
Axes Z, W, C	==>	K

If the center of the arc is not defined, the CNC will assume that it coincides with the current polar origin.

Programming format:

Plane XY:	G02(G03)	Q±5.5	I±6.5	J±6.5
Plane ZX:	G02(G03)	Q±5.5	I±6.5	K±6.5
Plane YZ:	G02(G03)	Q±5.5	J±6.5	K±6.5

Cartesian coordinates with radius programming

The coordinates of the endpoint of the arc and radius R are defined.

Programming format:

Plane XY:	G02(G03)	X±5.5	Y±5.5	R±6.5
Plane ZX:	G02(G03)	X±5.5	Z±5.5	R±6.5
Plane YZ:	G02(G03)	Y±5.5	Z±5.5	R±6.5

If a complete circle is programmed, with radius programming, the CNC will show the corresponding error, as infinite solutions exist.

6.

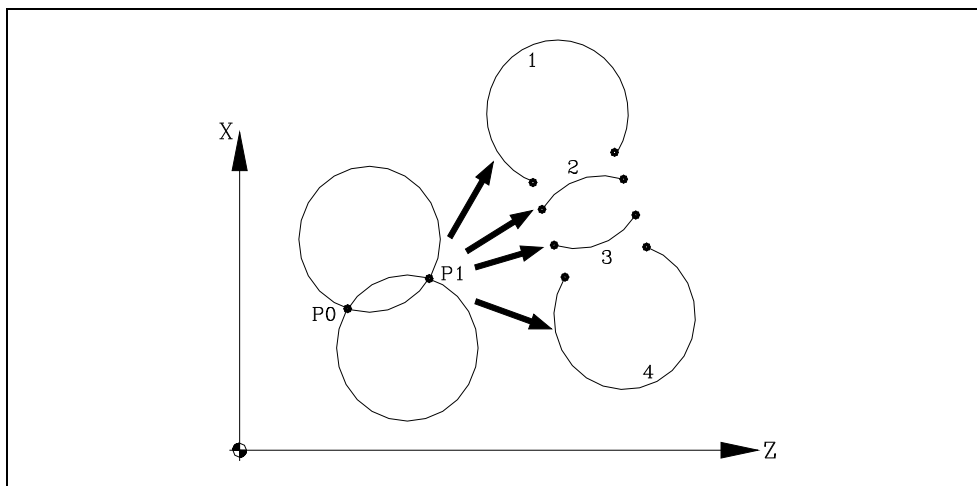
PATH CONTROL
Circular interpolation (G02, G03)

FAGOR 

CNC 8037

·T· MODEL
SOFT: V01.4x

If an arc is less than 180°, the radius is programmed with a plus sign, and a minus sign if it is more than 180°.



If P0 is the starting point and P1 the endpoint, there are 4 arcs which have the same value passing through both points.

Depending on the circular interpolation G02 or G03, and on the radius sign, the relevant arc is defined. Thus the programming format of the sample arcs is as follows:

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

Execution of the circular interpolation

From the programmed path, the CNC will calculate the radius for both the starting point and end point of the arc. Although both of them should be "exactly" the same, general parameter "CIRINERR" allows a certain calculation tolerance by establishing the maximum difference between these two radii. When exceeding this value, the CNC will issue the corresponding error message.

In all these programming cases, the CNC checks that the center or radius coordinates do not exceed 214748.3647mm. Otherwise, the CNC will display the corresponding error.

The programmed feedrate "F" may vary between 0% and 120% via the switch located on the Control Panel of the CNC, or by selecting between 0% and 255% from the PLC, or via the DNC or the program.

The CNC, however, is equipped with the general machine parameter "MAXFOVR" to limit maximum feedrate variation.

If the general machine parameter "PORGMOVE" has been selected and a circular interpolation (G02 or G03) is programmed, the CNC assumes the center of the arc to be a new polar origin.

Functions G02 and G03 are modal and incompatible both among themselves and with G00, G01, G33 and G34. Functions G02 and G03 can be programmed as G2 and G3.

Also, function G74 (home search) and G75 (probing) cancel the G02 and G03 functions.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G00 or G01, depending on how general machine parameter "IMOVE" has been set.

6.

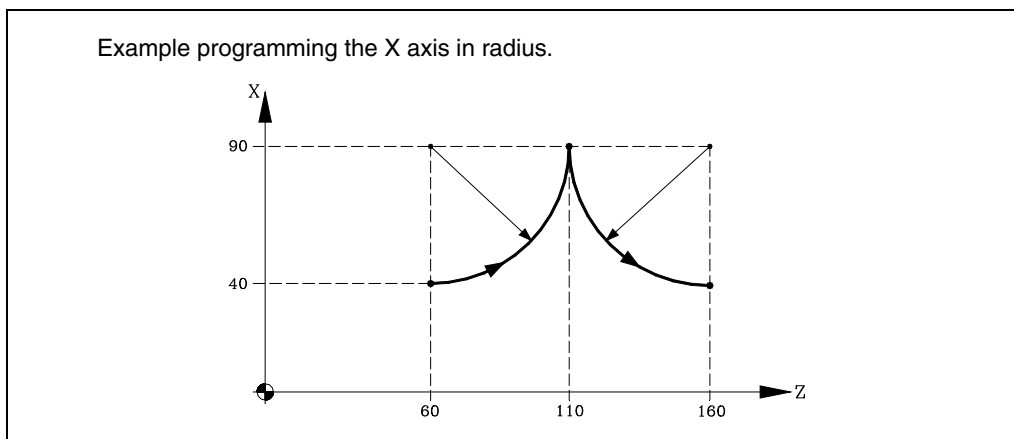
PATH CONTROL Circular interpolation (G02, G03)



CNC 8037

·T· MODEL
SOFT: V01.4x

Programming examples



Various programming modes analyzed below, point X40 Z60 being the starting point.

Cartesian coordinates:

```
G90 G03 X90 Z110 I50 K0
X40 Z160 I10 K50
```

Polar coordinates:

```
G90 G03 Q0 I50 K0
Q-90 I0 K50
```

or:

```
G93 I90 J60          ; Define Polar center
G03 Q0
G93 I90 J160         ; Define new Polar center
Q-90
```

Cartesian coordinates with radius programming:

```
G90 G03 X90 Z110 R50
X40 Z160 R50
```

6.

PATH CONTROL
Circular interpolation (G02, G03)

FAGOR 

CNC 8037

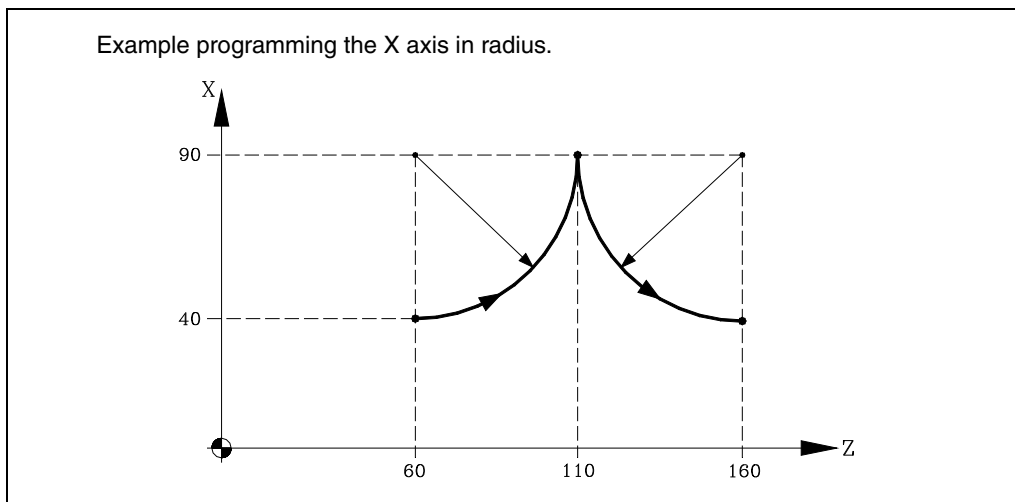
·T· MODEL
SOFT: V01.4x

6.4 Circular interpolation with absolute arc center coordinates (G06)

By adding function G06 to a circular interpolation block you can program the coordinates of the center of the arc (I,J, or K) in absolute coordinates i.e. with respect to the zero origin and not to the beginning of the arc.

The center coordinates of each axis are programmed in either radius or diameter depending on the programming units set by axis machine parameter "DFORMAT".

Function G06 is not modal, so it should be programmed any time the coordinates of the center of the arc are required in absolute coordinates. G06 can be programmed as G6.



Various programming modes analyzed below, point X40 Z60 being the starting point.

Cartesian coordinates:

```
G90 G06 G03 X90 Z110 I90 K60
G06 X40 Z160 Y40 I90 K160
```

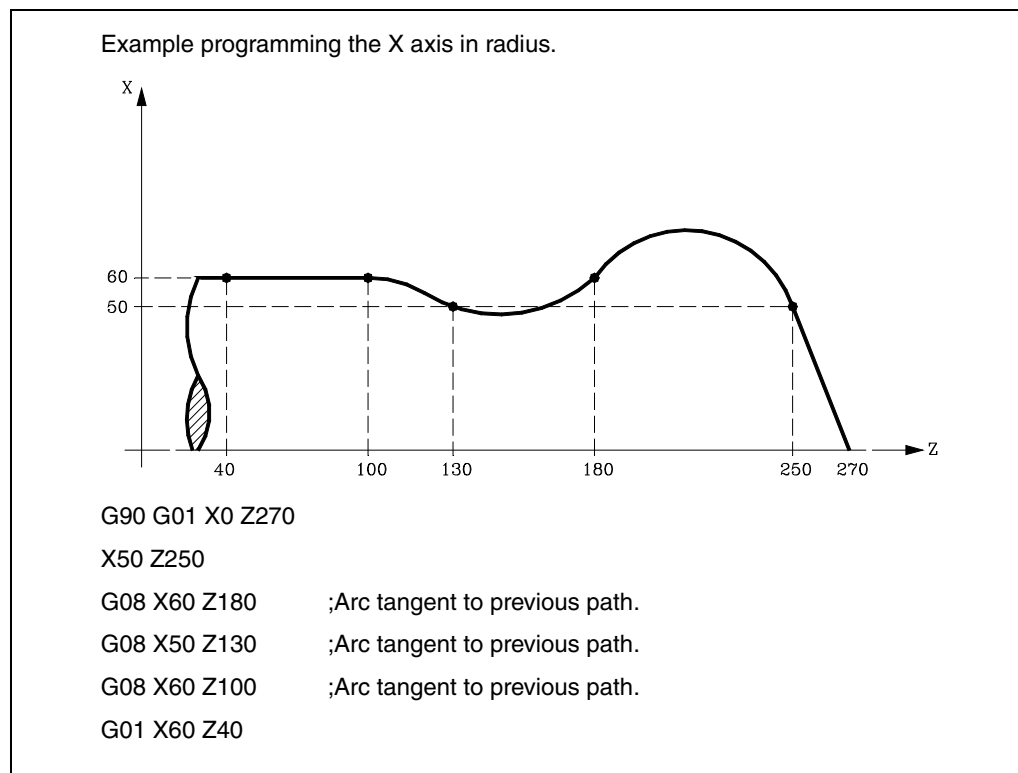
Polar coordinates:

```
G90 G06 G03 Q0 I90 K60
G06 Q-90 I90 K160
```

6.5 Arc tangent to previous path (G08)

Via function G08 you can program an arc tangential to the previous path without having to program the coordinates (I,J &K) of the center.

Only the coordinates of the endpoint of the arc are defined, either in polar coordinates or in Cartesian coordinates according to the axes of the work plane.



Function G08 is not modal, so it should always be programmed if you wish to execute an arc tangential to the previous path. Function G08 can be programmed as G8.

Function G08 enables the previous path to be a straight line or an arc and does not alter its history. The same function G01, G02 or G03 stays active after the block is finished.



When using function G08 it is not possible to execute a complete circle, as an infinite range of solutions exists. The CNC displays the corresponding error code.

6.

PATH CONTROL
Arc tangent to previous path (G08)

FAGOR

CNC 8037

·T· MODEL
SOFT: V01.4x

6.6 Arc defined by three points (G09)

Through function G09 you can define an arc by programming the endpoint and an intermediate point (the starting point of the arc is the starting point of the movement). In other words, instead of programming the coordinates of the center, you program any intermediate point.

The endpoint of the arc is defined in Cartesian or polar coordinates, and the intermediate point is always defined in Cartesian coordinates by the letters I, J, or K, each one being associated to the axes as follows:

Axes X, U, A ==> I

Axes Y, V, B ==> J

Axes Z, W, C ==> K

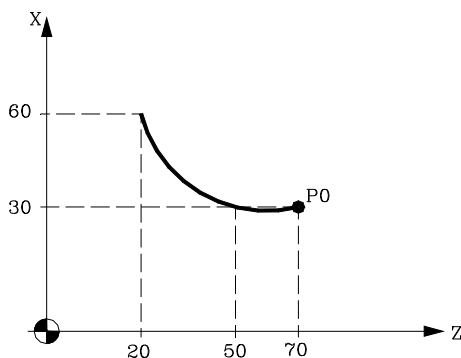
In Cartesian coordinates:

G18 G09 X±5.5 Z±5.5 I±5.5 K±5.5

Polar coordinates:

G18 G09 R±5.5 Q±5.5 I±5.5 K±5.5

Example:



Example where P0 is the starting point.

Programming the X axis in radius.

G09 X60 Z20 I30 K50

Programming the X axis in diameter.

G09 X120 Z20 I60 K50

Function G09 is not modal, so it should always be programmed if you wish to execute an arc defined by three points. Function G09 can be programmed as G9.

When G09 is programmed it is not necessary to program the direction of movement (G02 or G03).

Function G09 does not alter the history of the program. The same G01, G02 or G03 function stays active after finishing the block.

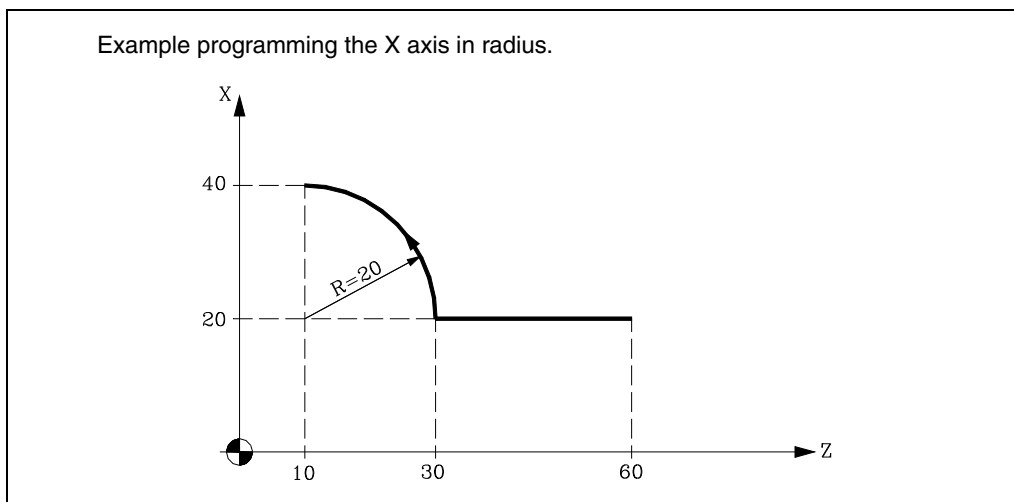


Function G09 may not be used to programmed a full circle because all three points must be different. The CNC displays the corresponding error code.

6.7 Tangential entry at the beginning of a machining operation (G37)

Via function G37 you can tangentially link two paths without having to calculate the intersection points.

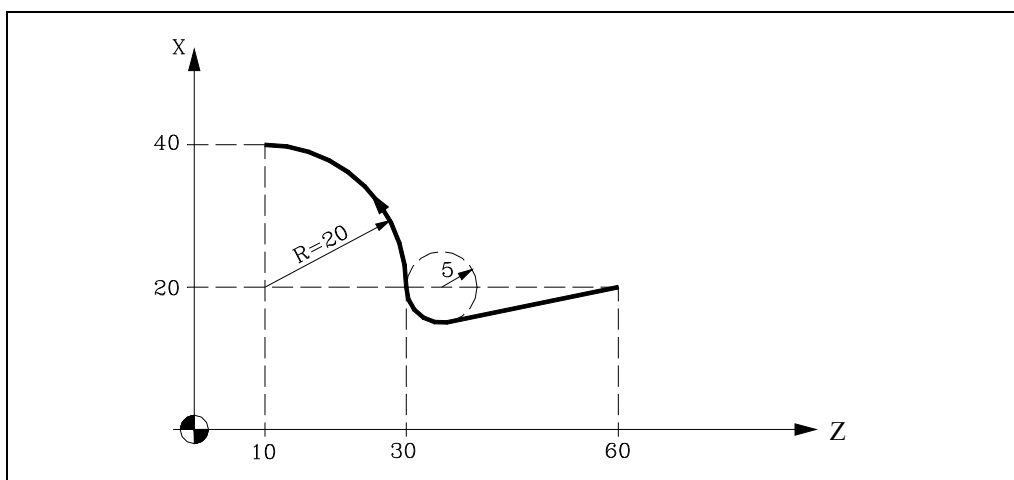
Function G37 is not modal, so it should always be programmed if you wish to start a machining operation with tangential entry:



If the starting point is X20 Z60 and you wish to machine an arc (the path of approach being straight) you should program:

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

If, however, in the same example you require the entrance of the tool to the part to be machined tangential to the path and describing a radius of 5 mm, you should program:



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

As can be seen in the figure, the CNC modifies the path so that the tool starts to machine with a tangential entry to the part.

You have to program Function G37 plus value R in the block which includes the path you want to modify.

R5.5 should appear in all cases following G37, indicating the radius of the arc which the CNC enters to obtain tangential entry to the part. This R value must always be positive.

Function G37 should only be programmed in the block which includes a straight-line movement (G00 or G01). If you program in a block which includes circular movement (G02 or G03), the CNC displays the corresponding error.

6.

PATH CONTROL
Tangential entry at the beginning of a machining operation (G37)

FAGOR 

CNC 8037

·T· MODEL
SOFT· V01.4x

6.8 Tangential exit at the end of a machining operator (G38)

Function G38 enables the ending of a machining operation with a tangential exit of the tool. The path should be in a straight line (G00 or G01). Otherwise, the CNC will display the corresponding error.

Function G38 is not modal, so it should be programmed whenever a tangential exit of the tool is required.

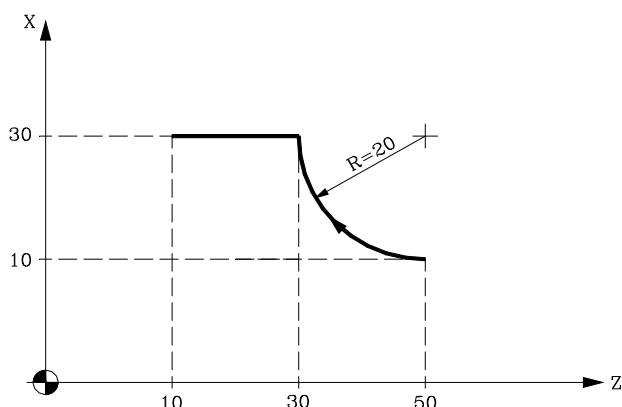
Value R 5.5 should always appear after G38. It also indicates the radius of the arc which the CNC applies to get a tangential exit from the part. This R value must always be positive.

6.

PATH CONTROL

Tangential exit at the end of a machining operator (G38)

Example programming the X axis in radius.

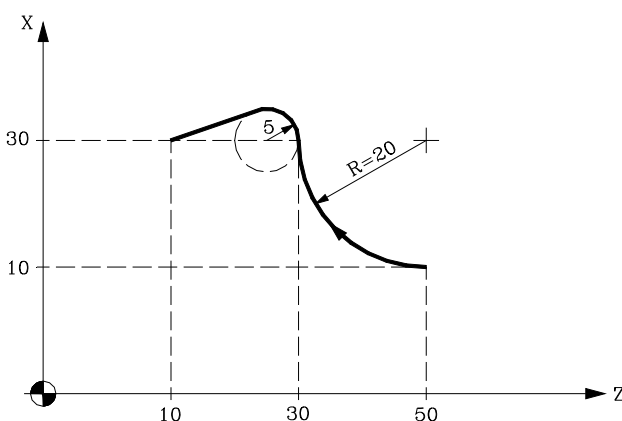


If the starting point is X0 Y30 and you wish to machine an arc (the exit paths being straight) you must program:

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

If, however, in the same example you wish the exit from machining to be done tangentially and describing a radius of 5 mm, you should program :

```
G90 G02 G38 R5 X30 Z30 R20
G00 X30 Z10
```



CNC 8037

·T· MODEL
SOFT: V01.4x

6.9 Automatic radius blend (G36)

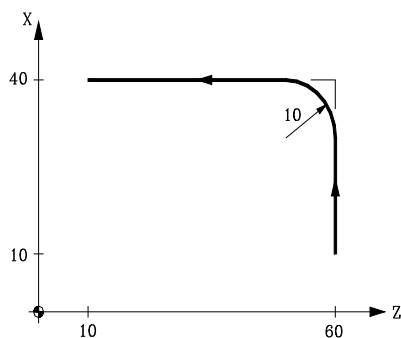
It is possible to round a corner via function G36 with a particular radius, without having to calculate the center nor the start and end points of the arc.

Function G36 is not modal, so it should be programmed whenever controlled corner rounding is required.

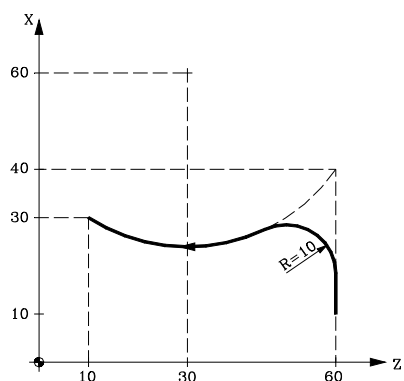
This function should be programmed in the block in which the movement the end you want to round is defined.

The R5.5 value should always follow G36. It also indicates the rounding radius which the CNC applies to get the required corner rounding. This R value must always be positive.

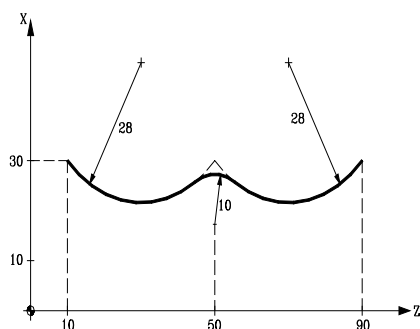
Example programming the X axis in diameter.



```
G90 G01 X20 Z60
G01 G36 R10 X80
Z10
```



```
G90 X20 Z60
G01 G36 R10 X80
G02 X60 Z10 I20 K-30
```



```
G90 X60 Z90
G02 G36 R10 X60 Z50 R28
X60 Z10 R28
```

6.

PATH CONTROL
Automatic radius blend (G36)

FAGOR 

CNC 8037

·T· MODEL
SOFT: V01.4x

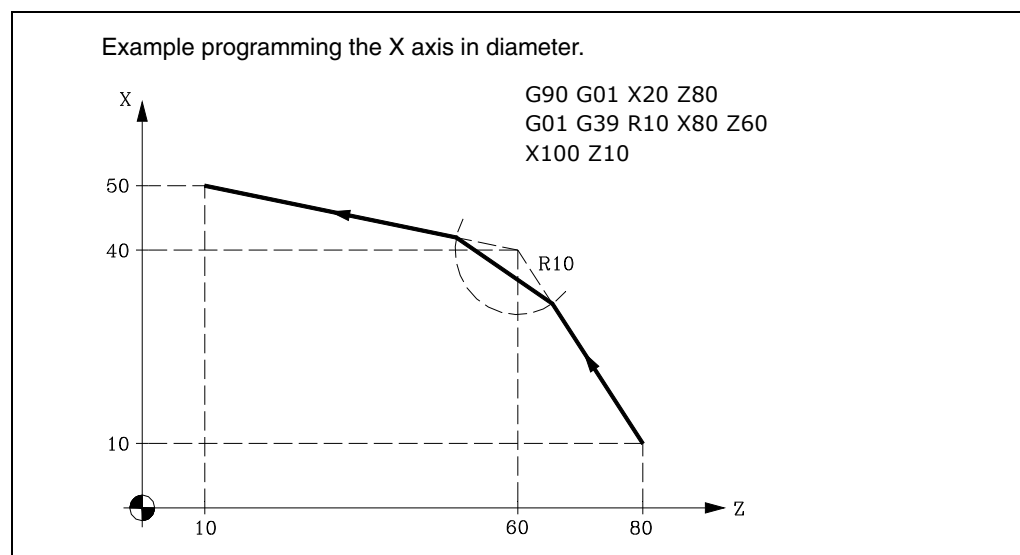
6.10 Chamfer (G39)

In machining operations it is possible (using G39) to chamfer corners between two straight lines, without having to calculate intersection points.

Function G39 is not modal, so it should be programmed whenever the chamfering of a corner is required.

This function should be programmed in the block in which the movement whose end you want to chamfer is defined.

The R5.5 value should always follow G39. It also indicates the distance from the end of the programmed movement as far as the point where you wish to carry out the chamfering. This R value must always be positive.



6.11 Threading (G33)

If the machine spindle is equipped with a rotary encoder, you can thread with a tool tip via function G33.

Although this threading is often done along the entire length of an axis, the CNC enables threading to be done interpolating more than one axis at a time.

Programming format:

G33 X....C L Q

X...C ± 5.5 End point of the thread

L 5.5 Thread pitch

Q ± 3.5 Optional. It indicates the spindle angular position (± 359.9999) of the thread's starting point. This makes it possible to make threads with multiple entries. If not programmed, a value of 0 is assumed.

Some points to consider:

Whenever G33 is executed, if s.m.p. M19TYPE (P43) =0, and before making the thread, the CNC references the spindle (home search).

Spindle machine parameter M19TYPE (P43) must be set to "1" in order to be able to program parameter Q (angular spindle position).

When executing function G33 Q (s.m.p. M19TYPE (P43) =1), spindle had to be homed after the last power-up and before executing the thread.

When executing function G33 Q (s.m.p. M19TYPE (P43) =1) and s.m.p. DECINPUT (P31) =NO, no home search is needed because the spindle will be homed automatically after power-up when turning the spindle in M3 or in M4 for the first time.

This search will be carried out at the feedrate set by s.m.p. REFEED2 (P35). After finding home, the spindle will speed up or slow down to the programmed speed without stopping.

If the spindle has motor feedback with a SINCOS encoder (without reference mark), the home search will be done directly at the programmed S speed without going through the speed set by s.m.p. REFEED2.

If after power-up, an M19 is executed before an M3 or M4, that M19 will be executed without homing the spindle when executing the first M3 or M4.

If the feedback device does not have the reference mark synchronized, the home search in M3 might not coincide with the home search in M4. This does not happen with FAGOR feedback.

When working in round corner mode (G05), it is possible to blend different threads in the same part. When joining threads, only the first one may have an entry angle (Q).

While function G33 is active, neither the programmed feedrate "F" nor the programmed Spindle speed "S" can be varied. They will both be set to 100%. The spindle override is ignored both during machining and when moving back.

Function G33 is modal and incompatible with G00, G01, G02, G03, G34 and G75.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G00 or G01, depending on how general machine parameter "IMOVE" has been set.

6.

PATH CONTROL
Threading (G33)

FAGOR 

CNC 8037

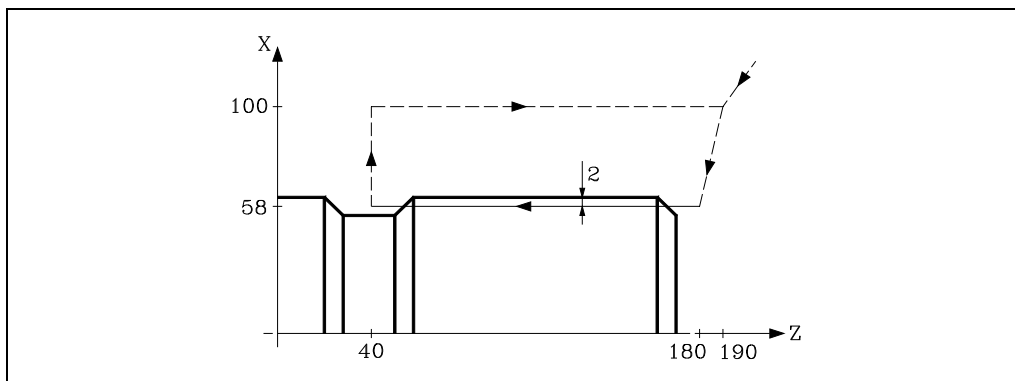
·T· MODEL
SOFT: V01.4x

Programming examples:

In the following examples, the X axis is programmed in diameter.

Longitudinal threadcutting

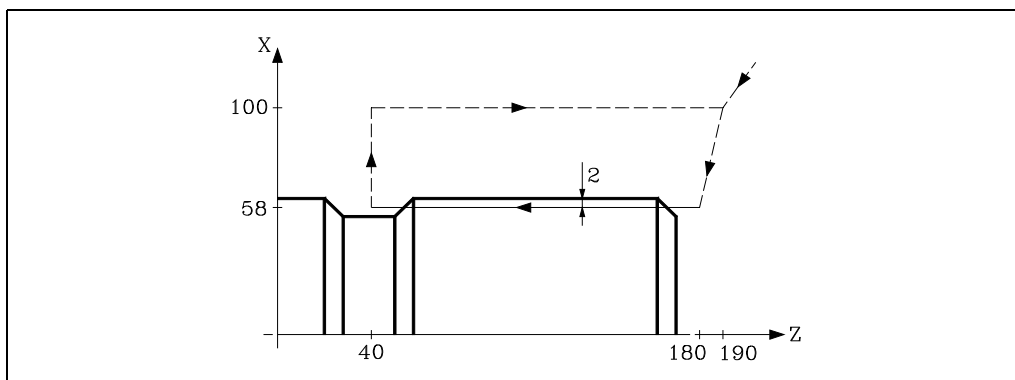
We would like to make a thread in a single pass with a depth of 2mm and a pitch of 5 mm.



```
G90 G00 X200 Z190
X116 Z180
G33 Z40 L5 ; Tapping.
G00 X200
Z190
```

Longitudinal multiple threadcutting

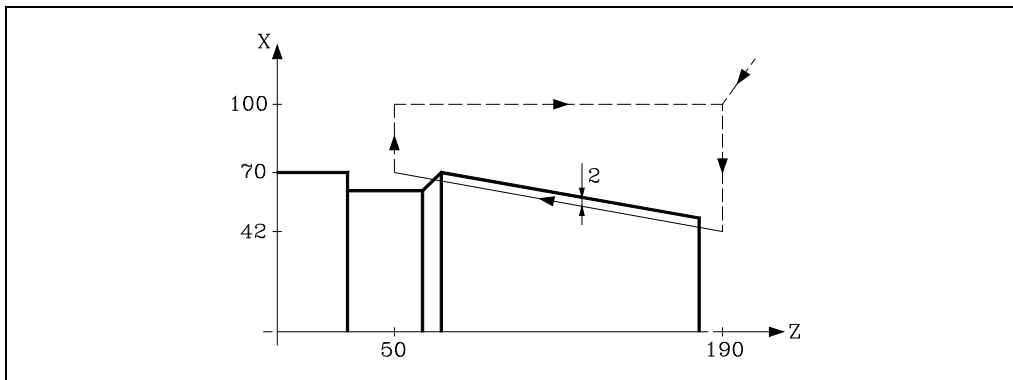
To make a longitudinal thread with two entries. The threads are shifted 180° and each one has a depth of 2mm and a pitch of 5 mm.



```
G90 G00 X200 Z190
X116 Z180
G33 Z40 L5 Q0 ; First threadcutting.
G00 X200
Z190
X116 Z180
G33 Z40 L5 Q180 ; Second threadcutting.
G00 X200
Z190
```

Taper thread

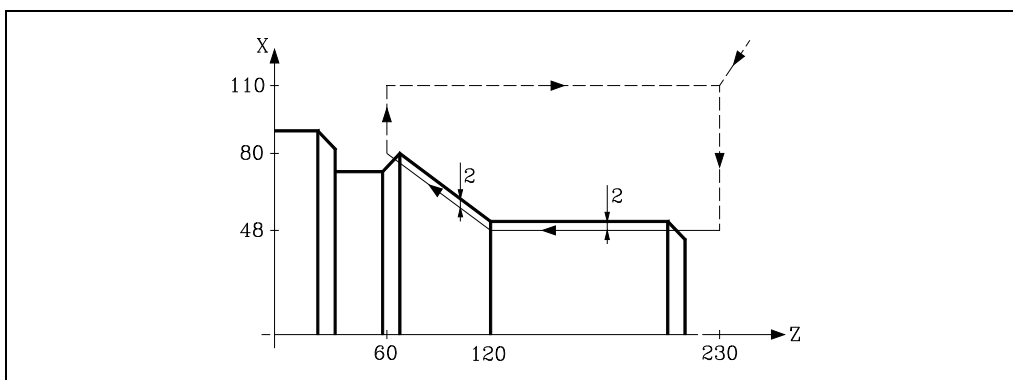
We would like to make a taper thread in a single pass with a depth of 2mm and a pitch of 5 mm.



```
G90 G00 X200 Z190
X84
G33 X140 Z50 L5 ; First threadcutting.
G00 X200
Z190
```

Thread blending (joining)

We would like to blend a longitudinal thread with a taper thread with a depth of 2mm and a pitch of 5 mm.



```
G90 G00 G05 X220 Z230
X96
G33 Z120 L5 ; Longitudinal threadcutting.
Z160 Z60 L5 ; Taper thread.
G00 X200
Z230
```

6.

PATH CONTROL
Threading (G33)

FAGOR

CNC 8037

·T· MODEL
SOFT: V01.4x

6.12 Withdrawal of axes when interrupting a threading operation (G233).

With function G233 it is possible to program the safety distance the axes withdraw to when interrupting a thread (G33) either using the [STOP] key or feedhold.

This function is modal, it must be programmed alone in the block and it is displayed in the history. This function determines the thread exit for all the threads programmed after it. If there are several G33 functions in a row and you would like to do a different withdrawal on each one of them, program the corresponding function G233 before each G33 function.

Function G233 may be canceled by programming it alone in a block or by programming all the coordinates as zero. In either case, G233 disappears from history.

If the [STOP] key is pressed while executing a thread and function G233 is active, the axes withdraw according to the distances programmed in that function. If after the withdrawal, there is an active DSTOP, it will keep executing the blocks until an ESTOP is found.

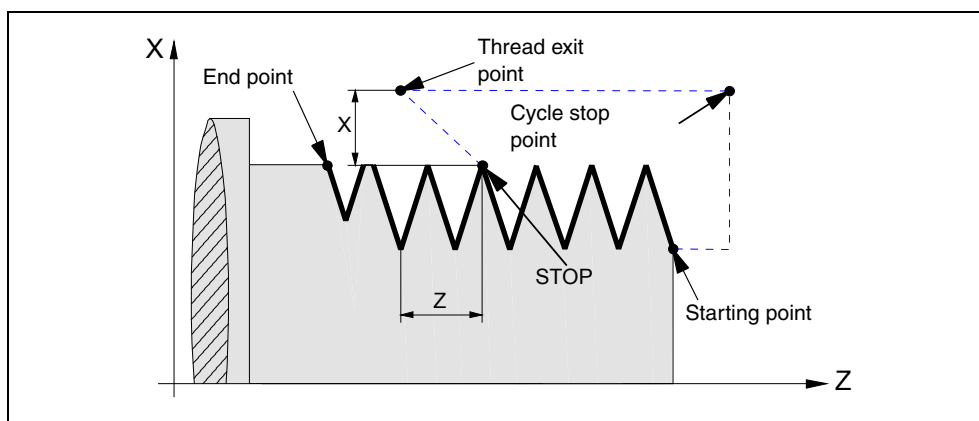
Programming format

G233 X...C

X...C +/-5.5 Thread exit distances.

Example:

- X: Incremental distance (positive or negative) to move along the thread exit axis (X axis).
- Z: Incremental distance to move along the thread axis (Z axis).



Programming example

Programming a thread with G33 and G233.

```

N10 G90 G18 S500 M3
N20 G0 X20
N30 Z0
N40 X10
N50 G233 X5 Z-20 ;Block activating the thread withdrawal (5mm in X and -20mm in Z).
N60 G33 Z-30 L5 ;Threading block that may be interrupted with STOP.
N70 G33 X15 Z-50 L5 ;Exit block previous threading.

```

Withdrawal of axes when interrupting a threading canned cycle (G86, G87)

In the threading cycles (G86, G87) where a thread exit has been programmed, there is no need to program a G233. The withdrawal distances will be indicated in parameters D and J of that cycle.

While executing those cycles, when pressing the [STOP] key or feedhold, once the tool has been withdrawn, it returns to the starting point of the cycle. Then, the machine remains stopped waiting for the [START] command to repeat the interrupted pass.

6.

PATH CONTROL

Withdrawal of axes when interrupting a threading operation (G233).



CNC 8037

·T· MODEL
SOFT: V01.4x

6.13 Variable pitch threads (G34)

To make variable-pitch threads, the spindle of the machine must have a rotary encoder.

Although this threading is often done along the entire length of an axis, the CNC enables threading to be done interpolating more than one axis at a time.

Programming format:

G34 X.....C L Q K

X...C ± 5.5	End point of the thread
L 5.5	Thread pitch
Q ± 3.5	Optional. It indicates the spindle angular position (± 359.9999) of the thread's starting point. If not programmed, a value of "0" is assumed.
K ± 5.5	Thread pitch increase or decrease per spindle turn.

Some points to consider:

Whenever G34 is executed and before making the thread, the CNC references the spindle (home search) and positions the spindle at the angular position indicated by parameter Q.

Parameter "Q" is available when spindle machine parameter "M19TYPE" has been set to "1".

When working in round corner mode (G05), it is possible to blend different threads in the same part.

While function G34 is active, neither the programmed feedrate "F" nor the programmed Spindle speed "S" can be varied. They will both be set to 100%.

Function G34 is modal and incompatible with G00, G01, G02, G03, G33 and G75.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G00 or G01, depending on how general machine parameter "IMOVE" has been set.

Blending a fixed-pitch thread (G33) with a variable-pitch thread (G34).

The starting thread pitch (L) of the G34 must coincide with the thread pitch of the G33.

The pitch increase in the first turn of the spindle in variable pitch will be half and increment (K/2) and in the rest of the turns it will be the full increment K.

Blending a variable-pitch thread (G34) with a fixed-pitch thread.

It is used to finish a variable-pitch thread (G34) with a portion of the thread keeping the final pitch of the previous thread. The fixed-pitch thread is not programmed with G33 but with G34 ... L0 K0.

Blending two variable-pitch threads (G34).

Two variable-pitch threads (G34) cannot be blended together.

6.

PATH CONTROL
Variable pitch threads (G34)



CNC 8037

·T· MODEL
SOFT: V01.4x

6.14 Move to hardstop (G52)

By means of function G52 it is possible to program the movement of an axis until running into an object. This feature may be interesting for forming machines, live tailstocks, bar feeders, etc.

The programming format is:

G52 X..C ±5.5

After G52, program the desired axis as well as the target coordinate of the move.

The axis will move towards the programmed target coordinate until running into something. If the axis reaches the programmed target coordinate without running into the hardstop it will stop.

Function G52 is not modal; therefore, it must be programmed every time this operation is to be carried out.

Also, it assumes functions G01 and G40 modifying the program history. It is incompatible with functions G00, G02, G03, G33, G34, G41, G42, G75 and G76.

6.

PATH CONTROL
Move to hardstop (G52)

6.15 Feedrate "F" as an inverted function of time (G32)

There are instances when it is easier to define the time required by the various axes of the machine to reach the target point instead of defining a common feedrate for all of them.

A typical case may be when a linear axis (X, Z) has to move together (interpolated) with a rotary axis programmed in degrees.

Function G32 indicates that the "F" functions programmed next set the time it takes to reach the target point.

In order for a greater value of "F" to indicate a greater feedrate, the value assigned to "F" is defined as "Inverted function of time" and it is assumed as the activation of this feature.

"F" units: 1/min

Example: G32 X22 F4

indicates that the movement must be executed in 1/4 minute; i.e. in 0.25 minutes.

Function G32 is modal and incompatible with G94 and G95.

On power-up, after executing M02, M30 or after an Emergency or Reset, the CNC assumes G94 or G95 depending on the setting of general machine parameter "IFFED".

Some points to consider:

The CNC variable PROGFIN will show the feedrate programmed as an inverted function of time and variable FEED will show the resulting feedrate in mm/min or inches/min.

If the resulting feedrate of any axis exceeds the maximum value set by machine parameter "MAXFEED", the CNC will apply this maximum value.

The programmed "F" is ignored on G00 movements. All the movements will be carried out at the feedrate set by axis machine parameter "G00FEED".

When programming "F0" the movement will be carried out at the feedrate set by axis machine parameter "MAXFEED".

Function G32 may be programmed and executed in the PLC channel.

Function G32 is canceled in JOG mode.

6.

PATH CONTROL
Feedrate "F" as an inverted function of time (G32)



CNC 8037

·T· MODEL
SOFT: V01.4x

ADDITIONAL PREPARATORY FUNCTIONS

7

7.1 Interruption of block preparation (G04)

The CNC reads up to 20 blocks ahead of the one it is executing, with the aim of calculating beforehand the path to be followed.

Each block is evaluated (in its absence) at the time it is read, but if you wish to evaluate it at the time of execution of the block you use function G04.

This function holds up the preparation of blocks and waits for the block in question to be executed in order to start the preparation of blocks once more.

A case in point is the evaluation of the "status of block-skip inputs" which is defined in the block header.

Example:

```
.  
.   
G04                ; Interrupts block preparation  
/1 G01 X10 Z20     ; Jump condition "/1"  
.   
.
```

Function G04 is not modal, so it should be programmed whenever you wish to interrupt block preparation.

It should be programmed on its own and in the block previous to the one in which the evaluation in execution is required. Function G04 can be programmed as G4.

Every time G04 is programmed, active radius and length compensation are cancelled.

For this reason, care needs to be taken when using this function, because if it is introduced between machining blocks which work with compensation, unwanted profiles may be produced.

Example:

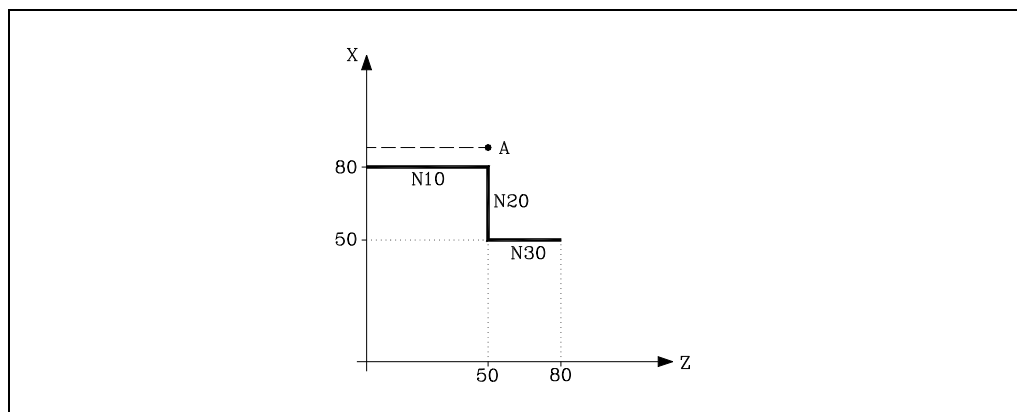
The following program blocks are performed in a section with G41 compensation.

```

...
N10 X80 Z50
N15 G04
/1 N17 M10
N20 X50 Z50
N30 X50 Z80
...

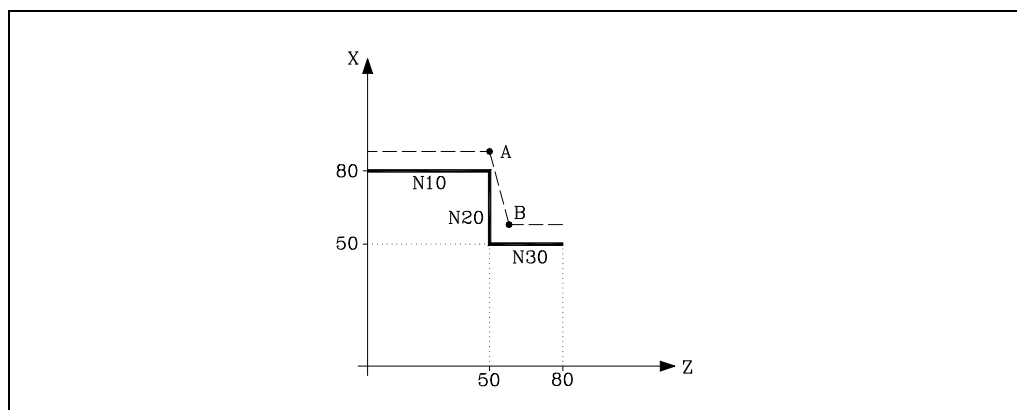
```

Block N15 interrupts block preparation and the execution of block N10 will finish at point A.



Once the execution of block N15 has been carried out, the CNC continues preparing blocks starting from block N17.

As the next point corresponding to the compensated path is point "B", the CNC will move the tool to this point, executing path "A-B".



As you can see, the resulting path is not the required one, so we recommend avoiding the use of function G04 in sections which work with compensation.

7.1.1 G04 K0: Block preparation interruption and coordinate update

The function associated with G04 K0 may be used to update the coordinates of the axes of the channel after finishing particular PLC routines.

The PLC routines that require updating the coordinates of the axes of the channel are the following:

- PLC routine using the SWITCH* marks.
- PLC routines where an axis goes into DRO mode and then back into normal axis mode during the execution of part programs.

G04 operation:

Function	Description
G04	Interrupts block preparation.
G04 K50	It executes a dwell 50 hundredths of a second.
G04 K0 or G04 K	It interrupts block preparation and updates the CNC coordinates to the current position. (G4 K0 works in the CNC and PLC channel).

7.

ADDITIONAL PREPARATORY FUNCTIONS

Interruption of block preparation (G04)

7.2 Dwell (G04 K)

A dwell can be programmed via function G04 K.

The dwell value is programmed in hundredths of a second via format K5 (1..99999).

Example:

```
G04 K50      ; Dwell of 50 hundredths of a second (0.5 seconds)
G04 K200     ; Dwell of 200 hundredths of a second (2 seconds)
```

Function G04 K is not modal, so it should be programmed whenever a dwell is required. Function G04 K can be programmed as G4 K.

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

Note: When programming G04 K0 or G04 K, instead of applying a delay, it only interrupts block preparation and it will refresh the coordinates. See ["7.1.1 G04 K0: Block preparation interruption and coordinate update"](#) on page 99.

7.**ADDITIONAL PREPARATORY FUNCTIONS**

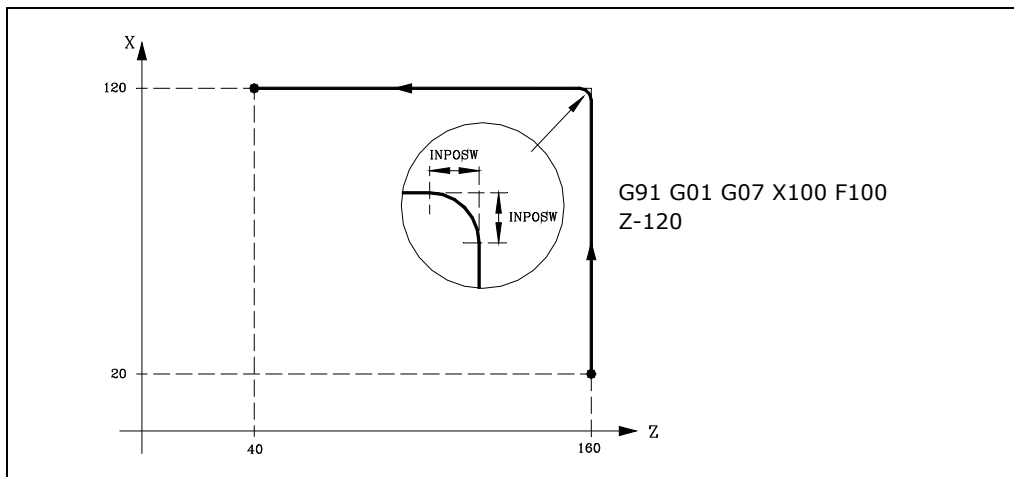
Dwell (G04 K)

7.3 Working with square (G07) and round (G05,G50) corners

7.3.1 G07 (square corner)

When working in G07 (square corner) the CNC does not start executing the following program block until the position programmed in the current block has been reached.

The CNC considers that the programmed position has been reached when the axis is within the "INPOSW" (in-position zone or dead band) from the programmed position.



The theoretical and real profile coincide, obtaining square corners, as seen in the figure.

Function G07 is modal and incompatible with G05, G50 and G51. Function G07 can be programmed as G7.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G05 or G07 depending on how the general machine parameter "ICORNER" is set.

7.

ADDITIONAL PREPARATORY FUNCTIONS

Working with square (G07) and round (G05,G50) corners

FAGOR 

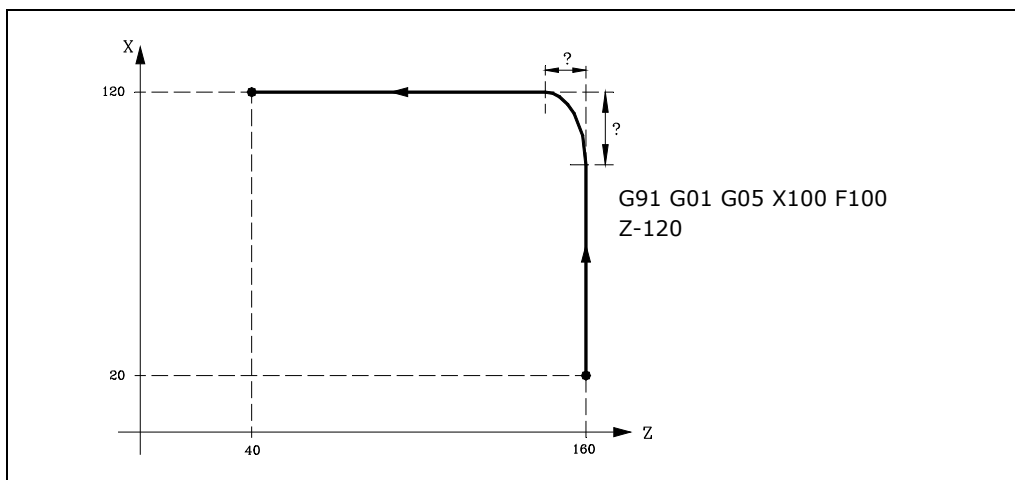
CNC 8037

·T· MODEL
SOFT: V01.4x

7.3.2 G05 (round corner)

When working in G05 (round corner), the CNC starts executing the following block of the program as soon as the theoretical interpolation of the current block has concluded. It does not wait for the axes to physically reach the programmed position.

The distance prior to the programmed position where the CNC starts executing the next block depends on the actual axis feedrate.



Via this function round corners can be obtained, as shown in the figure.

The difference between the theoretical and real profiles depends on the programmed feedrate value "F". The higher the feedrate, the greater the difference between both profiles.

Function G05 is modal and incompatible with G07, G50 and G51. Function G05 can be programmed as G5.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G05 or G07 depending on how the general machine parameter "ICORNER" is set.

7.

ADDITIONAL PREPARATORY FUNCTIONS

Working with square (G07) and round (G05, G50) corners

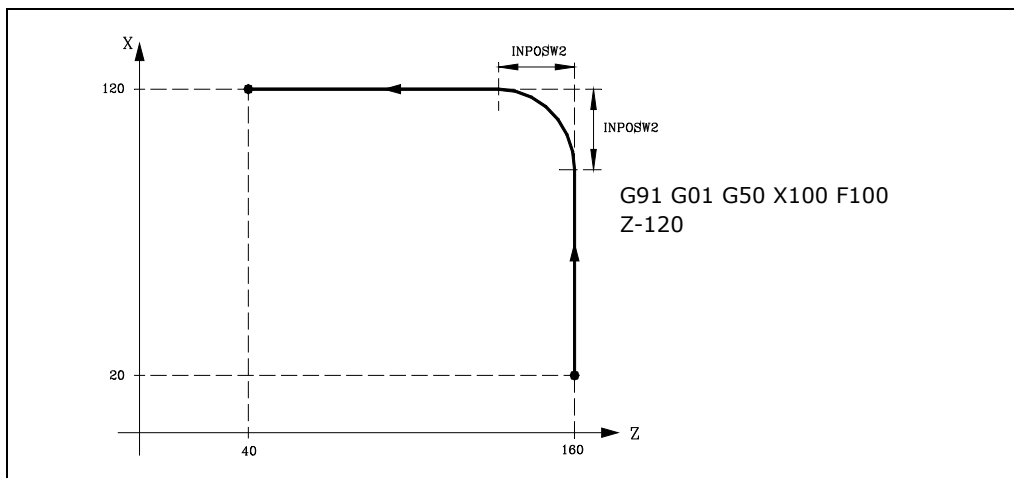


CNC 8037

·T· MODEL
SOFT: V01.4x

7.3.3 Controlled round corner (G50)

When working in G50 (controlled round corner); once the theoretical interpolation of the current block has concluded, the CNC waits for the axis to enter the area defined by machine parameter "INPOSW2" and it then starts executing the following block of the program.



Function G50 assures that the difference between the theoretical and actual paths stays smaller than what was set by machine parameter "INPOSW2".

On the other hand, when working in G05, the difference between the theoretical and real profiles depends on the programmed feedrate value "F". The higher the feedrate, the greater the difference between both profiles.

Function G50 is modal and incompatible with G07, G05 and G51.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G05 or G07 depending on how the general machine parameter "ICORNER" is set.

7.

ADDITIONAL PREPARATORY FUNCTIONS

Working with square (G07) and round (G05, G50) corners

FAGOR 

CNC 8037

· T· MODEL
SOFT: V01.4x

7.4 Look-ahead (G51)

Programs consisting of very small movement blocks (CAM, etc.) tend to run very slowly. Those programs may be executed at high machining speed using the "Look-Ahead" function.

The look-ahead function analyzes in advance the path to be machined (up to 75 blocks) in order to calculate the maximum feedrate for each section of the path. This function provides smoother and faster machining in programs with very small movements, even in the order of microns.

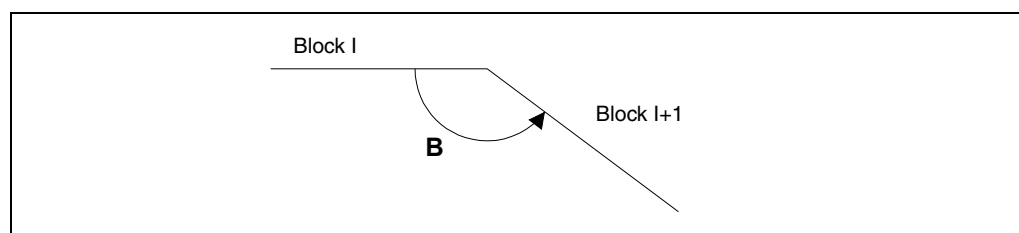
When operating with "Look-Ahead", it is a good idea to adjust the axes so their following error (lag) is as small as possible because the contouring error will be at least equal to the minimum following error.

Programming format.

The programming format is:

G51 [A] E B

A (0-255)	It is optional and it defines the percentage of acceleration to be applied. When not programmed or programmed with a "0" value, the CNC assumes the acceleration value set by machine parameter for each axis.
E (5.5)	Maximum contouring error. The lower this parameter value is, the lower the machining feedrate will be.
B (0-180)	It makes it possible to machine square corners using the Look-ahead function. It indicates the angular value (in degrees) of the programmed corners, below which, the machining operation will be carried out in square corner mode.



Parameter "A" permits using a standard working acceleration and another one to be used when executing with Look-Ahead.

If parameter "B" is not programmed, the square corner management at the corners is canceled.

The square-corner management at the corners is valid for the look-ahead algorithm with jerk management and for the look-ahead algorithm without jerk management.

Considerations for execution:

When calculating the feedrate, the CNC takes the following into account:

- The programmed feedrate.
- The curvature and the corners.
- The maximum feedrates of the axes.
- The maximum accelerations.

If any of the circumstances listed below occurs while executing with Look-Ahead, the CNC slows down to "0" at the previous block and it recovers the machining conditions for Look-Ahead in the next motion block.

- Motionless block.
- Execution of auxiliary functions (M, S, T).
- Single block execution mode.
- MDI mode.
- Tool inspection mode.

If a Cycle Stop, Feed-Hold, etc. occurs while executing in Look-Ahead mode, the machine may not stop at the current block, several additional blocks will be necessary to stop with the permitted deceleration.

7.

ADDITIONAL PREPARATORY FUNCTIONS
Look-ahead (G51)



CNC 8037

•T• MODEL
SOFT: V01.4X

To prevent motionless blocks from causing a square-corner effect, change bit 0 of general machine parameter MANTFCON (P189).

Function properties.

Function G51 is modal and incompatible with G05, G07 and G50. Should any of them be programmed, function G51 will be canceled and the new one will be selected.

Function G51 must be programmed alone in a block and there must be no more information in that block.

On power-up, after executing an M02, M30, or after an EMERGENCY or RESET, the CNC will cancel G51, if it was active, and it will assume G05 or G07 according to the setting of general machine parameter "ICORNER".

On the other hand, the CNC will issue Error 7 (Incompatible G functions) when programming any of the following functions while G51 is active:

G33	Electronic threading.
G34	Variable-pitch threading.
G52	Movement until making contact.
G95	Feedrate per revolution.

7.

ADDITIONAL PREPARATORY FUNCTIONS

Look-ahead (G51)



CNC 8037

·T· MODEL
SOFT: V01.4x

7.4.1 Advanced look-ahead algorithm (integrating Fagor filters)

This mode is indicated when machining accuracy is required, especially if Fagor filters have been set for the axes by machine parameters.

The advanced look-ahead algorithm calculates the feedrates at the corners so as to take the effect of those filters into consideration. When programming G51 E, the contouring errors when machining corners will be closer to the value programmed in that G51 function depending on the filters.

To activate the advanced look-ahead algorithm, use bit 15 of g.m.p. LOOKATYP (P160).

Considerations

- If there are no Fagor filters set by machine parameters in the axes of the main channel, activating the advanced look-ahead algorithm will internally activate FIR filters of the 5th order and a frequency of 30 Hz in all the axes of the channel.
- If there are Fagor filters set by machine parameters, activating the advanced look-ahead algorithm will keep the values of those filters as long as their frequency is not higher than 30Hz. If their frequency is higher than 30 Hz, it will assume the values of 5th order and a frequency of 30 Hz.
If there are several filters defined in the axes of the channel, the one with the lowest frequency will be assumed as long as it is not higher than 30Hz.
- Even if the advanced look-ahead algorithm (using Fagor filters) is active with bit 15 of g.m.p. LOOKATYP (P160), it will not start working in the following cases:
 - If g.m.p. IPOTIME (P73) = 1.
 - If a.m.p. SMOTIME (P58) of any of the axes of the main channel is other than 0.
 - If any of the axes of the main channel has defined by parameter a non-Fagor type filter, a.m.p. TYPE (P71) is other than 2.

In these cases, when activating G51, the CNC will issue the corresponding error message.

7.

ADDITIONAL PREPARATORY FUNCTIONS

Look-ahead (G51)



CNC 8037

·T· MODEL
SOFT: V01.4x

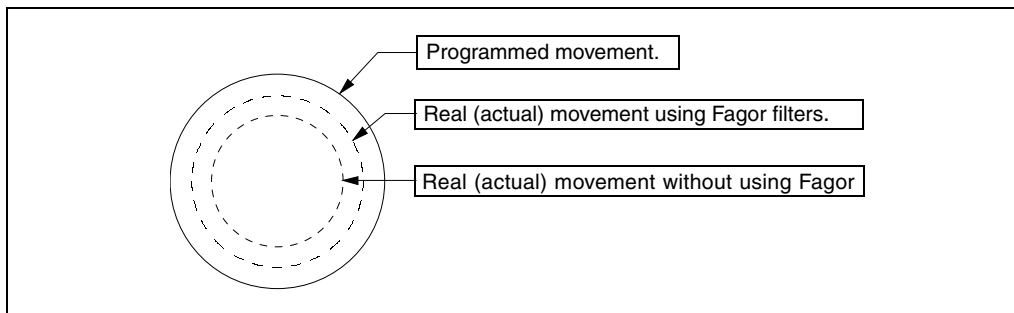
7.4.2 Look-ahead operation with Fagor filters active

This option makes it possible to use Fagor filters with Look-ahead (not advanced look-ahead algorithm). It will only be considered if the advanced look-ahead algorithm is deactivated; that is, if bit 15 of g.m.p. LOOKATYP (P160) = 0.

To activate/deactivate this option, use bit 13 of g.m.p. LOOKATYP (P160).

Effect of Fagor filters when machining circles

When machining circles, the error will be smaller when using Fagor filters than without using these filters:



7.

ADDITIONAL PREPARATORY FUNCTIONS

Look-ahead (G51)

FAGOR 

CNC 8037

·T· MODEL
SOFT: V01.4x

7.5 Mirror image (G10, G11, G12, G13, G14)

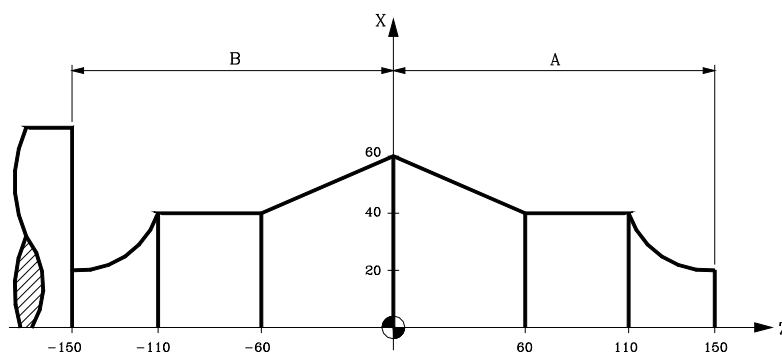
The functions to activate the mirror image are the following.

- G10: Cancel mirror image.
- G11: Mirror image on X axis.
- G12: Mirror image on the Y axis.
- G13: Mirror image on the Z axis
- G14: Mirror image on any axis (X..C), or in several at the same time.

Examples:

G14 W
G14 X Z A B

When the CNC works with mirror images, it executes the movements programmed in the axes that have mirror image selected, with the sign changed.



The following subroutine defines the machining of part "A".

```
G90 G00 X40 Z150
G02 X80 Z110 R60
G01 Z60
X120 Z0
```

The programming of all parts would be :

```
Subroutine execution      ; Machines "A".
G13                      ; Mirror image on the Z axis.
Subroutine execution      ; Machines "B".
M30                      ; End of program
```

Functions G11, G12, G13, and G14 are modal and incompatible with G10.

G11, G12, and G13 can be programmed in the same block, because they are not incompatible with each other. Function G14 must be programmed alone in the block.

If while one of the mirror imaging functions (G11, G12, G13, and G14) is active, a new coordinate origin (part zero) is preset with G92, this new origin will not be affected by the mirror imaging function.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G10.

7.6 Scaling factor (G72)

By using function G72 you can enlarge or reduce programmed parts.

In this way, you can produce families of parts which are similar in shape but of different sizes with a single program.

Function G72 should be programmed on its own in a block. There are two formats for programming G72 :

- Scaling factor applied to all axes.
- Scaling factor applied to one or more axes.

7.

ADDITIONAL PREPARATORY FUNCTIONS

Scaling factor (G72)



CNC 8037

·T· MODEL
SOFT: V01.4x

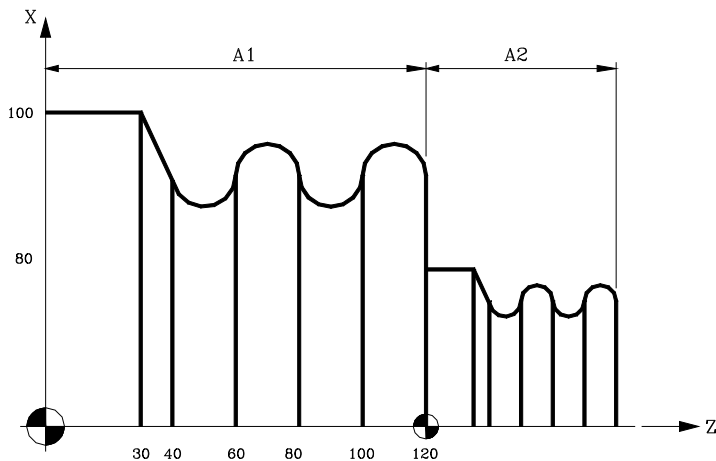
7.6.1 Scaling factor applied to all axes.

The programming format is:

G72 S5.5

Following G72 all coordinates programmed are multiplied by the value of the scaling factor defined by S until a new G72 scaling factor definition is read or the definition is canceled.

Example programming the X axis in diameter.



The following subroutine defines the basic machining operation.

```
G90 X200 Z0
G01 X200 Z30 F150
G01 X160 Z40
G03 X160 Z60 I0 J10
G02 X160 Z80 I0 J10
G03 X160 Z100 I0 J10
G02 X160 Z120 I0 J10
```

The programming of the parts would be :

Execution of subroutine Machines "A1".

```
G92 Z0 ; Coordinate preset
        (zero offset)
```

```
G72 S0.5 ; Applies a scaling factor of 2
```

Execution of subroutine Machines "A2".

```
G72 S1 ; Cancel the scaling factor
M30 ; End of program
```

Function G72 is modal and is cancelled when another scaling factor with a value of S1 is programmed, or on power-up, after executing M02, M30 or after EMERGENCY or RESET.

7.6.2 Scaling factor applied to one or more axes.

The programming format is:

G72 X...C 5.5

After G72 the axis or axes and the required scaling factor are programmed.

All blocks programmed after G72 are treated by the CNC as follows :

1. The CNC calculates the movement of all the axes in relation to the programmed path and compensation.
2. It then applies the scaling factor indicated to the calculated movement of the corresponding axis or axes.

If the scaling factor is applied on one or more axes, the CNC will apply the scaling factor indicated both to the movement of the corresponding axis or axes and to their feedrate.

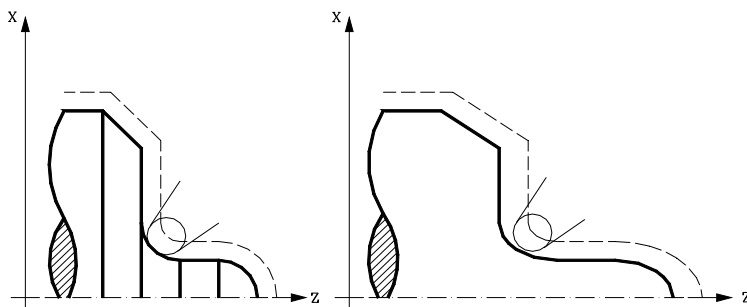
If, within the same program, both scaling factor types are applied, the one applied to all the axes and the one for one or several axes, the CNC applies a scaling factor equal to the product of the two scaling factors programmed for this axis to the axis or axes affected by both types.

Function G72 is modal and will be cancelled when the CNC is turned on, after executing M02, M30 or after an EMERGENCY or RESET.



This type of scaling factor is ignored when simulating without moving the axes.

Application of the scaling factor to the Z axis, working with tool radius compensation.



As it can be observed, the tool path does not coincide with the required path, as the scaling factor is applied to the calculated movement.

7.

ADDITIONAL PREPARATORY FUNCTIONS

Scaling factor (G72)

FAGOR

CNC 8037

·T· MODEL
SOFT: V01.4x

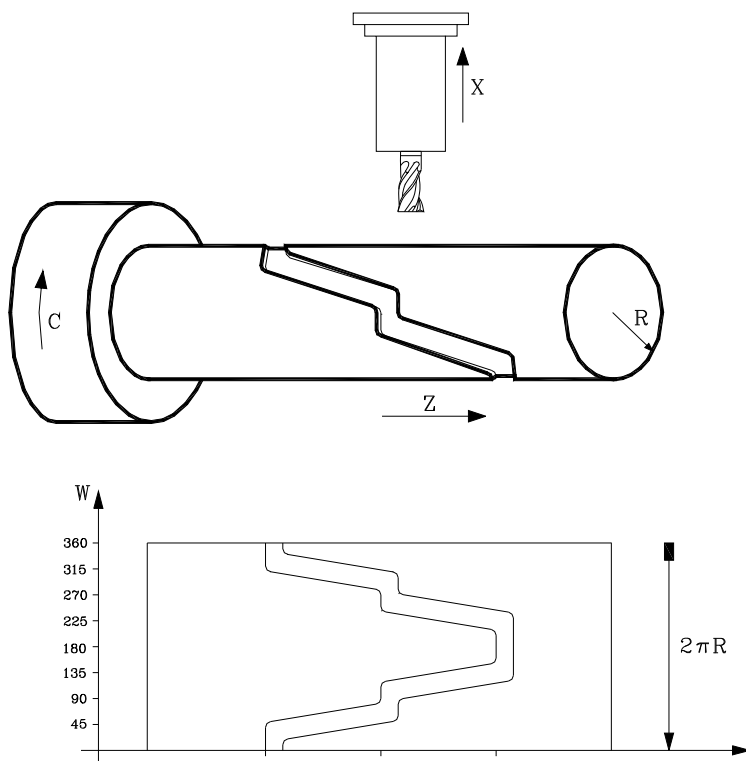
If a scaling factor equal to $360/2\pi R$ is applied to a rotary axis, R being the radius of the cylinder on which you wish to machine, this axis can be considered linear, and any figure with tool radius compensation can be programmed on the cylindrical surface.

7.

ADDITIONAL PREPARATORY FUNCTIONS
Scaling factor (G72)

Example programming the X axis in diameter, assuming that the radius used to make the groove on the cylinder is R20.

Scaling factor to apply = $360/(2\pi R) = 2.86$



G16 ZC

G90 G42 G01 Z70 C0

; Positioning at starting point

G91 X-4

; Penetration

G72 C2.86

; Scaling factor

G90 G36 R5 C45

G36 R5 Z130 C90

G36 R5 C112.5

G36 R5 Z190 C157.5

G36 R5 C202.5

G36 R5 Z130 C247.5

G36 R5 C270

G36 R5 Z70 C315

G36 R5 C360

G91 X4

; Retract

G72 C1

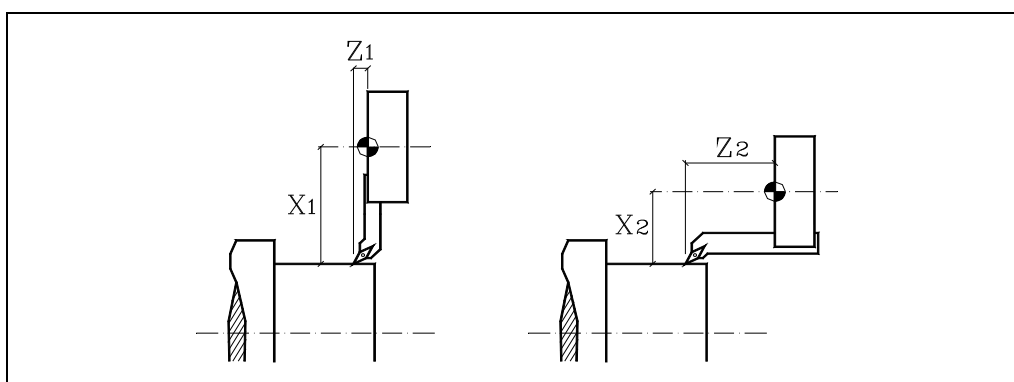
; Cancels the scaling factor

M30

8.1 Tool length compensation

It is always applied to compensate for the length difference between the various tools programmed.

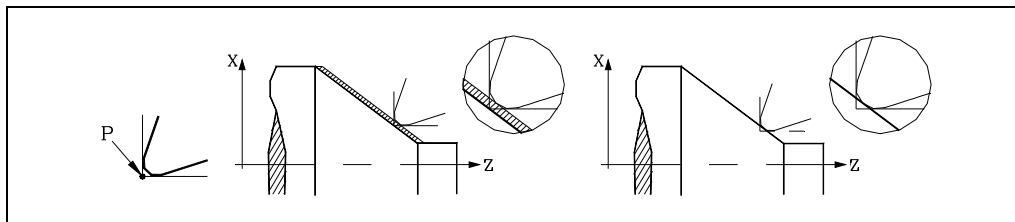
When selecting a new tool, the CNC takes its dimensions into account, defined in the corresponding offset and moves the tool holding turret so the new tool occupies the same position (coordinate) as the previous one.



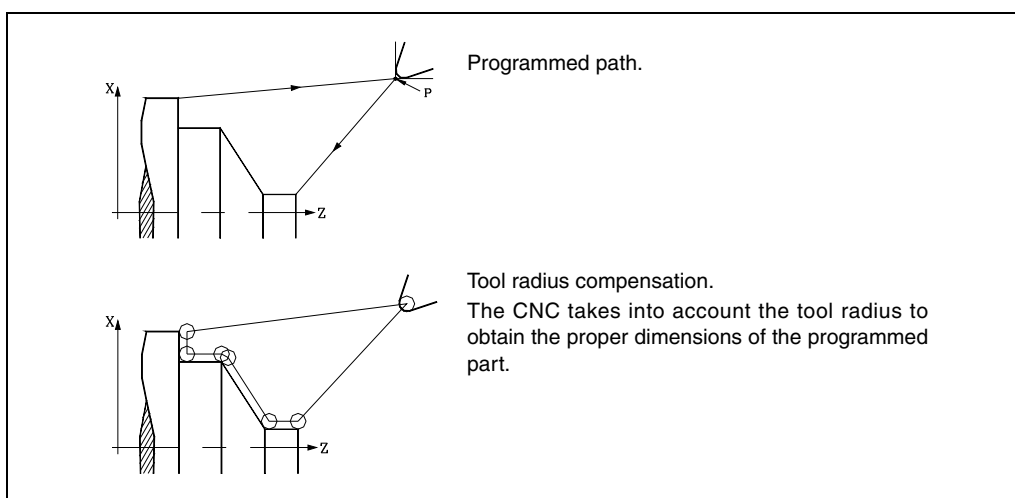
8.2 Tool radius compensation

It must be programmed. The CNC assumes as theoretical tip (P) the result of the sides used when calibrating the tool (left figure). Without radius compensation, the theoretical tip (P) travels the programmed path (center figure) leaving machining ridges.

With tool radius compensation, it takes into account the radius of the tip (nose) and the location code or tool type which results in the proper dimensions of the programmed part (right figure).

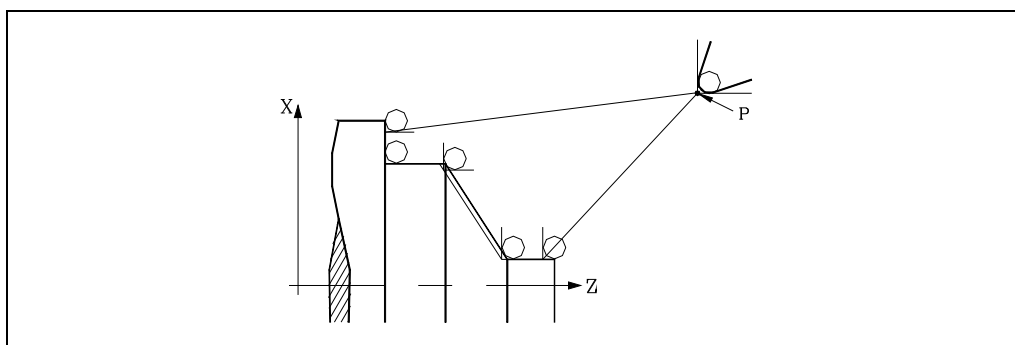


The CNC always shows the position of the theoretical tip. Therefore, when using tool radius compensation, the coordinates and the graphic representation do not always coincide with the programmed path.



The CNC does not show the tool center path; it shows the position of the theoretical tip.

The path of the theoretical tip coincides, in part, with the profile programmed in turning and facing, but never coincides with the incline or curved sections.

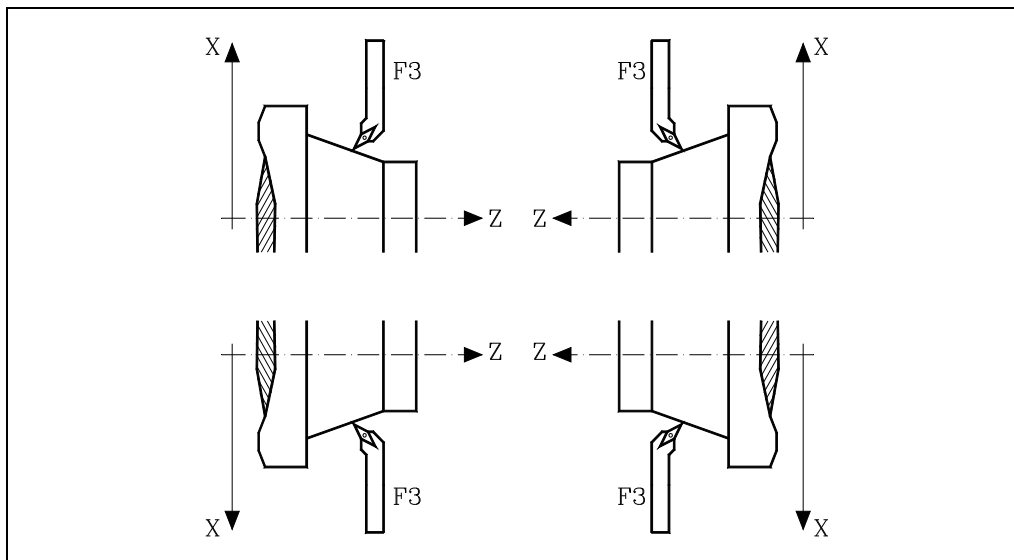


8.2.1 The location code of the tool (tool type)

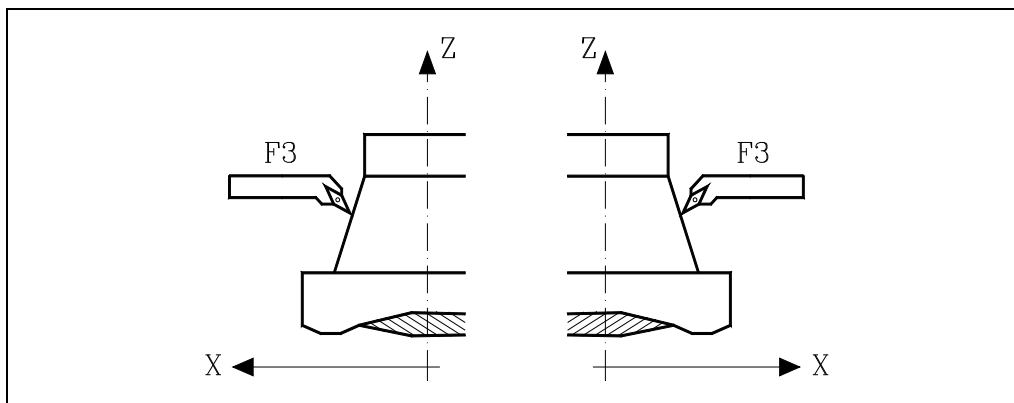
The location code indicates the tool type and the sides used to calibrate it. It depends on the position of the tool and the orientation of the axes on the machine.

The following example shows the location code F3 on different machines. Observe how the relative tool position is maintained with respect to the axes.

Horizontal lathes:



Vertical lathes:



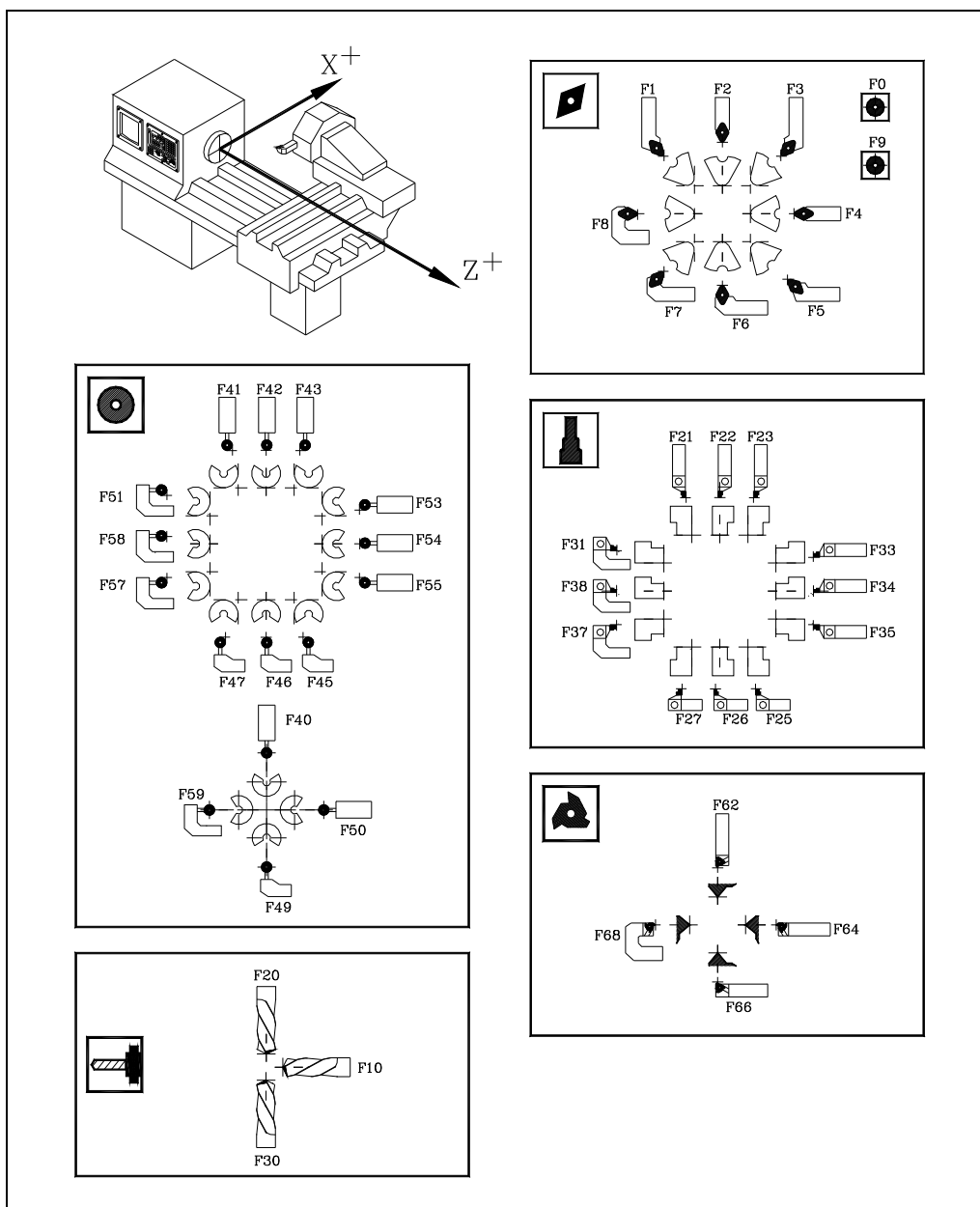
8.

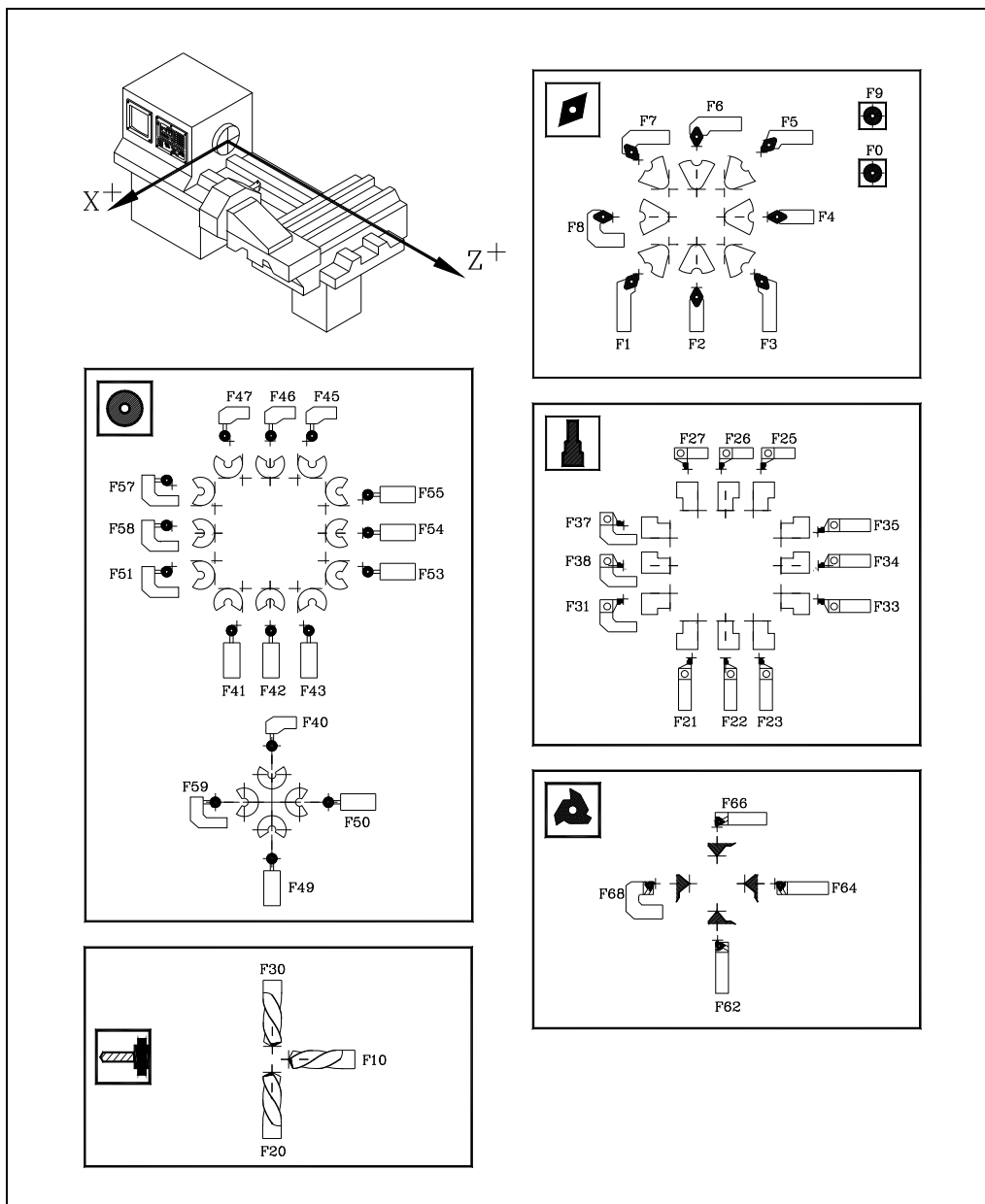
TOOL COMPENSATION
Tool radius compensation

We now show the location codes available on the most common horizontal lathes.

8.

TOOL COMPENSATION Tool radius compensation





8.

TOOL COMPENSATION
Tool radius compensation

FAGOR

CNC 8037

·T· MODEL
SOFT: V01.4x

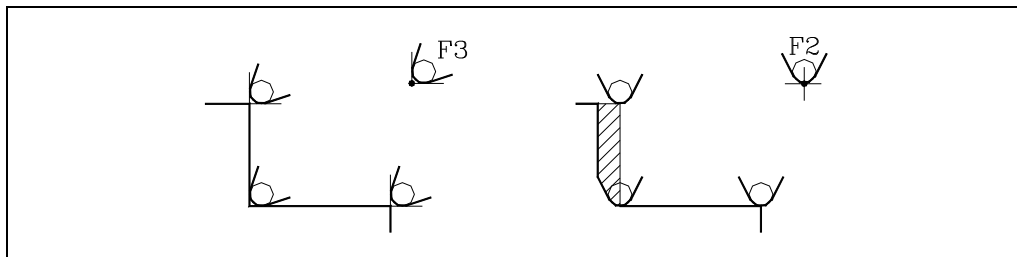
8.2.2 Working without tool radius compensation

There are certain limitations to work without tool radius compensation.

Tool location code (tool type).

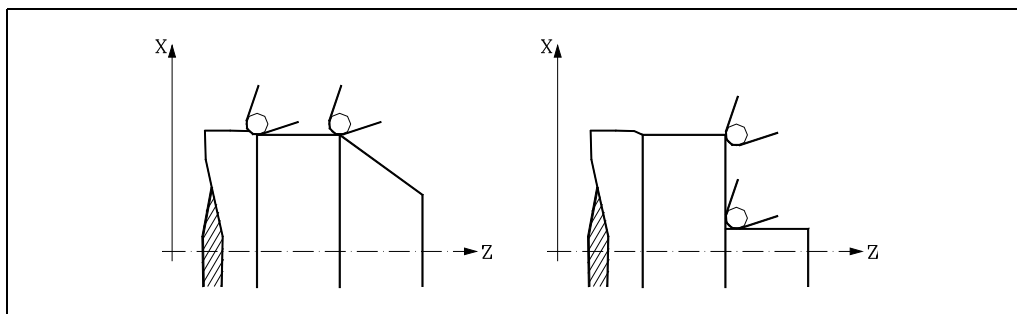
Only tools that have been calibrated by touching two of its sides may be used, location codes F1, F3, F5, F7, etc.

It is not recommend to machine with the rest of the tools because the theoretical tip of the tool travels along the programmed path (the shaded area of the right figure is eliminated).

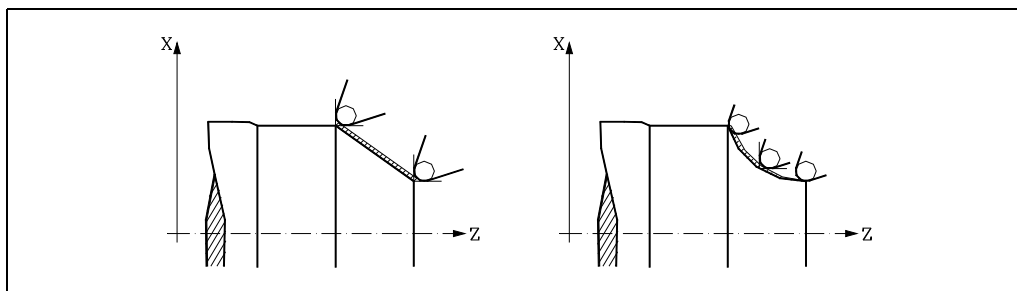


Machining sections.

Only sides with constant diameter may be turned (left figure) or straight walls may be faced (right figure).

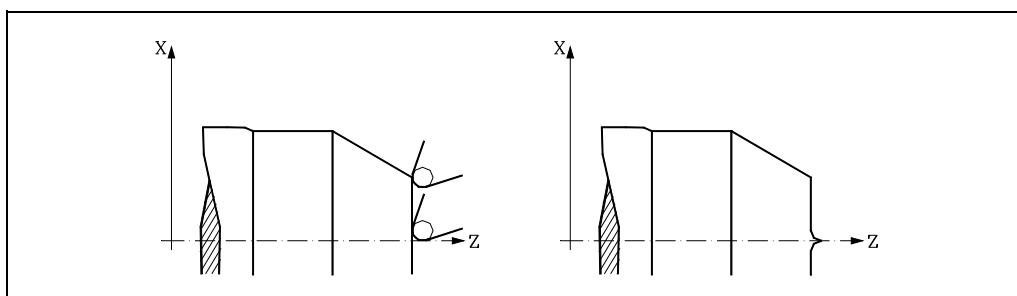


There are problems in incline sections (left figure) and on rounded sides (right figure).



Facing straight walls.

When facing to a 0 coordinate (for example from position 40 to 0) the theoretical tip of the tool reaches the 0 coordinate, but due to the tip's rounding, it leaves a ridge on the part. To solve this problem, face to a negative coordinate (for example from 40 to -3).



8.

TOOL COMPENSATION
Tool radius compensation



CNC 8037

·T· MODEL
SOFT: V01.4x

8.2.3 Working with tool radius compensation

When working with tool radius compensation, it takes into account the radius of the tip (nose) and the location code or tool type stored in the tool offset table that corresponds to the tool to obtain the proper dimensions of the programmed part.

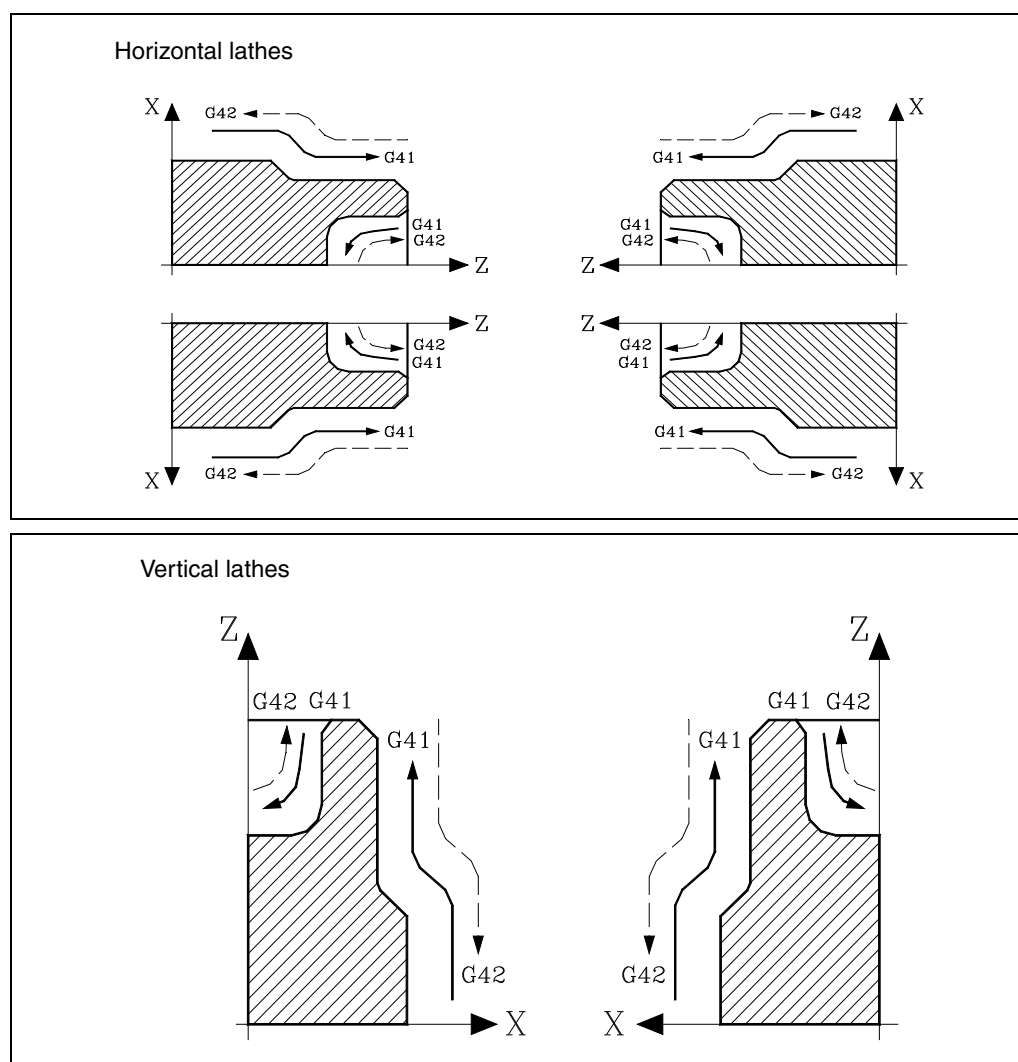
All the tools have an associated offset (in the tool table). Use the "D" code to select another offset. If no tool offset has been programmed, the CNC applies offset D0, with X=0, Z=0, F=0, R=0, I=0 and K=0.

Tool dimensions can also be defined with the variables TOX, TOZ, TOF, TOR, TOI, TOK.

There are three preparatory functions for tool radius compensation:

- G40 Cancellation of tool radius compensation
- G41 Tool radius compensation to the left of the part.
- G42 Tool radius compensation to the right of the part.

Functions G41 and G42 are modal and incompatible to each other. They are canceled by G40, G04 (interruption of block preparation), G53 (programming with reference to machine zero), G74 (home search), G66, G68, G69, G83 (machining canned cycles) and also on power-up, after executing M02, M30 or after an emergency or a reset.



8.

TOOL COMPENSATION
Tool radius compensation

FAGOR 

CNC 8037

·T· MODEL
SOFT: V01.4x

8.2.4 Beginning of tool radius compensation (G41, G42)

Once the plane in which tool radius compensation has been selected, functions G41 or G42 must be used to activate it.

G41 Tool radius compensation to the left of the part.

G42 Tool radius compensation to the right of the part.

In the same block (or a previous one) in which G41 or G42 is programmed, functions T, D, or only T must be programmed so that the tool offset value to be applied can be selected from the tool offset table. If no tool offset is selected, the CNC takes D0 with X0 Z0 F0 R0 I0 K0.

When the new selected tool has function M06 associated with it and this, in turn, has an associated subroutine, the CNC will first process the motion block of that subroutine as the starting block of the compensation.

If in that subroutine, a block is executed that contains function G53 (programming in machine coordinates), the previously selected G41 or G42 is temporarily canceled in that block.

The selection of tool radius compensation (G41 or G42) can only be made when functions G00 or G01 are active (straight-line movements). If the compensation is selected while G02 or G03 are active, the CNC will display the corresponding error message.

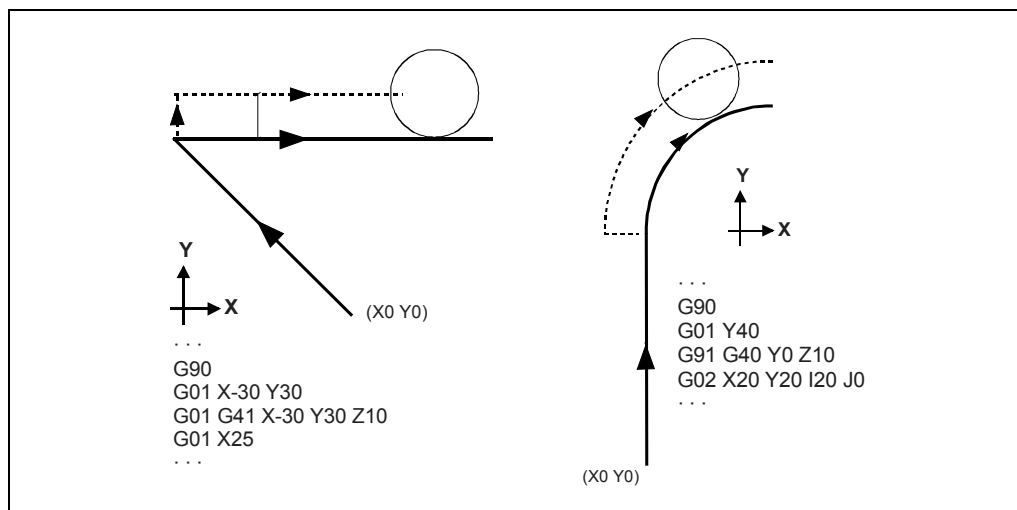
We now show different cases of starting tool radius compensation, in which the programmed path is represented by a thick line and the tool center path with a thin line.

Beginning of compensation without programming a movement

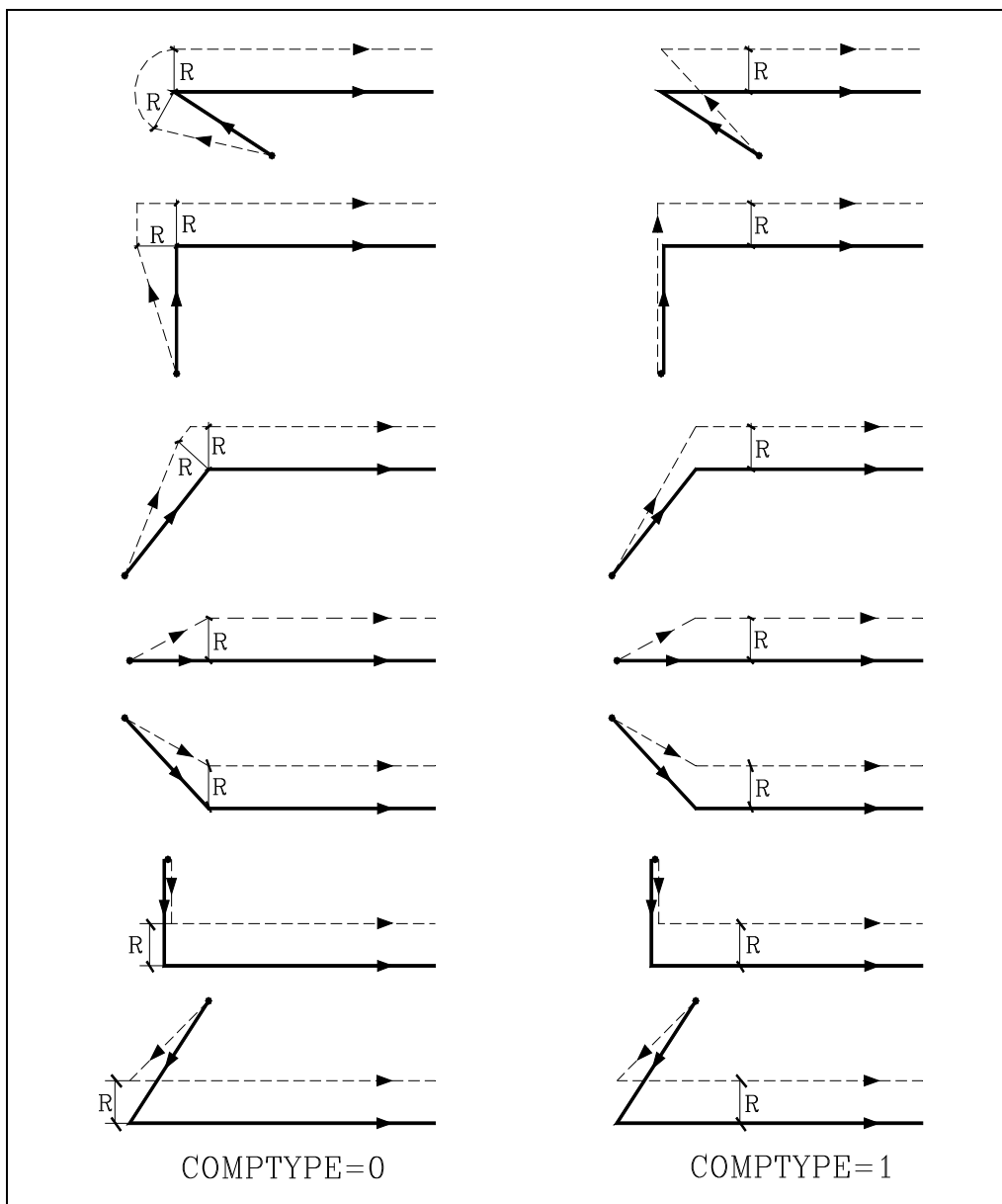
After activating the compensation, it could happen that the plane axes do not get involved in the first motion block either because they have not been programmed or because the same point as the tool position has been programmed or because a null incremental move has been programmed.

In this case, the compensation is applied in the current tool position; depending on the first movement programmed in the plane, the tool moves perpendicular to the path on its starting point.

The first movement programmed in the plane may be either linear or circular.



STRAIGHT-STRAIGHT path



8.

TOOL COMPENSATION
Tool radius compensation

FAGOR 

CNC 8037

·T· MODEL
SOFT: V01.4x

STRAIGHT-CURVED path

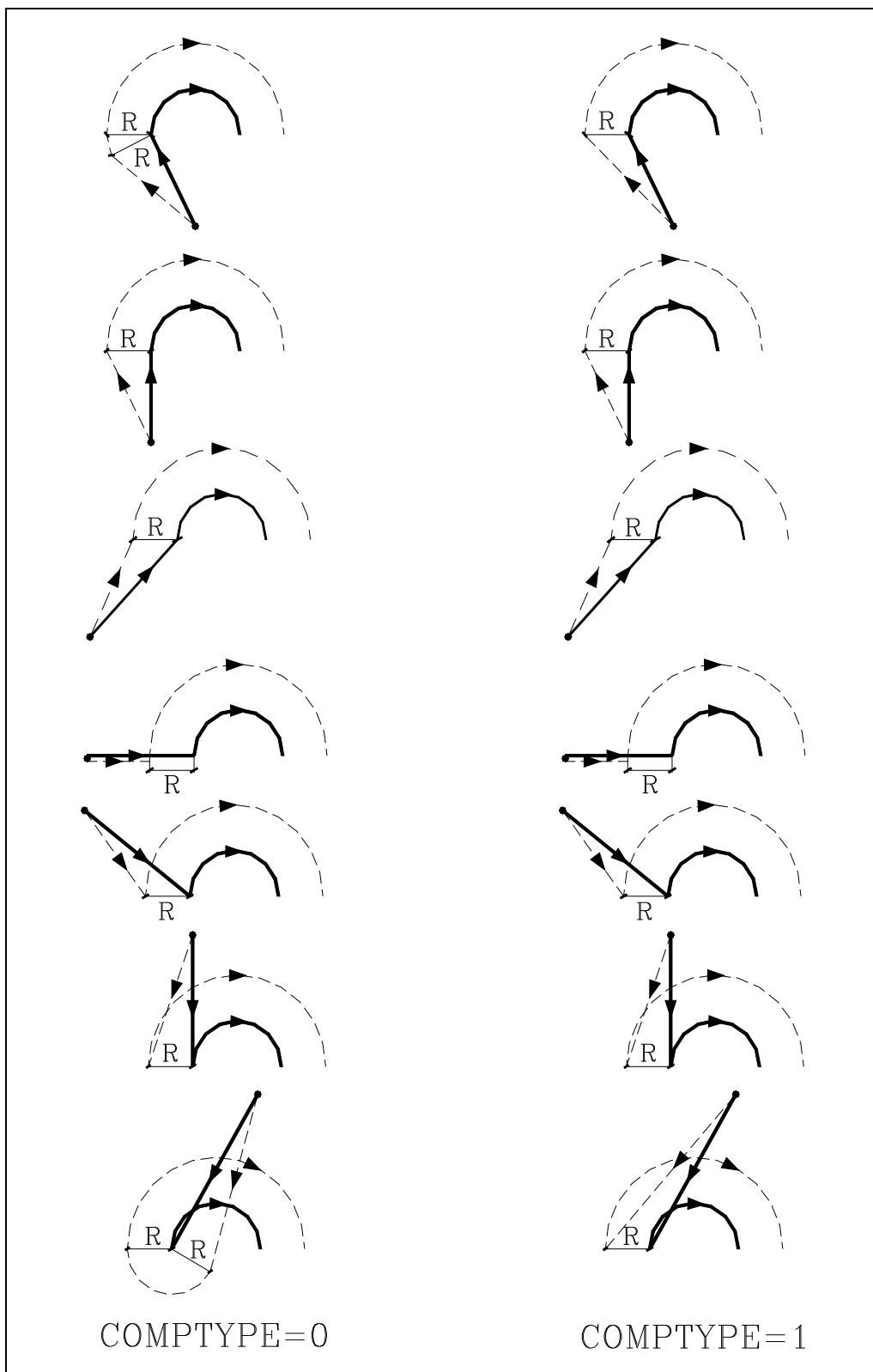
8.

TOOL COMPENSATION
Tool radius compensation



CNC 8037

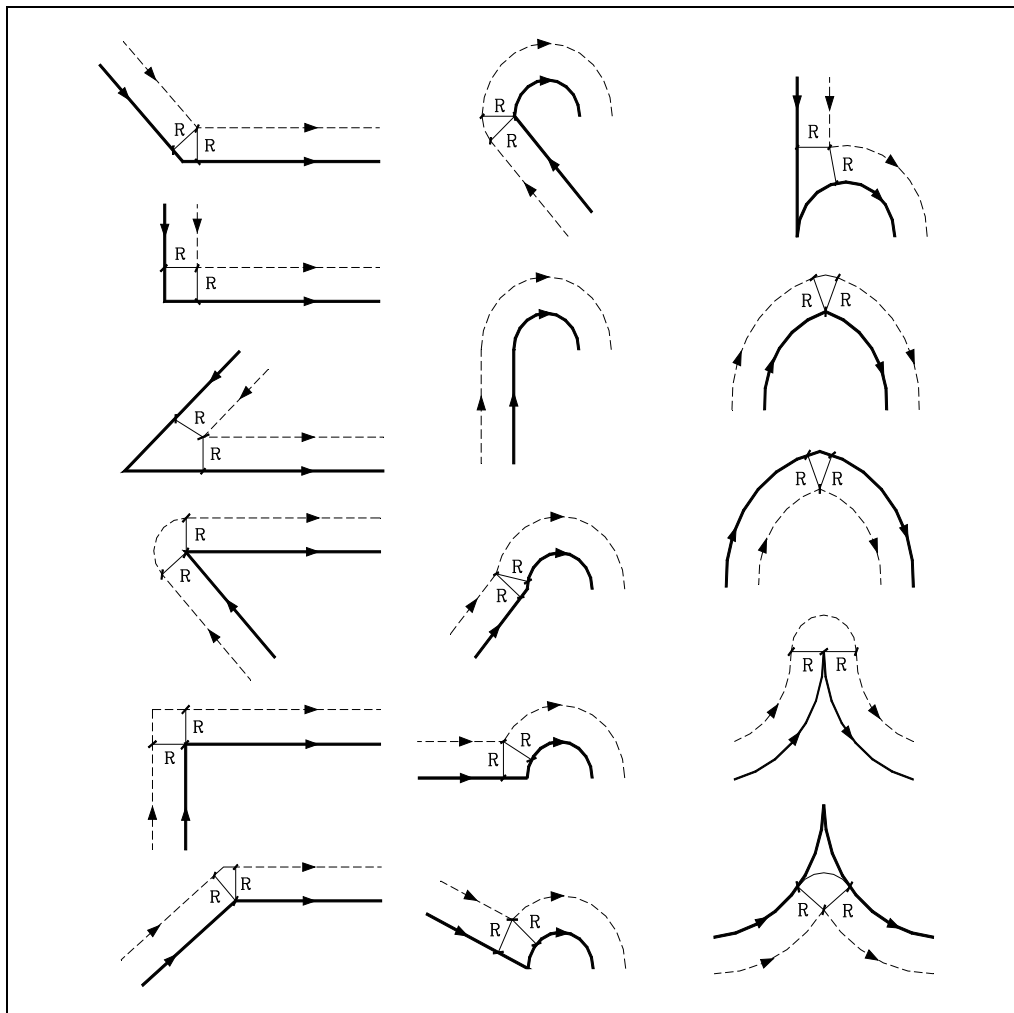
·T· MODEL
SOFT: V01.4x



8.2.5 Sections of tool radius compensation

The CNC reads up to 50 blocks ahead of the one it is executing, with the aim of calculating beforehand the path to be followed. When working with tool radius compensation, the CNC needs to know the next programmed movement to calculate the path to follow; therefore, no more than 48 blocks can be programmed in a row.

The diagrams (below) show the different paths followed by a tool controlled by a programmed CNC with tool radius compensation. The programmed path is represented by a thick line and the tool center path by a thin line.



The way the various paths are blended (joined) depends on the setting of machine parameter COMPMODE.

- If it is set to -0-, the compensation method depends on the angle between paths.
With an angle between paths of up to 300° , both paths are joined with straight sections. In the rest of the cases, both paths are joined with arcs.
- If it is set to -1-, both paths are joined with arcs.
- If it is set to -2-, the compensation method depends on the angle between paths.
With an angle between paths of up to 300° , it calculates the intersection. In the rest of the cases it is compensated like COMPMODE = 0.

8.

TOOL COMPENSATION
Tool radius compensation

FAGOR 

CNC 8037

·T· MODEL
SOFT: V01.4x

8.2.6 Cancellation of tool radius compensation (G40)

Tool radius compensation is canceled by using function G40.

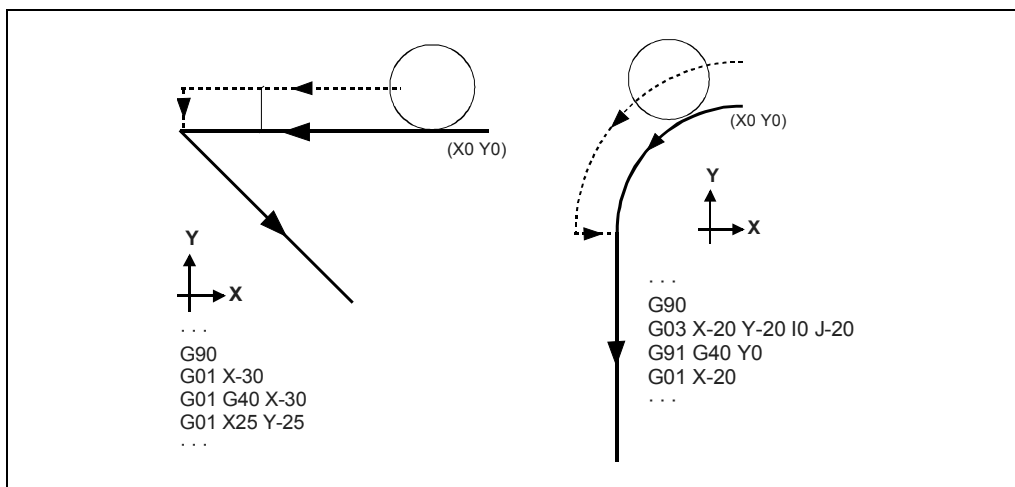
It should be remembered that canceling radius compensation (G40) can only be done in a block in which a straight-line movement is programmed (G00 or G01). If G40 is programmed while functions G02 or G03 are active, the CNC displays the corresponding error message.

We now show different cases of starting tool radius compensation, in which the programmed path is represented by a thick line and the tool center path with a thin line.

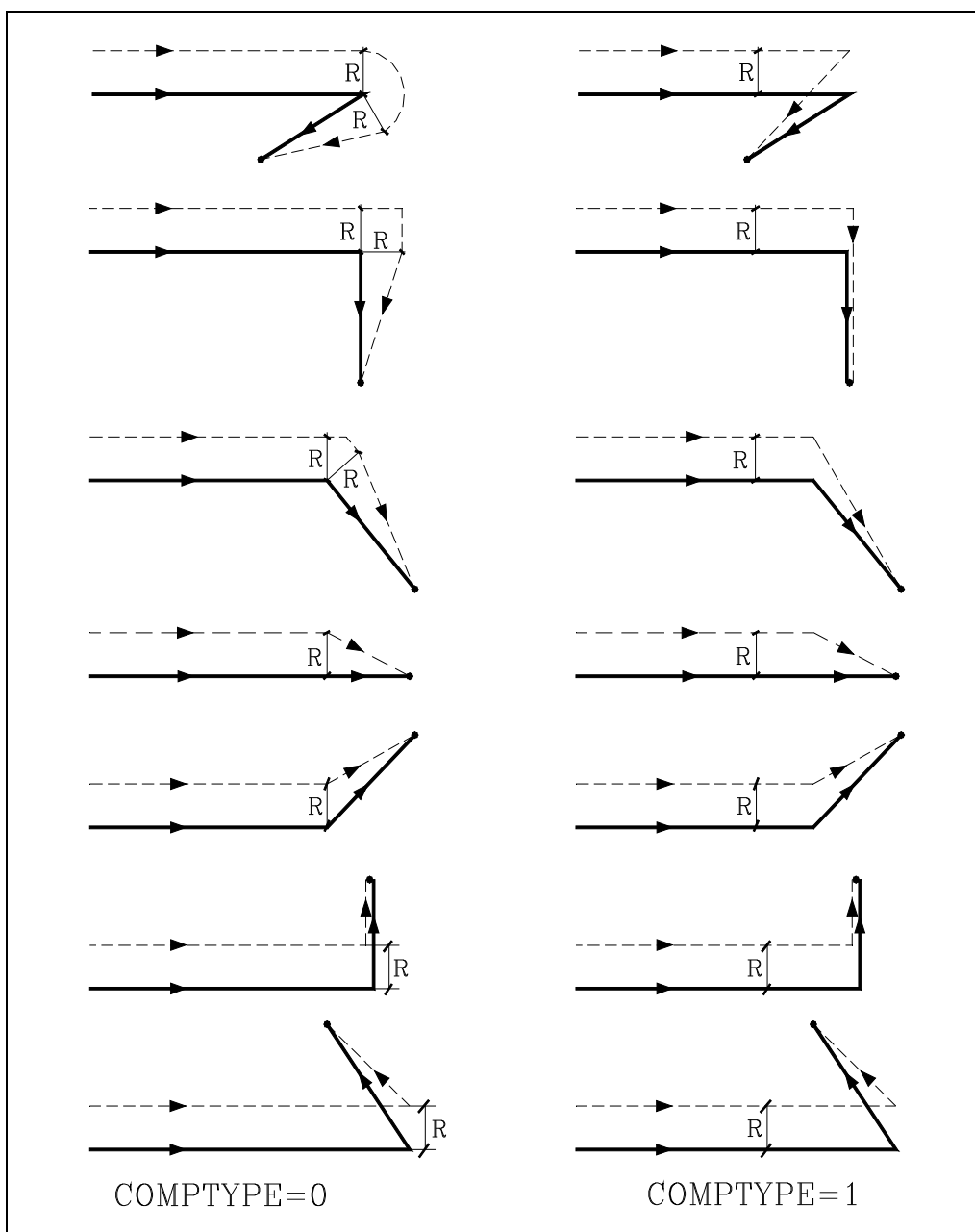
End of compensation without programming a movement

After cancelling the compensation, it could happen that the plane axes do not get involved in the first motion block either because they have not been programmed or because the same point as the tool position has been programmed or because a null incremental move has been programmed.

In this case, the compensation is canceled in the current tool position; depending on the last movement executed in the plane, the tool moves to the end point without compensating the programmed path.



STRAIGHT-STRAIGHT path



8.

TOOL COMPENSATION
Tool radius compensation

FAGOR 

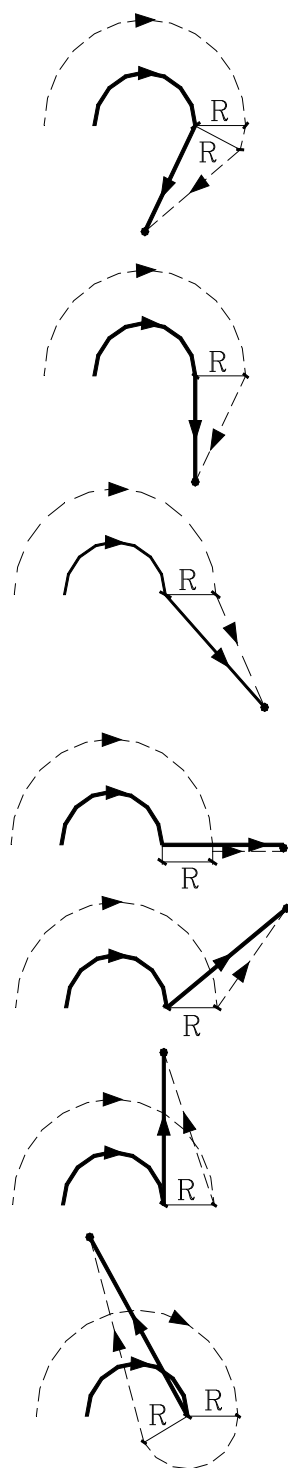
CNC 8037

·T· MODEL
SOFT: V01.4x

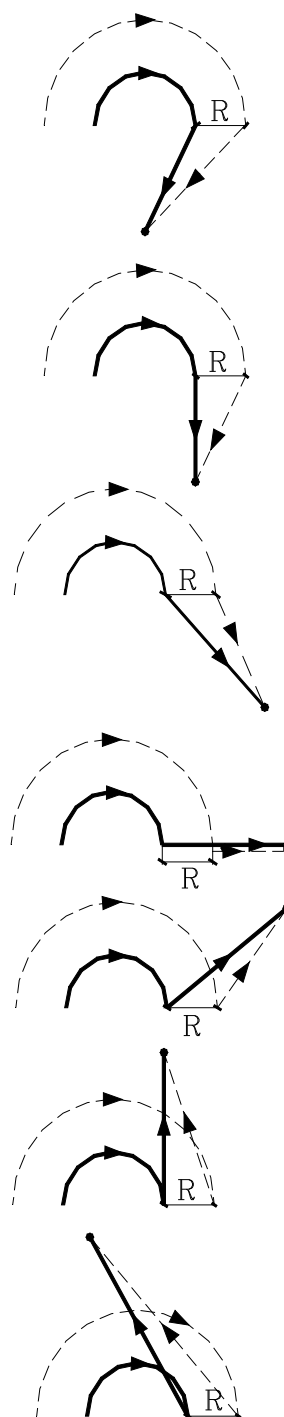
CURVED-STRAIGHT path

8.

TOOL COMPENSATION
Tool radius compensation



COMPTYPE=0



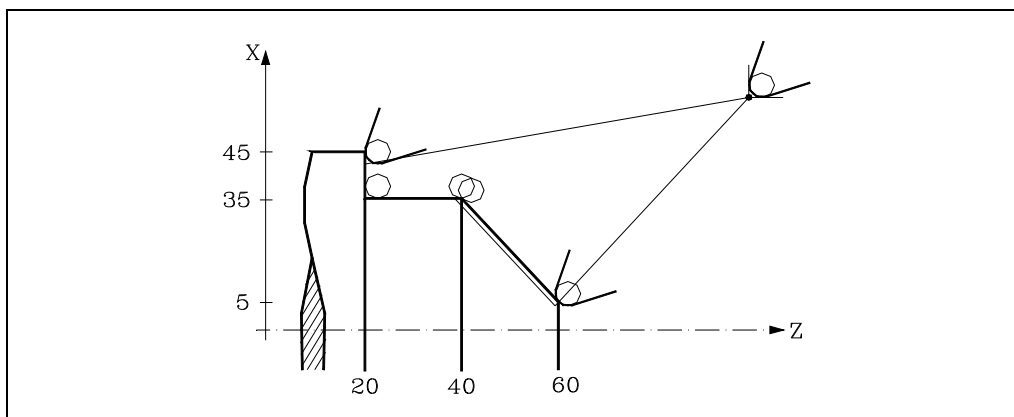
COMPTYPE=1



CNC 8037

·T· MODEL
SOFT: V01.4x

Programming example



T1 D1

G0 G90 X110 Z100 Positioning at the starting point.

G1 G42 X10 Z60 It activates the compensation and moves to the starting point.

X70 Z40

X70 Z20

X90 Z20 Movement to end point (compensation active).

G40 X110 Z100 It cancels the compensation and moves to the starting point.

8.

TOOL COMPENSATION
Tool radius compensation

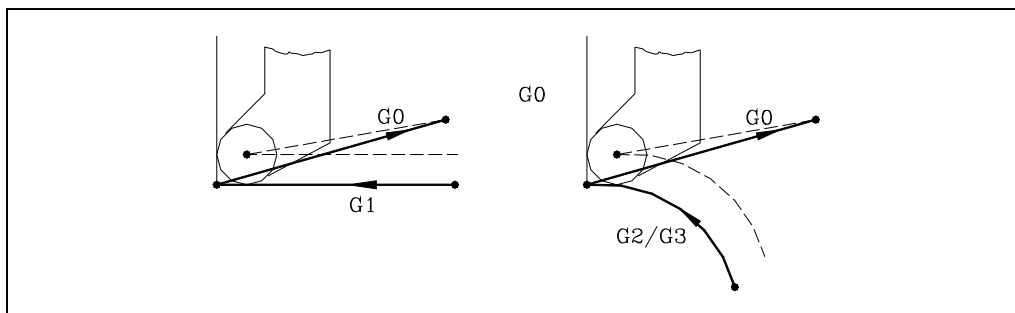
FAGOR 

CNC 8037

· T· MODEL
SOFT: V01.4x

8.2.7 Temporary cancellation of tool compensation with G00

When detecting a change from G01, G02, G03, G33 to G00, the CNC temporarily cancels tool radius compensation and keeps the tool tangent to the perpendicular at the end of the movement programmed in the block containing G01, G02, G03, G33 or G34.

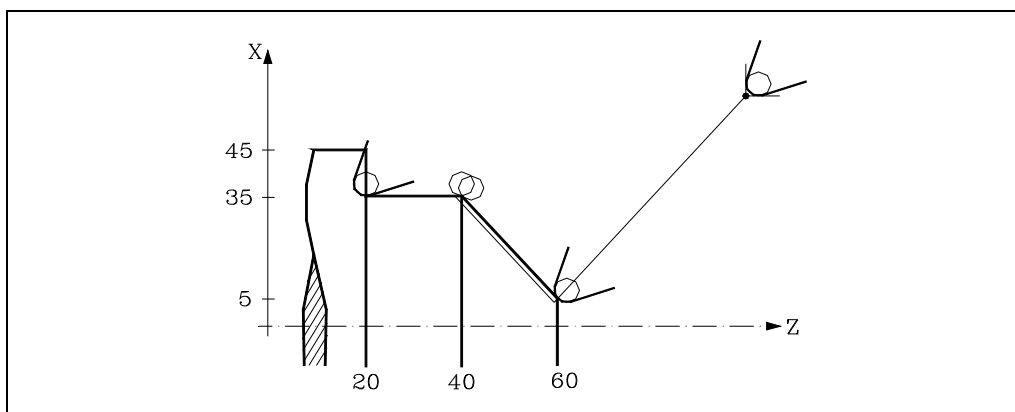


When detecting a change from G00 to G01, G02, G03, G33 or G34, the new block is treated like the first compensated point resuming tool radius compensation.

Special case: If the CNC does not have enough information for compensating, but the movement is in G00, it will execute it without tool radius compensation.

Programming example

Wrong programming example. Compensation is canceled in the last block of the profile and the machining does not coincide with the desired one because the CNC compensates the whole defined section. When compensating the last section, the tool runs into the faced side.



T1 D1

G0 G90 X110 Z100 Positioning at the starting point

G1 G42 X10 Z60 It activates the compensation and moves to the starting point

X70 Z40

X70 Z20

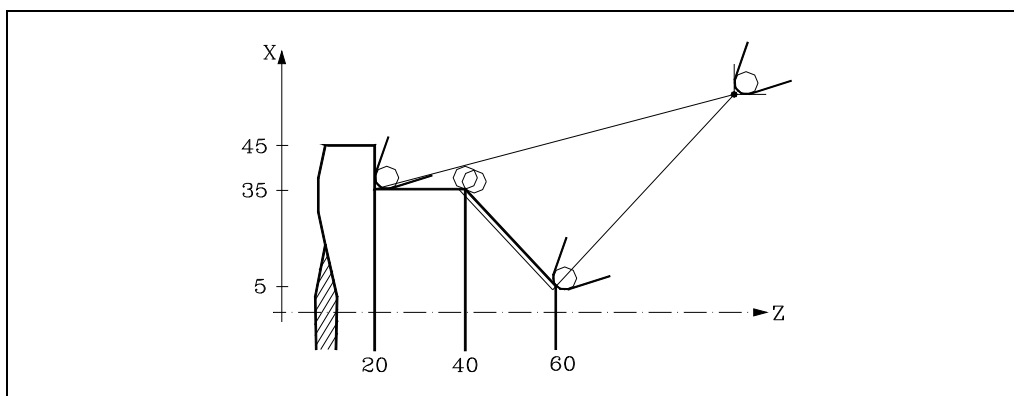
G40 X110 Z100 It cancels the compensation and moves to the starting point



CNC 8037

·T· MODEL
SOFT: V01.4x

This problem is solved using function G00 as described next.



T1 D1

G0 G90 X110 Z100 Positioning at the starting point

G1 G42 X10 Z60 It activates the compensation and moves to the starting point

X70 Z40

X70 Z20

G40 G0 X110 Z100 It cancels the compensation and moves to the starting point

8.

TOOL COMPENSATION
Tool radius compensation

FAGOR 

CNC 8037

·T· MODEL
SOFT: V01.4x

8.2.8 Change of type of tool radius compensation while machining

The compensation may be changed from G41 to G42 or vice versa without having to cancel it with G40. It may be changed in any motion block and even in a motionless one, i.e. without moving the axes of the plane or by programming the same point twice.

It compensates independently the last movement before the change and the first one after the change. To change the type of compensation, the different cases are solved according to these criteria:

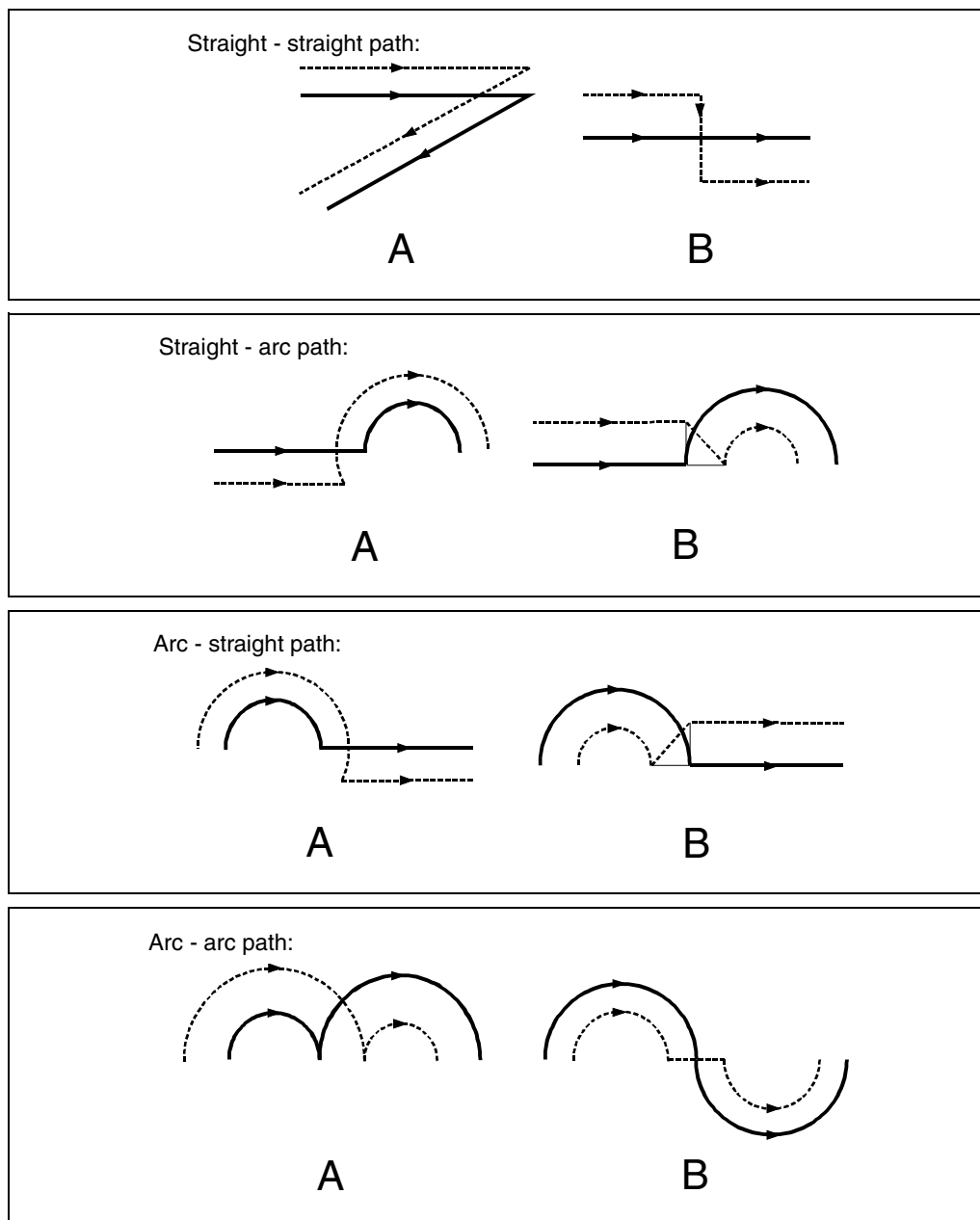
A. The compensated paths cut each other.

The programmed paths are compensated each on its corresponding side. The change of sides takes place at the intersection point of both paths.

B. The compensated paths do not cut each other.

An additional section is inserted between the two paths. From the point perpendicular to the first path at the end point up to the point perpendicular to the second path at the starting point. Both points are located at a distance R from the programmed path.

Here is a summary of the different cases:



8.2.9 Tool compensation in any plane

With general machine parameter "PLACOM", it is possible to use tool compensation either in all planes or just in the ZX plane. When "PLACOM = 1" to apply tool compensation in all planes, the CNC interprets the tool table as follows:

	ZX plane	WX plane	AB plane
Parameters Z and K. Abscissa axis.	Z axis	W axis	A axis
Parameters X and I. Ordinate axis.	X axis	X axis	B axis

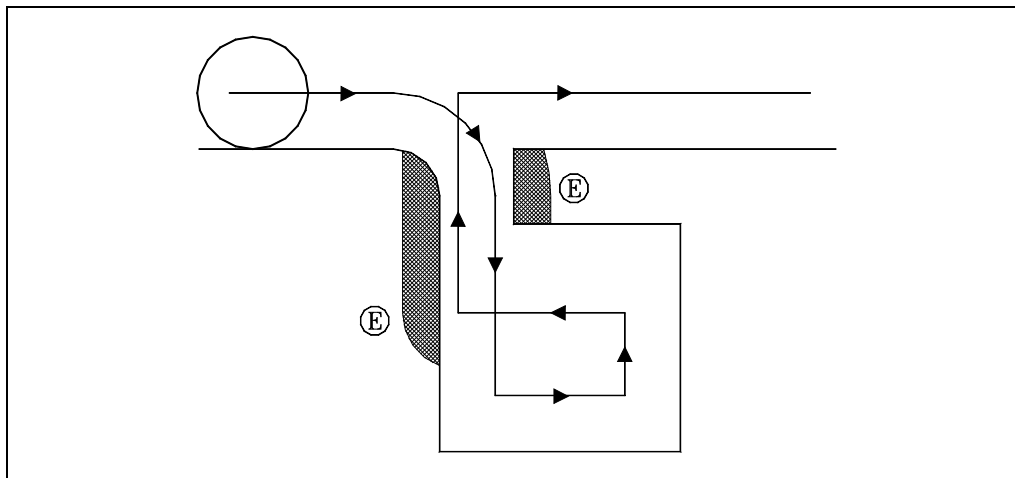
8.

TOOL COMPENSATION
Tool radius compensation

8.3 Collision detection (G41 N, G42 N)

Using this option, the CNC analyzes in advance the blocks to be executed in order to detect loops (profile intersections with itself) or collisions of the programmed profile. The number of blocks to be analyzed (up to 50) may be defined by the user.

The example shows machining errors (E) due to a collision in the programmed profile. This type of errors may be avoided using collision detection.



When detecting a loop or a collision, the blocks that caused it will not be executed and a warning will be issued for each loop or collision eliminated.

Possible cases: a step on a straight path, a step in a circular path and tool radius compensation too large.

The information contained in the eliminated blocks, not being the moving in the active plane, will be executed (including the movements of the axes).

Block detection is defined and activated with tool radius compensation functions G41 and G42. A new parameter N (G41 N y G42 N) has been added to activate the feature and define the number of blocks to analyze.

Possible values from N3 to N50. Without "N", or with N0, N1 and N2, it behaves like in older versions.

In CAD generated programs that are made up of lots of very short blocks, it is recommended to use very low N values (around 5) so as not to jeopardize block processing time.

When this function is active, the history of active G functions shows G41 N or G42 N.

8.

TOOL COMPENSATION
Collision detection (G41 N, G42 N)



CNC 8037

·T· MODEL
SOFT: V01.4x

This CNC offers the following machining canned cycles:

G66	Pattern repeat canned cycle.
G68	Stock removal cycle along X axis.
G69	Stock removal canned cycle along Z axis.
G81	Turning canned cycle with straight sections.
G82	Facing canned cycle with straight sections.
G83	Drilling canned cycle.
G84	Turning canned cycle with arcs.
G85	Facing canned cycle with arcs.
G86	Longitudinal threading canned cycle.
G87	Face threading canned cycle.
G88	Grooving canned cycle along the X axis.
G89	Z axis grooving canned cycle.

A canned cycle is defined by the G function indicating the canned cycle and its corresponding parameters. A canned cycle may be defined anywhere in the program, that is, in the main program as well as in a subroutine.

When working in a plane other than the ZX, for example: G16 WX, the CNC interprets the canned cycle parameters as follows:

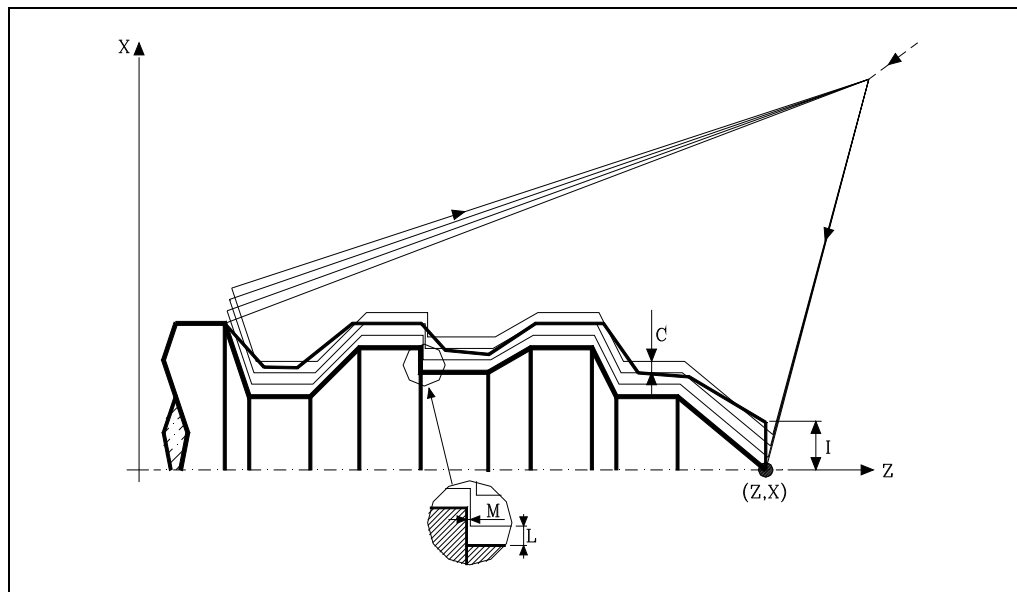
	ZX plane	WX plane	AB plane
Parameter Z and all those related to it, with the abscissa axis.	Z axis	W axis	A axis
Parameter Z and all those related to it, with the ordinate axis.	X axis	X axis	B axis

9.1 G66. Pattern repeat cycle canned cycle

This cycle machines the programmed profile maintaining the indicated step between consecutive machining passes. Triangular, round and square tools may be used in this cycle.

The basic structure of the block is:

G66 X Z I C A L M H S E Q



X±5.5

It sets the X coordinate of the profile's starting point. It must be programmed in absolute coordinates and in the active units, radius or diameter.

Z±5.5

It sets the Z coordinate of the profile's starting point. It must be programmed in absolute coordinates.

I5.5

It defines the excess stock to be removed from the original part. It is defined in radius and depending on the value assigned to parameter "A", this value will be interpreted as excess stock in X or in Z.

If its value is not greater than the finishing stock (L or M), it will only run the finishing pass, if H is other than zero.

C5.5

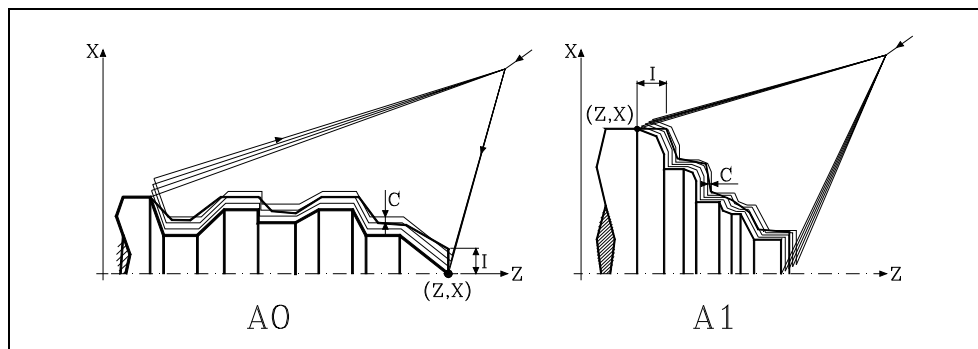
It defines the machining pass. All the machining passes will be run with this step, except the last one that will remove the residual stock.

It is defined in radius and depending on the value assigned to parameter "A", this value will be interpreted, Like "I", as excess stock in X or in Z. If programmed with a 0 value, the CNC will display the corresponding error message.

A1

It defines the main machining axis.

- When programming A0, Z will be the main axis. The "I" value is assumed as excess material in X and the "C" value as a pass in X.
- When programming "A1", the X axis will be the main axis, "I" will be taken as Z residual value and "C" as the pass along Z.



When not programming the A parameter, the "I" and "C" values depend on the tool dimensions..

- If the X length of the tool is greater than its Z length, the "I" value is assumed as excess material in X and the "C" value as a pass in X.
- If the X length of the tool is smaller than its Z length, the "I" value is assumed as excess material in Z and the "C" value as a pass in Z.

L±5.5

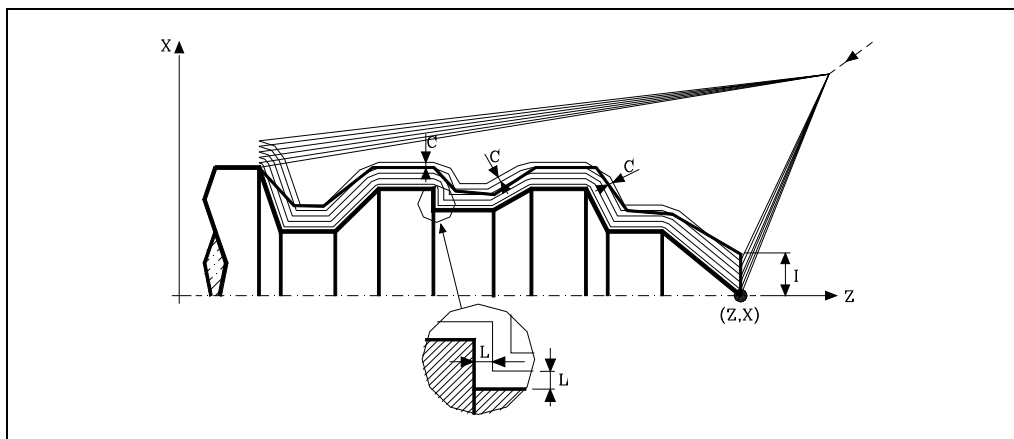
It defines the finishing stock in X. It is defined in radius and if not programmed, it assumes a value of "0".

M±5.5

It defines the finishing stock in Z.

If "L" or "M" is programmed with a negative value, the finishing pass is carried out in round corner mode (G05). When both parameters are programmed with a positive value, the finishing pass is carried out in square corner mode (G07).

If the parameter "M" is not programmed, the residual stock in X and Z will be the one indicated in parameter "L" and the roughing passes will be identical keeping the "C" distance between 2 consecutive passes.



H5.5

It defines the feedrate during the finishing pass.

If not programmed or programmed with a 0 value, it does not run the finishing pass.

S4

Defines the label number of the block where the geometrical description of the profile of the part starts.

E4

Defines the label number of the block where the geometrical description of the profile of the part ends.

Q6

Defines the number of the program that contains the geometrical description of the profile.

9.

CANNED CYCLES
G66. Pattern repeat cycle canned cycle

FAGOR

CNC 8037

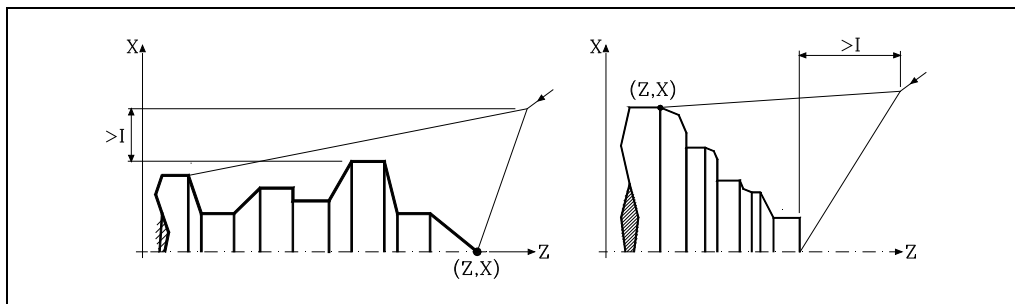
·T· MODEL
SOFT: V01.4x

This parameter is optional and if it is not defined, the CNC assumes that the profile is defined in the same program that is calling the cycle.

Considerations

The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the cycle.

The canned cycle calling point must be off the part to be machined and at a distance greater than the one defined as excess stock (I) of the most external profile of the part.

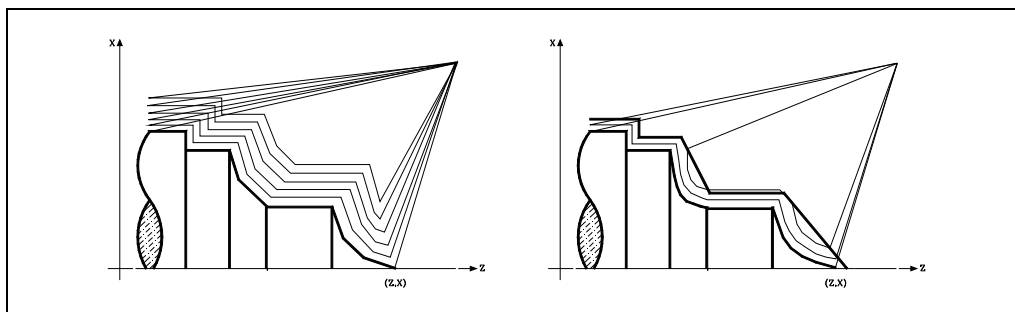


If the tool position is not correct to execute the cycle, the CNC will display the corresponding error.

Once the canned cycle has ended, the active feedrate will be the last one programmed, i.e. the one corresponding to the roughing operation (F) or to the finishing operation (H). The CNC will assume functions G00, G40 and G90.

Optimizing the machining operation.

If only the desired profile is defined, the CNC assumes that the original stock is cylindrical and machines it as shown on the left.

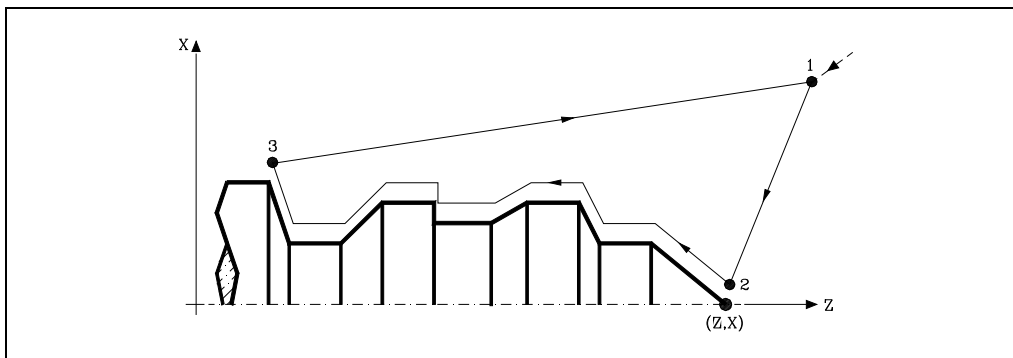


When the stock profile is known, it is recommended to define both profiles: The profile of the original stock and the desired final profile. The machining operation is much faster because only the stock delimited by both profiles is removed.

See ["9.1.2 Profile programming syntax"](#) on page 139.

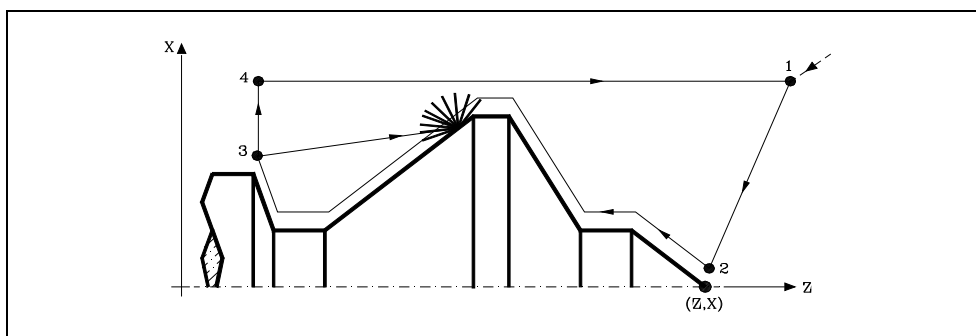
9.1.1 Basic operation

Each pass is carried out as follows:



1. The approach movement "1-2" is made in rapid traverse (G00).
2. The movement "2-3" is made at programmed feedrate (F).
3. The withdrawal movement "3-1" is made in rapid traverse (G00).

If there is a chance of colliding with the part, this movement will consist of two movements in G00 ("3-4" and "4-1") as shown in the following figure.



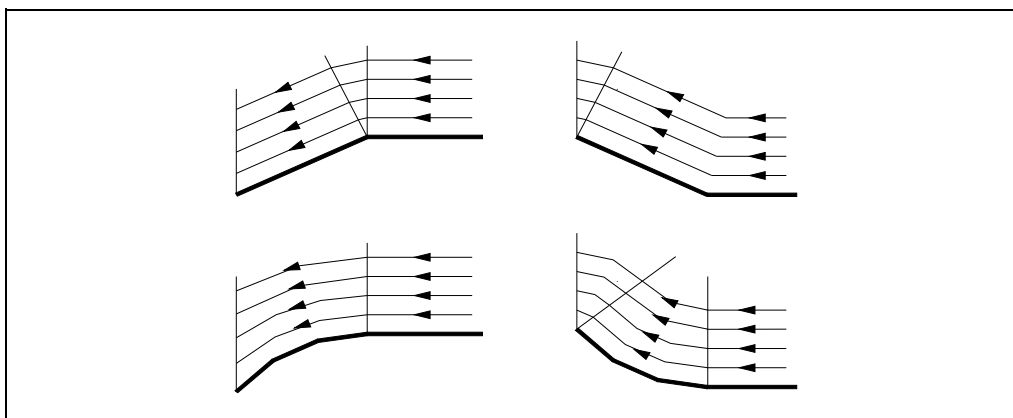
4. The canned cycle will always end at the point where the cycle was called from.

The machining passes

Once the profile to be executed is calculated, the CNC calculates all the necessary passes to remove the programmed excess stock (I).

The machining operation will be performed keeping the square corner mode (G07) or round corner mode (G05) that is active when calling the cycle.

If the parameter "M" is not programmed, the passes will be identical keeping the "C" distance between 2 consecutive passes. Also, if the last section of the profile is a curved section or an inclined plane, the CNC will calculate the different passes without exceeding the programmed maximum coordinate.



9.

CANNED CYCLES
G66. Pattern repeat cycle canned cycle

FAGOR 

CNC 8037

·T· MODEL
SOFT: V01.4x

9.

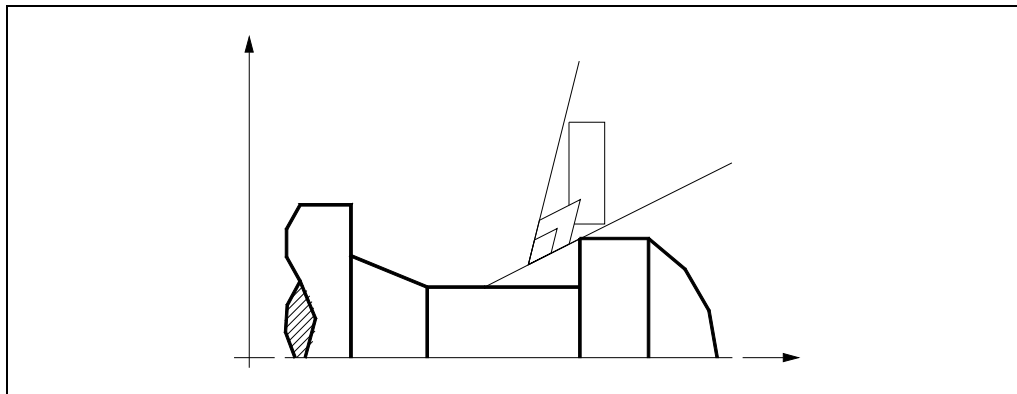
CANNED CYCLES

G66. Pattern repeat cycle canned cycle

The profile and the tool

After analyzing the programmed profile and depending on the tool used, it will execute that profile or the one closest to it if the one programmed cannot be executed. When the programmed profile (valleys) cannot be executed with the selected tool, a message will be displayed at the beginning of the execution of the cycle.

The operator will have the choice to stop the execution and select the right tool. If he doesn't do that, the CNC calculates a new profile in the areas that cannot be accessed by the selected tool and it machines as much as possible. The message is shown during the whole machining operation.



9.1.2 Profile programming syntax

When defining the profile, there is no need to program the starting point because it is specified by the cycle defining parameters X and Z.

When defining 2 profiles, first define the final profile and then that of the original stock.

The first and last profile defining blocks (where the profile or profiles end) must have the block label number. These label numbers will indicate the beginning and end of the profile geometry definition.

The profile programming syntax must follow these rules:

- It may be programmed in absolute or incremental coordinates and may be made up of simple geometrical elements such as straight lines, arcs, rounded corners, chamfered corners, etc. following the syntax rules defined for this purpose.
- G00 indicates the end of the definition of the final profile and that in this block contains the beginning of the definition of the profile of the original stock.
Program G01, G02 or G03 in the next block, because G00 is modal, thus preventing the CNC from issuing the corresponding error message.
- The profile description must not contain mirror images, scaling factor changes, pattern rotation or zero offsets.
- It must not contain high level blocks such as jumps, calls to subroutines or parametric programming.
- It must not contain other canned cycles.

The following functions may be used to define the profile:

G01	Linear interpolation.
G02	Clockwise circular interpolation.
G03	Counterclockwise circular interpolation.
G06	Circle center in absolute coordinates.
G08	Arc tangent to previous path.
G09	Three point arc.
G36	Automatic radius blend.
G39	Chamfer.
G53	Programming with respect to home.
G70	Programming in inches.
G71	Programming in millimeters.
G90	Absolute programming.
G91	Incremental programming.
G93	Polar origin preset.

The following functions may be programmed, although the cycle will ignore them.

G05	Round corner.
G07	Square corner.
G50	Controlled corner rounding.

F S T D or M functions.

9.

CANNED CYCLES

G66. Pattern repeat cycle canned cycle

FAGOR 

CNC 8037

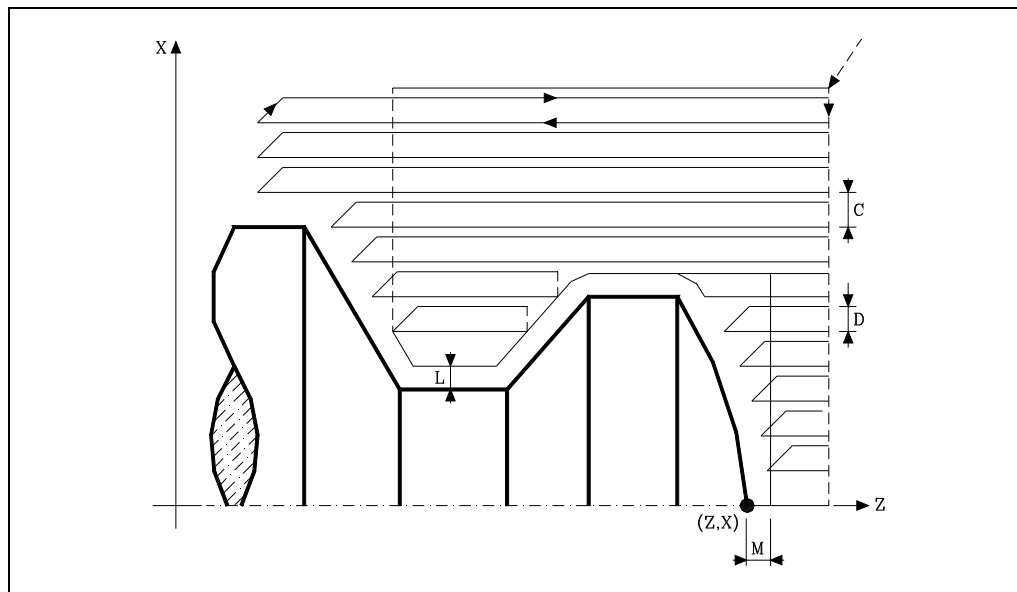
· T· MODEL
SOFT· V01.4x

9.2 G68. X axis roughing canned cycle

This cycle machines the programmed profile maintaining the indicated step between consecutive machining passes. Triangular, round and square tools may be used in this cycle.

The basic structure of the block is:

G68 X Z C D L M K F H S E Q



X±5.5

It sets the X coordinate of the profile's starting point. It must be programmed in absolute coordinates and in the active units, radius or diameter.

Z±5.5

It sets the Z coordinate of the profile's starting point. It must be programmed in absolute coordinates.

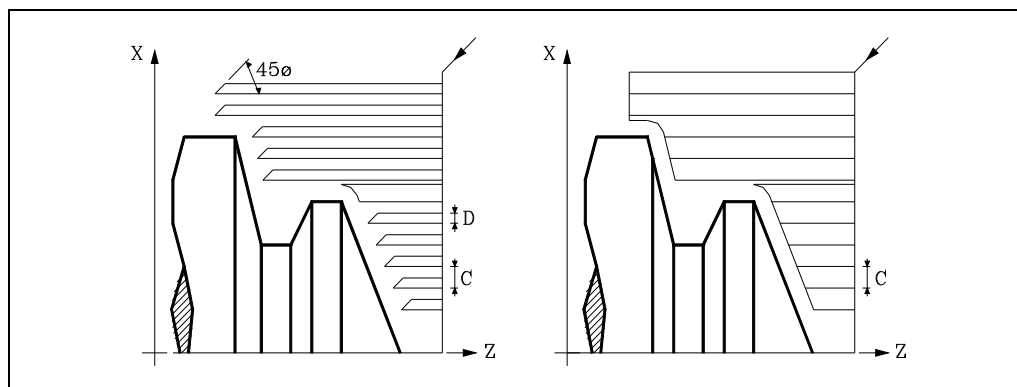
C5.5

It sets the machining pass and is programmed with a positive value given in radius. If programmed with a 0 value, the CNC will display the corresponding error message.

All the machining passes will be run with this step, except the last one that will remove the residual stock.

D5.5

It sets the safety distance the tool withdraws after each pass.



When programming D with a value other than 0, the cutter withdraws at 45° until reaching the safety distance (left figure).

When programming D with a 0 value, the exit path is the same as the entry path. This may be useful to groove complex profiles, to use these cycle on cylindrical grinders, etc.

9.

CANNED CYCLES
G68. X axis roughing canned cycle



CNC 8037

·T· MODEL
SOFT: V01.4x

If the parameter "D" is not programmed, the tool withdraws following the profile up to the previous pass, "C" distance (right figure).

It must be borne in mind that when not programming the parameter D, the cycle execution time is greater but the finishing stock is smaller.

L±5.5

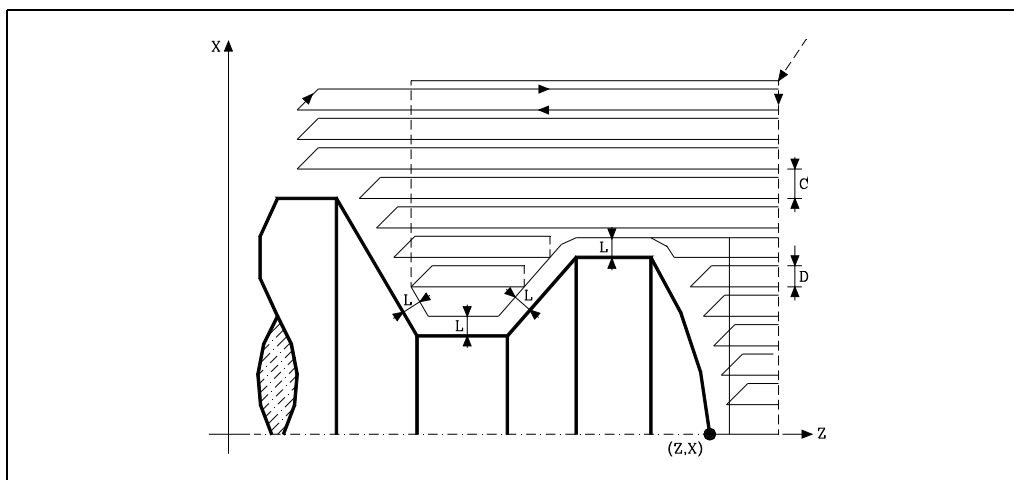
It defines the finishing stock in X. It is defined in radius and if not programmed, it assumes a value of "0".

M±5.5

It defines the finishing stock in Z.

If "L" or "M" is programmed with a negative value, the finishing pass is carried out in round corner mode (G05). When both parameters are programmed with a positive value, the finishing pass is carried out in square corner mode (G07).

If the parameter "M" is not programmed, the residual stock indicated in the parameter "L" will be constant throughout the whole profile.

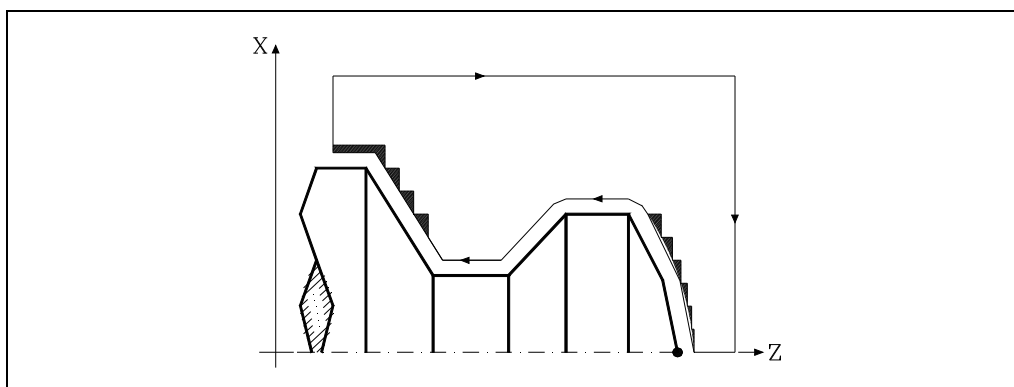


K5.5

It defines the tool feedrate to penetrate the valleys. If not programmed or programmed with a 0 value, it assumes the machining feedrate (the one programmed before calling the cycle).

F5.5

It defines the feedrate for the last roughing pass. If not programmed or programmed with a 0 value, it does not run the last roughing pass.



H5.5

It defines the feedrate during the finishing pass. If not programmed or programmed with a 0 value, it does not run the finishing pass.

S4

Defines the label number of the block where the geometrical description of the profile of the part starts.

9.

CANNED CYCLES
G68. X axis roughing canned cycle

FAGOR

CNC 8037

·T· MODEL
SOFT: V01.4x

E4

Defines the label number of the block where the geometrical description of the profile of the part ends.

Q6

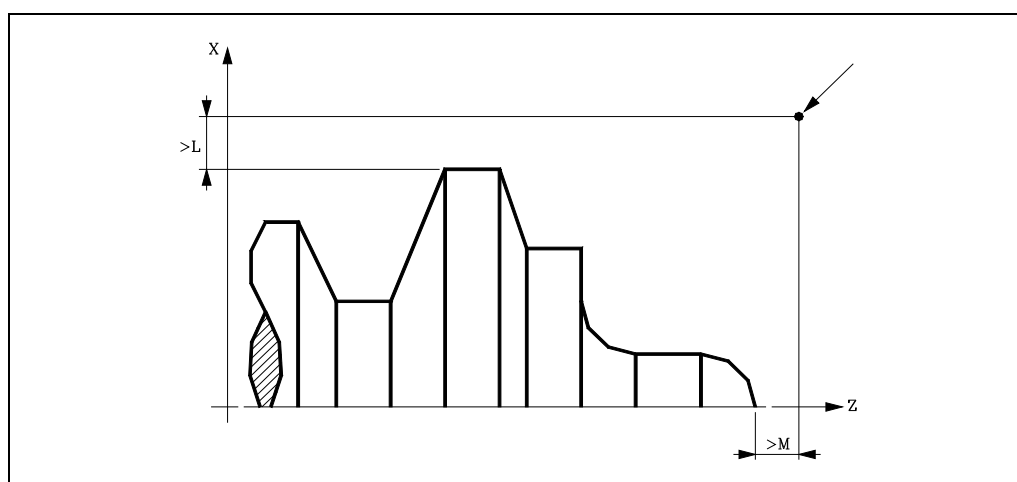
Defines the number of the program that contains the geometrical description of the profile.

This parameter is optional and if it is not defined, the CNC assumes that the profile is defined in the same program that is calling the cycle.

Considerations

The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the cycle.

The canned cycle calling point must be off the part to be machined and at a distance greater than the one defined as excess stock (L, M) along the X and Z axes.

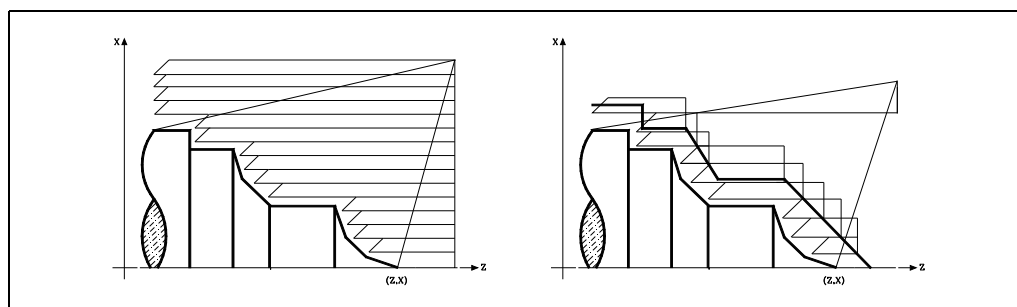


If the tool position is not correct to execute the cycle, the CNC will display the corresponding error.

Once the canned cycle has ended, the active feedrate will be the last one programmed, i.e. the one corresponding to the roughing operation (F) or to the finishing operation (H). The CNC will assume functions G00, G40 and G90.

Optimizing the machining operation.

If only the desired profile is defined, the CNC assumes that the original stock is cylindrical and machines it as shown on the left.



When the stock profile is known, it is recommended to define both profiles: The profile of the original stock and the desired final profile. The machining operation is much faster because only the stock delimited by both profiles is removed.

See ["9.2.2 Profile programming syntax"](#) on page 146.

9.

CANNED CYCLES
G68. X axis roughing canned cycle



CNC 8037

·T· MODEL
SOFT: V01.4x

9.2.1 Basic operation

The machining passes

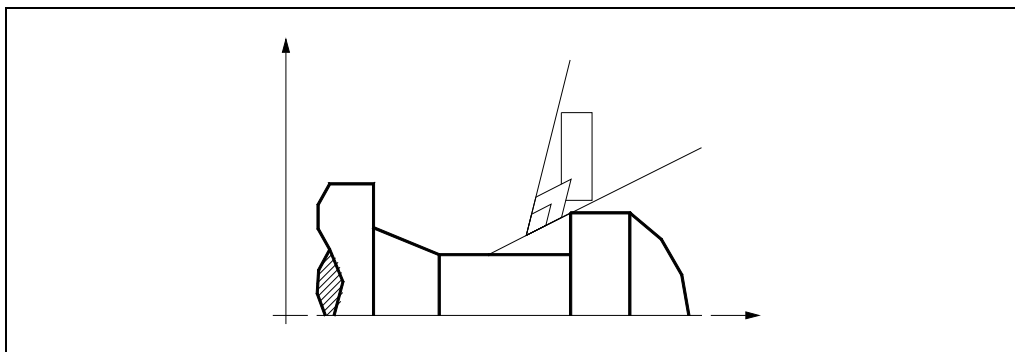
Once the necessary roughing passes have been calculated, it will machine the new resulting profile.

The machining operation will be performed keeping the square corner mode (G07) or round corner mode (G05) that is active when calling the cycle. Likewise, it maintains the same pass during the whole machining operation.

The profile and the tool

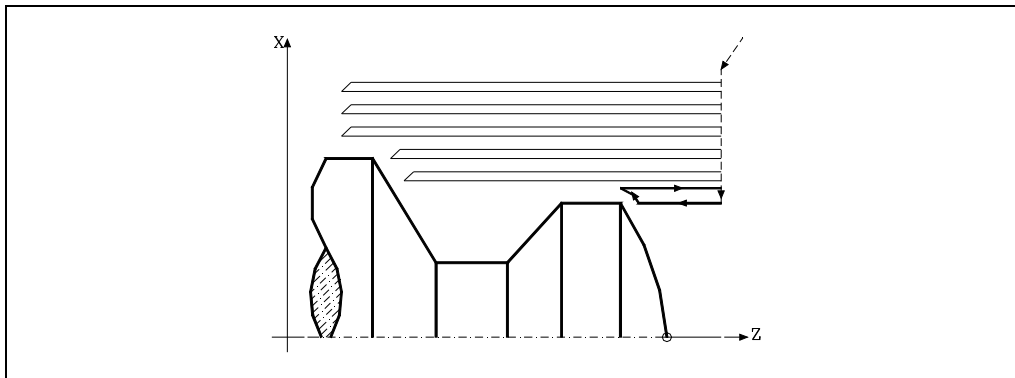
After analyzing the programmed profile and depending on the tool used, it will execute that profile or the one closest to it if the one programmed cannot be executed. When the programmed profile (valleys) cannot be executed with the selected tool, a message will be displayed at the beginning of the execution of the cycle.

The operator will have the choice to stop the execution and select the right tool. If he doesn't do that, the CNC calculates a new profile in the areas that cannot be accessed by the selected tool and it machines as much as possible. The message is shown during the whole machining operation.

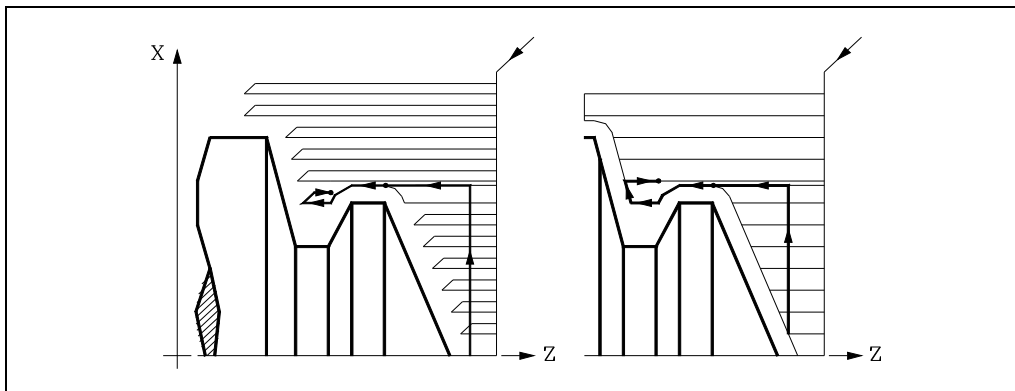


Machining of channels

If when running one of the roughing passes, the CNC detects that there is a channel, the CNC will go on executing the rest of the profile ignoring that channel. A profile may have an unlimited number of channels.



Once the CNC is done machining the excess profile, it will start executing the detected channels.



To do this, it will return in G00 to the point where the machining of the profile was interrupted.

9.

CANNED CYCLES

G68. X axis roughing canned cycle

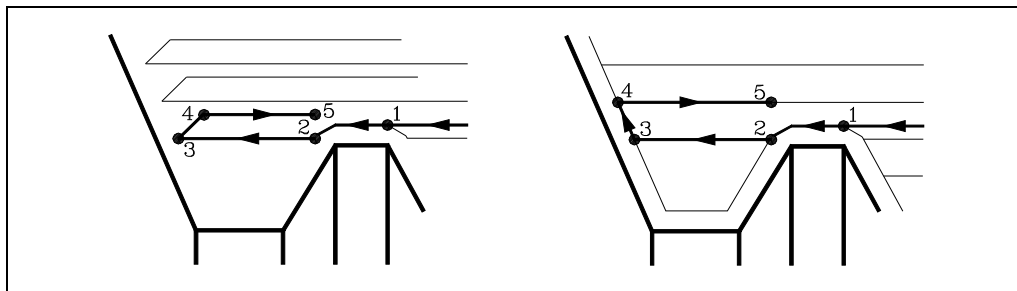
FAGOR 

CNC 8037

·T· MODEL
SOFT: V01.4x

9.

CANNED CYCLES
G68. X axis roughing canned cycle



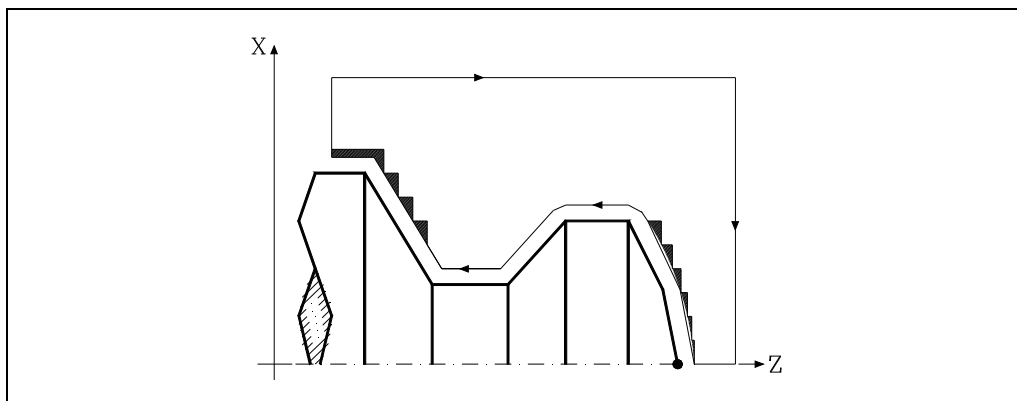
1. From this point on, it will follow the programmed contour in G01 maintaining the finishing stock until reaching the selected pass depth "C". Section 1-2.
2. In the new roughing pass, the movement "2-3" is made in G01 at the programmed feedrate (F).
3. If "D" has been programmed, the 3-4 move is done in rapid (G00); but if "D" has not been programmed, the 3-4 move is done following the profile in G01 at the programmed feedrate (F).
4. The withdrawal movement "4-5" is made in rapid traverse (G00).

If when machining a channel, other channels inside it are detected, the same procedure explained above will be followed.

Last roughing pass

If a final roughing pass has been programmed, the CNC will run a pass parallel to the profile maintaining an "L" stock at the set feedrate "F". This final roughing pass will eliminate the ridges that were left in the roughing stage.

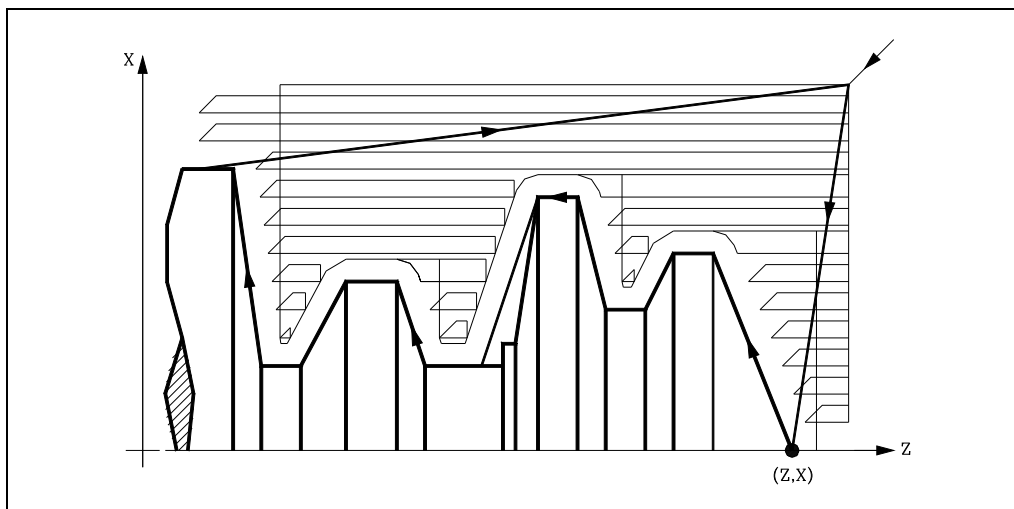
Once the profile roughing operation has ended, the tool will return to the point from where the cycle was called.



Last finishing pass

If a finishing pass has been defined, it will be carried out with tool radius compensation and at the indicated feedrate "H".

This profile will coincide or not with the programmed profile depending on whether there are areas not accessible to the selected tool.



Once the finishing pass has ended, the tool will return to the point from where the cycle was called.

9.

CANNED CYCLES

G68. X axis roughing canned cycle

FAGOR 

CNC 8037

·T· MODEL
SOFT: V01.4x

9.2.2 Profile programming syntax

When defining the profile, there is no need to program the starting point because it is specified by the cycle defining parameters X and Z.

When defining 2 profiles, first define the final profile and then that of the original stock.

The first and last profile defining blocks (where the profile or profiles end) must have the block label number. These label numbers will indicate the beginning and end of the profile geometry definition.

The profile programming syntax must follow these rules:

- It may be programmed in absolute or incremental coordinates and may be made up of simple geometrical elements such as straight lines, arcs, rounded corners, chamfered corners, etc. following the syntax rules defined for this purpose.
- G00 indicates the end of the definition of the final profile and that in this block contains the beginning of the definition of the profile of the original stock.
Program G01, G02 or G03 in the next block, because G00 is modal, thus preventing the CNC from issuing the corresponding error message.
- The profile description must not contain mirror images, scaling factor changes, pattern rotation or zero offsets.
- It must not contain high level blocks such as jumps, calls to subroutines or parametric programming.
- It must not contain other canned cycles.

The following functions may be used to define the profile:

G01	Linear interpolation.
G02	Clockwise circular interpolation.
G03	Counterclockwise circular interpolation.
G06	Circle center in absolute coordinates.
G08	Arc tangent to previous path.
G09	Three point arc.
G36	Automatic radius blend.
G39	Chamfer.
G53	Programming with respect to home.
G70	Programming in inches.
G71	Programming in millimeters.
G90	Absolute programming.
G91	Incremental programming.
G93	Polar origin preset.

The following functions may be programmed, although the cycle will ignore them.

G05	Round corner.
G07	Square corner.
G50	Controlled corner rounding.
F S T D or M functions.	

9.

CANNED CYCLES
G68. X axis roughing canned cycle



CNC 8037

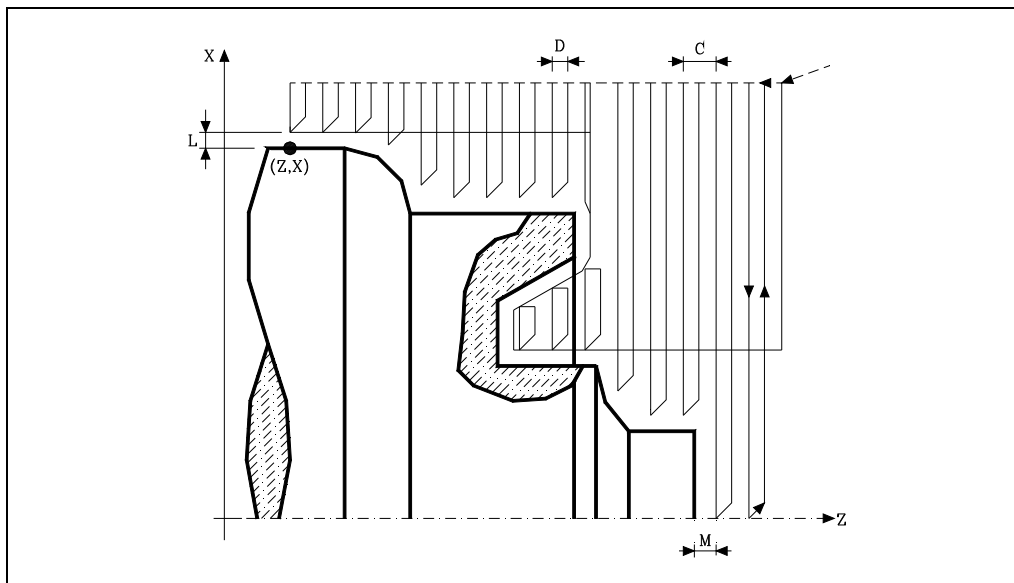
•T• MODEL
SOFT: V01.4x

9.3 G69. Z axis roughing canned cycle

This cycle machines the programmed profile maintaining the indicated step between consecutive machining passes. Triangular, round and square tools may be used in this cycle.

The basic structure of the block is:

G69X Z C D L M K F H S E Q



X±5.5

It sets the X coordinate of the profile's starting point. It must be programmed in absolute coordinates and in the active units, radius or diameter.

Z±5.5

It sets the Z coordinate of the profile's starting point. It must be programmed in absolute coordinates.

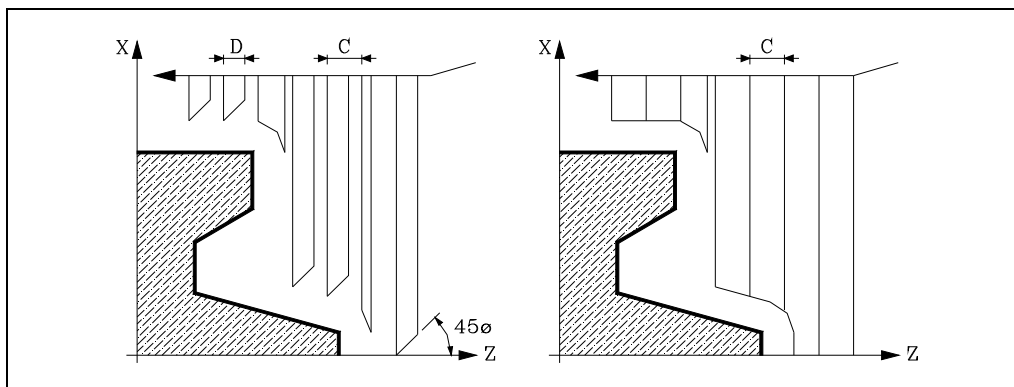
C5.5

It defines the machining pass. If programmed with a 0 value, the CNC will display the corresponding error message.

All the machining passes will be run with this step, except the last one that will remove the residual stock.

D5.5

It sets the safety distance the tool withdraws after each pass.



When programming D with a value other than 0, the cutter withdraws at 45° until reaching the safety distance (left figure).

When programming D with a 0 value, the exit path is the same as the entry path. This may be useful to groove complex profiles, to use these cycle on cylindrical grinders, etc.

If the parameter "D" is not programmed, the tool withdraws following the profile up to the previous pass, "C" distance (right figure).

9.

CANNED CYCLES
G69. Z axis roughing canned cycle

FAGOR 

CNC 8037

·T· MODEL
SOFT: V01.4x

It must be borne in mind that when not programming the parameter D, the cycle execution time is greater but the finishing stock is smaller.

L±5.5

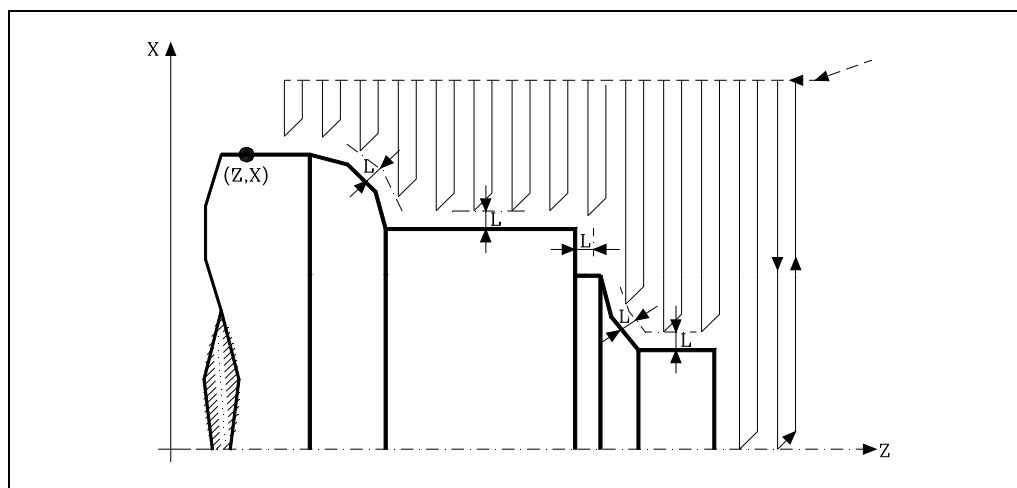
It defines the finishing stock in X. It is defined in radius and if not programmed, it assumes a value of "0".

M±5.5

It defines the finishing stock in Z.

If "L" or "M" is programmed with a negative value, the finishing pass is carried out in round corner mode (G05). When both parameters are programmed with a positive value, the finishing pass is carried out in square corner mode (G07).

If the parameter "M" is not programmed, the residual stock indicated in the parameter "L" will be constant throughout the whole profile.

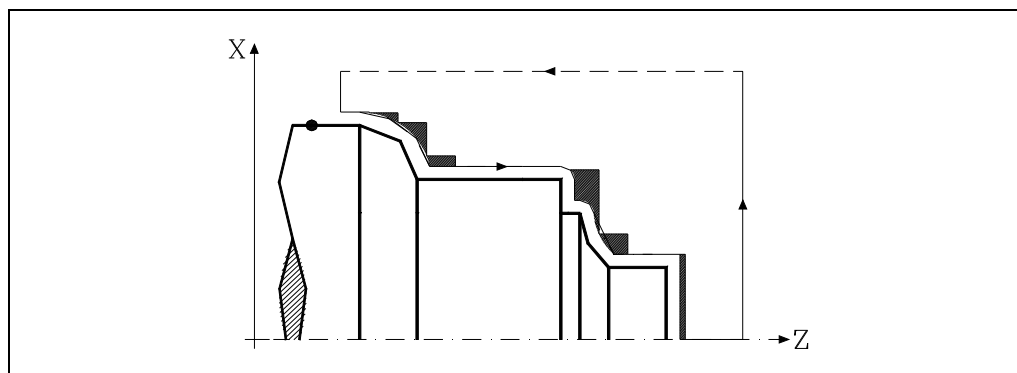


K5.5

It defines the tool feedrate to penetrate the valleys. If not programmed or programmed with a 0 value, it assumes the machining feedrate (the one programmed before calling the cycle).

F5.5

It defines the feedrate for the last roughing pass. If not programmed or programmed with a 0 value, it does not run the last roughing pass.



H5.5

It defines the feedrate during the finishing pass. If not programmed or programmed with a 0 value, it does not run the finishing pass.

S4

Defines the label number of the block where the geometrical description of the profile of the part starts.

9.

CANNED CYCLES
G69. Z axis roughing canned cycle



CNC 8037

·T· MODEL
SOFT: V01.4x

E4

Defines the label number of the block where the geometrical description of the profile of the part ends.

Q6

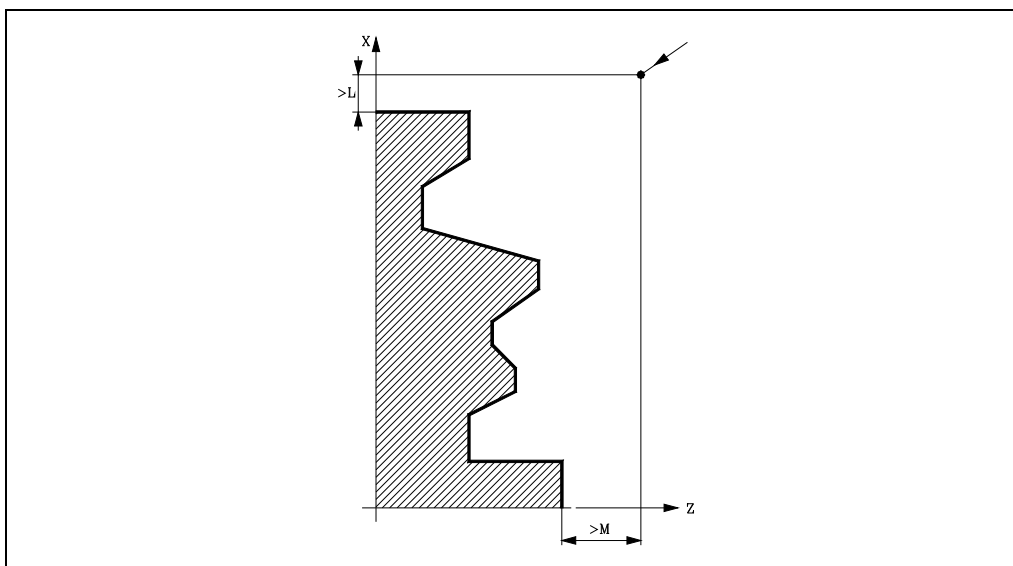
Defines the number of the program that contains the geometrical description of the profile.

This parameter is optional and if it is not defined, the CNC assumes that the profile is defined in the same program that is calling the cycle.

Considerations

The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the cycle.

The canned cycle calling point must be off the part to be machined and at a distance greater than the one defined as excess stock (L, M) along the X and Z axes.

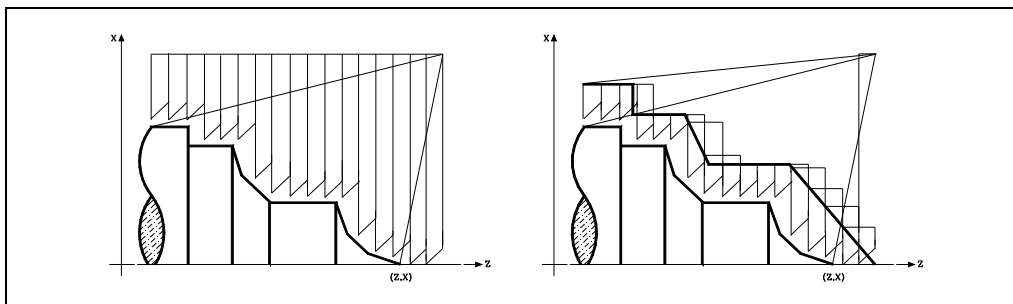


If the tool position is not correct to execute the cycle, the CNC will display the corresponding error.

Once the canned cycle has ended, the active feedrate will be the last one programmed, i.e. the one corresponding to the roughing operation (F) or to the finishing operation (H). The CNC will assume functions G00, G40 and G90.

Optimizing the machining operation.

If only the desired profile is defined, the CNC assumes that the original stock is cylindrical and machines it as shown on the left.



When the stock profile is known, it is recommended to define both profiles: The profile of the original stock and the desired final profile. The machining operation is much faster because only the stock delimited by both profiles is removed.

See ["9.3.2 Profile programming syntax"](#) on page 153.

9.

CANNED CYCLES

G69. Z axis roughing canned cycle

FAGOR 

CNC 8037

·T· MODEL
SOFT: V01.4x

9.3.1 Basic operation

The machining passes

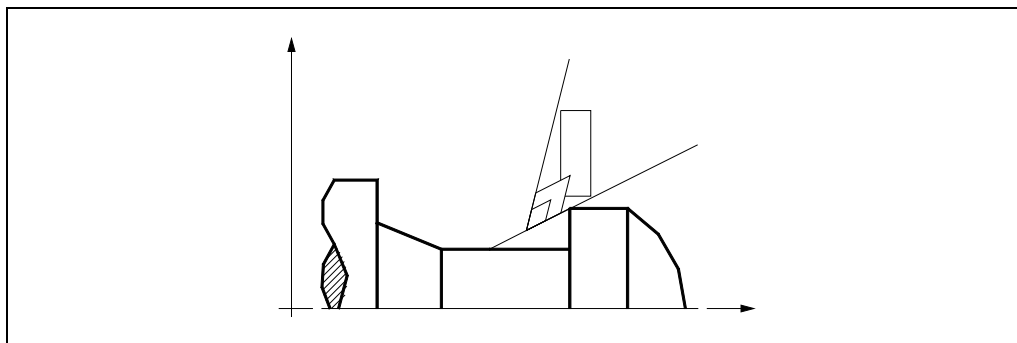
Once the necessary roughing passes have been calculated, it will machine the new resulting profile.

The machining operation will be performed keeping the square corner mode (G07) or round corner mode (G05) that is active when calling the cycle. Likewise, it maintains the same pass during the whole machining operation.

The profile and the tool

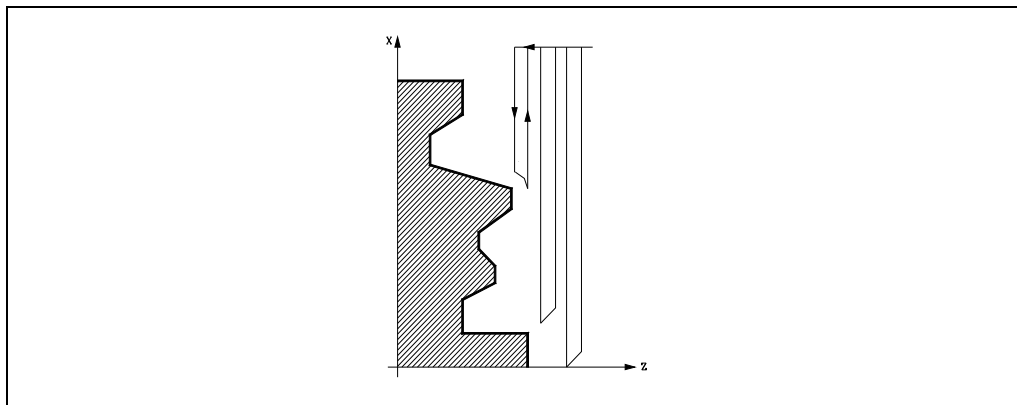
After analyzing the programmed profile and depending on the tool used, it will execute that profile or the one closest to it if the one programmed cannot be executed. When the programmed profile (valleys) cannot be executed with the selected tool, a message will be displayed at the beginning of the execution of the cycle.

The operator will have the choice to stop the execution and select the right tool. If he doesn't do that, the CNC calculates a new profile in the areas that cannot be accessed by the selected tool and it machines as much as possible. The message is shown during the whole machining operation.

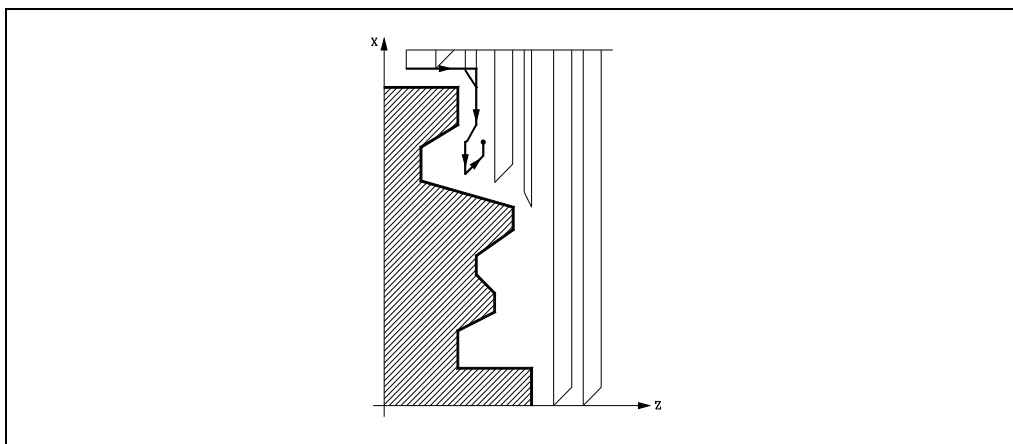


Machining of channels

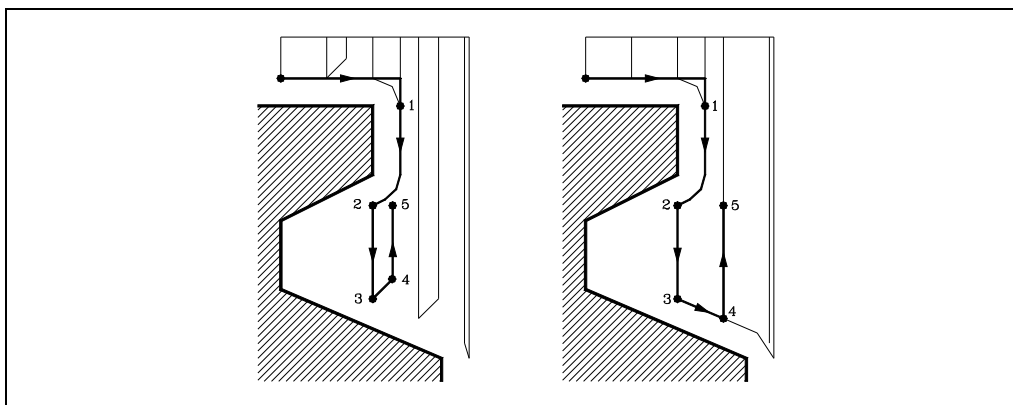
If when running one of the roughing passes, the CNC detects that there is a channel, the CNC will go on executing the rest of the profile ignoring that channel. A profile may have an unlimited number of channels.



Once the CNC is done machining the excess profile, it will start executing the detected channels.



To do this, it will return in G00 to the point where the machining of the profile was interrupted.



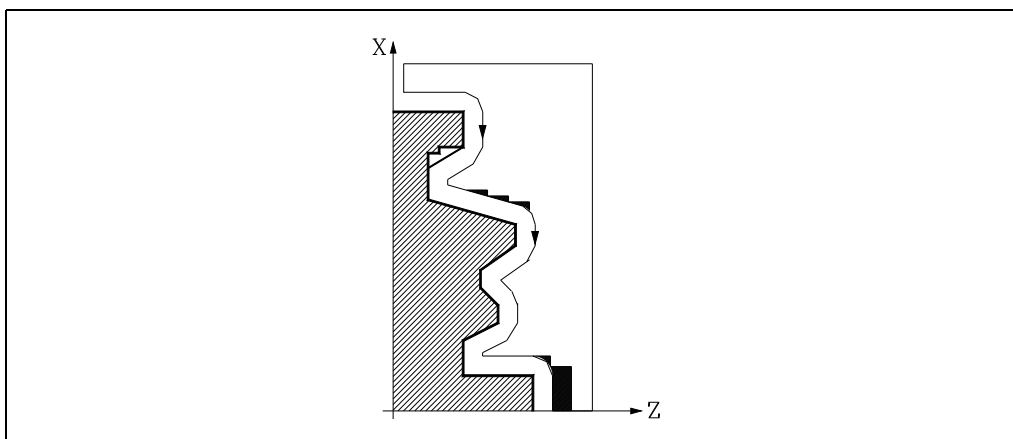
1. From this point on, it will follow the programmed contour in G01 maintaining the finishing stock until reaching the selected pass depth "C". Section 1-2.
2. In the new roughing pass, the movement "2-3" is made in G01 at the programmed feedrate (F).
3. If "D" has been programmed, the 3-4 move is done in rapid (G00); but if "D" has not been programmed, the 3-4 move is done following the profile in G01 at the programmed feedrate (F).
4. The withdrawal movement "4-5" is made in rapid traverse (G00).

If when machining a channel, other channels inside it are detected, the same procedure explained above will be followed.

Last roughing pass

If a final roughing pass has been programmed, the CNC will run a pass parallel to the profile maintaining an "L" stock at the set feedrate "F". This final roughing pass will eliminate the ridges that were left in the roughing stage.

Once the profile roughing operation has ended, the tool will return to the point from where the cycle was called.



9.

CANNED CYCLES

G69. Z axis roughing canned cycle

FAGOR

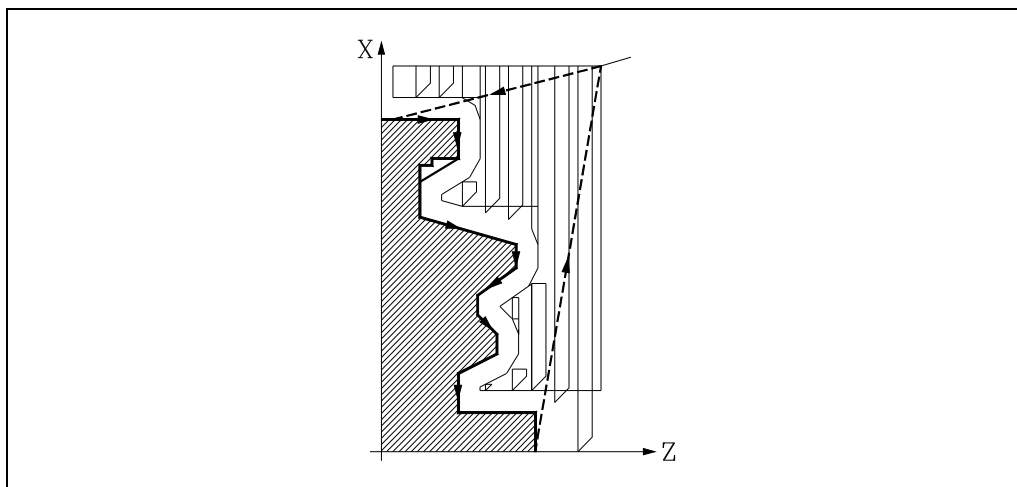
CNC 8037

·T· MODEL
SOFT: V01.4x

Last finishing pass

If a finishing pass has been defined, it will be carried out with tool radius compensation and at the indicated feedrate "H".

This profile will coincide or not with the programmed profile depending on whether there are areas not accessible to the selected tool.



Once the finishing pass has ended, the tool will return to the point from where the cycle was called.

9.

CANNED CYCLES

G69. Z axis roughing canned cycle



CNC 8037

·T· MODEL
SOFT: V01.4x

9.3.2 Profile programming syntax

When defining the profile, there is no need to program the starting point because it is specified by the cycle defining parameters X and Z.

When defining 2 profiles, first define the final profile and then that of the original stock.

The first and last profile defining blocks (where the profile or profiles end) must have the block label number. These label numbers will indicate the beginning and end of the profile geometry definition.

The profile programming syntax must follow these rules:

- It may be programmed in absolute or incremental coordinates and may be made up of simple geometrical elements such as straight lines, arcs, rounded corners, chamfered corners, etc. following the syntax rules defined for this purpose.
- G00 indicates the end of the definition of the final profile and that in this block contains the beginning of the definition of the profile of the original stock.
Program G01, G02 or G03 in the next block, because G00 is modal, thus preventing the CNC from issuing the corresponding error message.
- The profile description must not contain mirror images, scaling factor changes, pattern rotation or zero offsets.
- It must not contain high level blocks such as jumps, calls to subroutines or parametric programming.
- It must not contain other canned cycles.

The following functions may be used to define the profile:

G01	Linear interpolation.
G02	Clockwise circular interpolation.
G03	Counterclockwise circular interpolation.
G06	Circle center in absolute coordinates.
G08	Arc tangent to previous path.
G09	Three point arc.
G36	Automatic radius blend.
G39	Chamfer.
G53	Programming with respect to home.
G70	Programming in inches.
G71	Programming in millimeters.
G90	Absolute programming.
G91	Incremental programming.
G93	Polar origin preset.

The following functions may be programmed, although the cycle will ignore them.

G05	Round corner.
G07	Square corner.
G50	Controlled corner rounding.

F S T D or M functions.

9.

CANNED CYCLES

G69. Z axis roughing canned cycle



CNC 8037

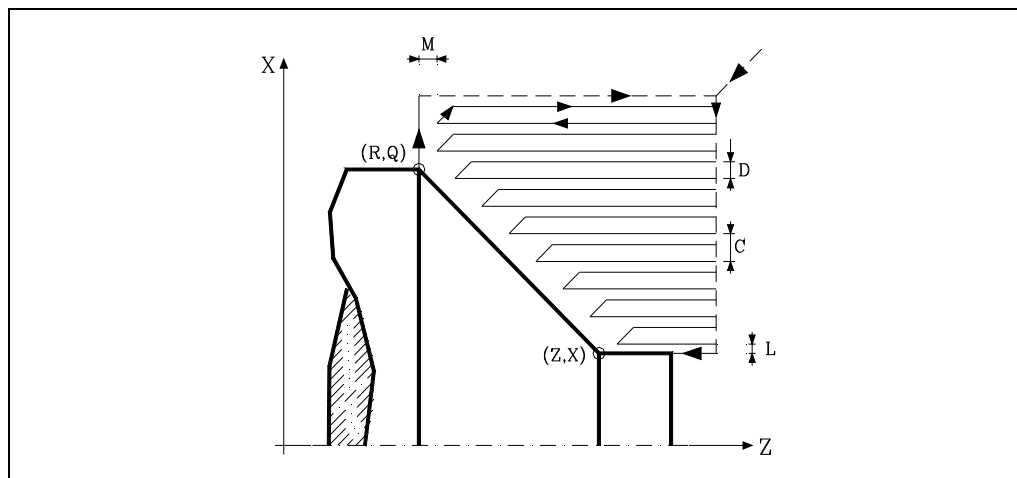
·T· MODEL
SOFT: V01.4x

9.4 G81. Turning canned cycle for straight sections

This cycle turns the programmed section maintaining the indicated pass between the successive turning passes. The cycle allows to select whether or not it will run a finishing pass after completing the programmed turning operation.

The basic structure of the block is:

G81 X Z Q R C D L M F H



X±5.5

It sets the X coordinate of the profile's starting point. It must be programmed in absolute coordinates and in the active units, radius or diameter.

Z±5.5

It sets the Z coordinate of the profile's starting point. It must be programmed in absolute coordinates.

Q±5.5

Defines the X coordinate of the final point of the profile. It must be programmed in absolute coordinates and in the active units, radius or diameter.

R±5.5

Defines the Z coordinate of the final point of the profile.

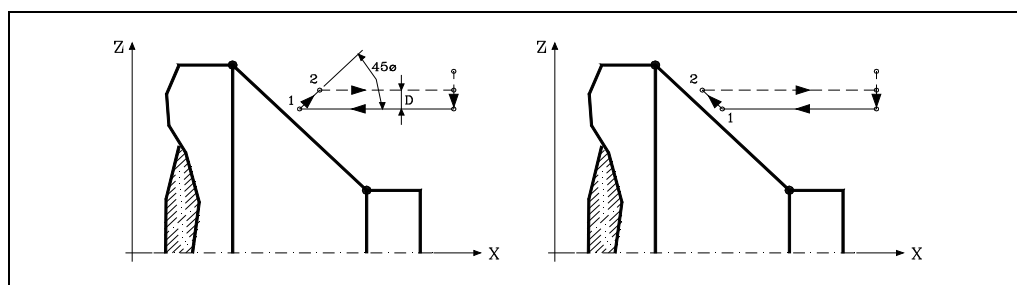
C5.5

Defines the turning pass. It is given by a positive radius value. If programmed with a 0 value, the CNC will display the corresponding error message.

The whole turning operation is carried out with the same pass which is the same or smaller than the programmed one "C".

D5.5

It sets the safety distance the tool withdraws after each pass.



When programming D with a value other than 0, the cutter withdraws at 45° until reaching the safety distance (left figure).

When programming D with a 0 value, the exit path is the same as the entry path.

9.

CANNED CYCLES
G81. Turning canned cycle for straight sections



CNC 8037

·T· MODEL
SOFT: V01.4x

If the parameter "D" is not programmed, the tool withdraws following the profile up to the previous pass, "C" distance (right figure).

It must be borne in mind that when not programming the parameter D, the cycle execution time is greater but the finishing stock is smaller.

L5.5

Defines the finishing stock along X and it is given in radius.

If not programmed, a value of 0 is assumed.

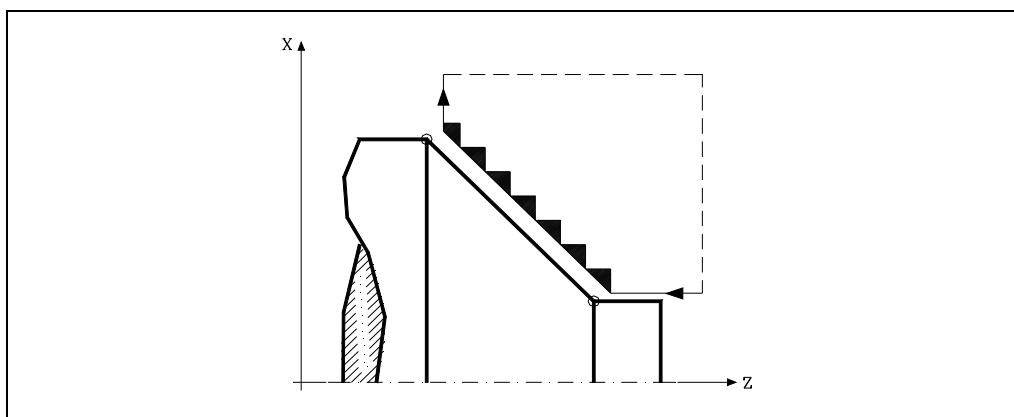
M5.5

Defines the finishing stock along Z.

If not programmed, a value of 0 is assumed.

F5.5

It defines the feedrate for the last roughing pass. If not programmed or programmed with a 0 value, it does not run the last roughing pass.



H5.5

It defines the feedrate during the finishing pass.

If not programmed or programmed with a 0 value, it does not run the finishing pass.

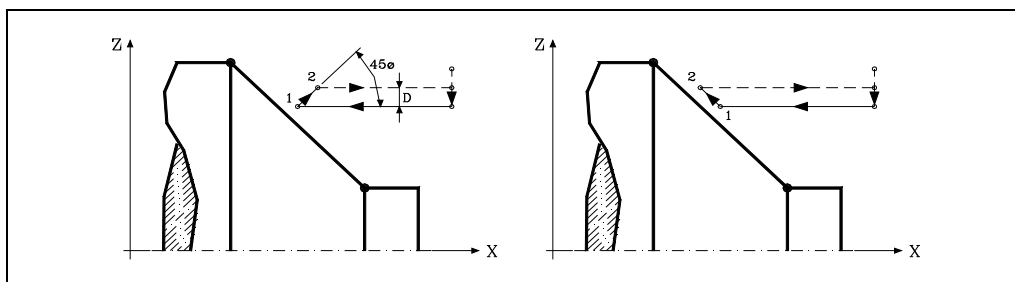
9.

CANNED CYCLES
G81. Turning canned cycle for straight sections

9.4.1 Basic operation

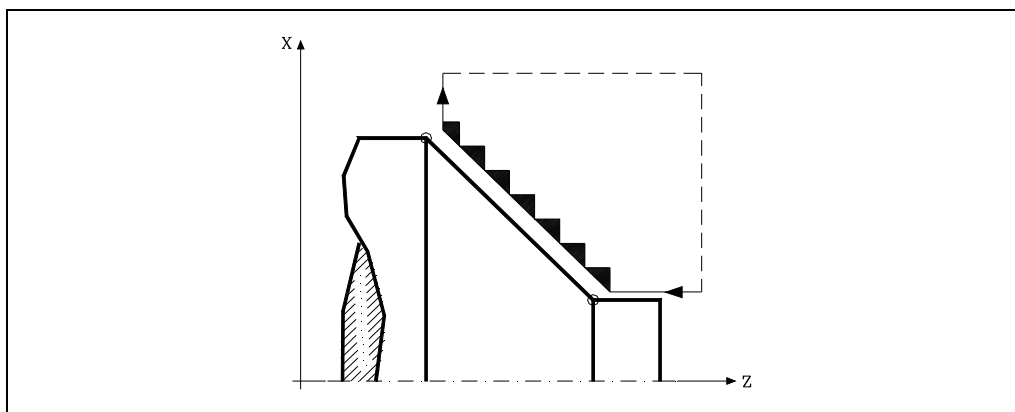
The canned cycle will analyze the programmed profile performing, if necessary, a horizontal turning operation until the defined profile is reached. The whole turning operation is carried out with the same pass which is the same or smaller than the programmed one "C".

Each turning pass is carried out as follows:

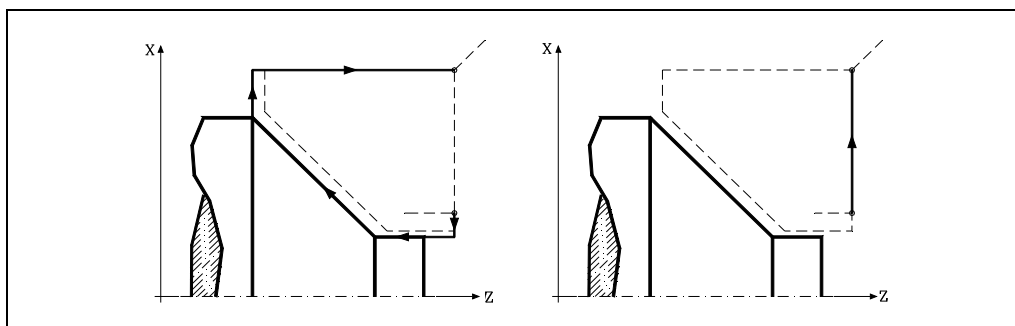


- "1-2" move in rapid (G00).
- Move "2-3" in G01 at programmed feedrate (F).
- If "D" has been programmed, the 3-4 move is done in rapid (G00); but if "D" has not been programmed, the 3-4 move is done following the profile in G01 at the programmed feedrate (F).
- The withdrawal movement "4-5" is made in rapid traverse (G00).

If a final roughing pass has been programmed, the CNC will run a pass parallel to the profile maintaining an "L" and "M" stock at the set feedrate "F". This final roughing pass will eliminate the ridges that were left in the roughing stage.



After the turning operation (with or without finishing pass) the canned cycle will always end at the cycle calling point.



Considerations

The machining conditions (feedrate, spindle speed, etc.) as well as the tool radius compensation (G41, G42) must be programmed before calling the cycle.

Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle.

9.

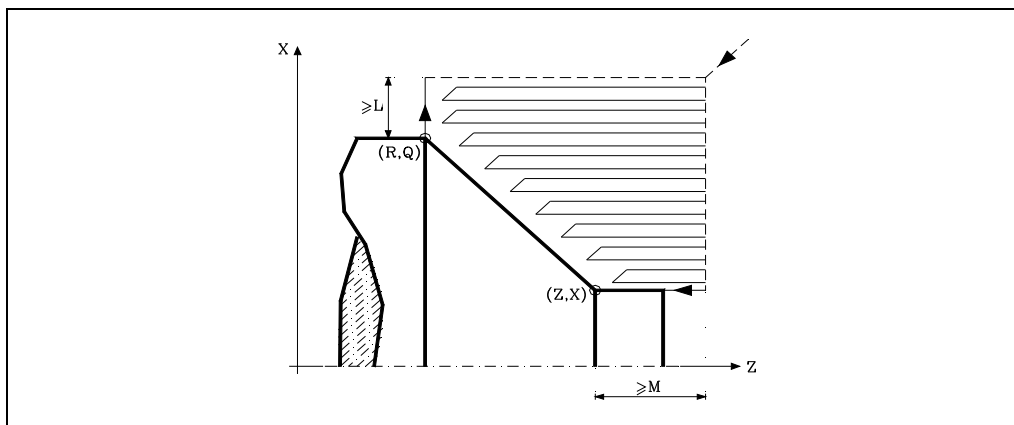
CANNED CYCLES
G81. Turning canned cycle for straight sections



CNC 8037

·T· MODEL
SOFT: V01.4x

The distance between the starting point and final point (R,Q) along the X axis must be equal to or greater than L. The distance between the starting point and the initial point (X, Z) along the Z axis must be equal to or greater than M.



If the tool position is not correct to execute the cycle, the CNC will display the corresponding error.

9.

CANNED CYCLES

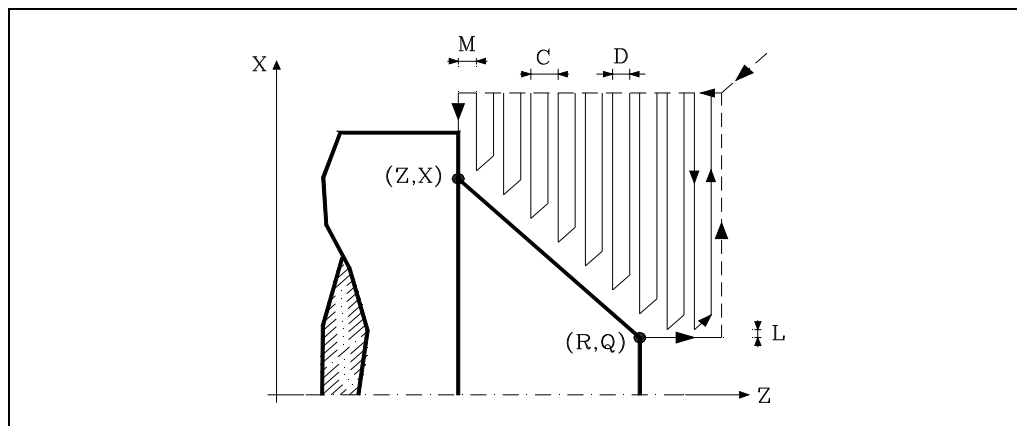
G81. Turning canned cycle for straight sections

9.5 G82. Facing canned cycle for straight sections

This cycle faces the programmed section maintaining the indicated pass between the successive facing passes. The cycle allows to select whether or not it will run a finishing pass after completing the programmed facing operation.

The basic structure of the block is:

G82 X Z Q R C D L M F H



X±5.5

It sets the X coordinate of the profile's starting point. It must be programmed in absolute coordinates and in the active units, radius or diameter.

Z±5.5

It sets the Z coordinate of the profile's starting point. It must be programmed in absolute coordinates.

Q±5.5

Defines the X coordinate of the final point of the profile. It must be programmed in absolute coordinates and in the active units, radius or diameter.

R±5.5

Defines the Z coordinate of the final point of the profile.

C5.5

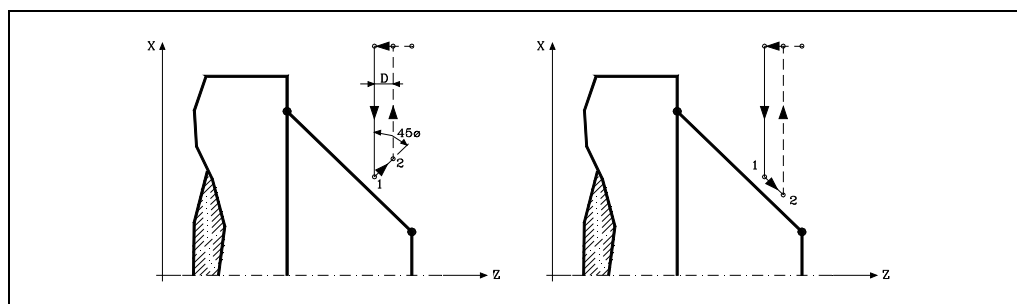
Defines the facing pass.

The whole facing operation is carried out with the same pass, this being equal to or smaller than the programmed one (C).

If programmed with a 0 value, the CNC will display the corresponding error message.

D5.5

It sets the safety distance the tool withdraws after each pass.



When programming D with a value other than 0, the cutter withdraws at 45° until reaching the safety distance (left figure).

When programming D with a 0 value, the exit path is the same as the entry path.

9.

CANNED CYCLES
G82. Facing canned cycle for straight sections



CNC 8037

·T· MODEL
SOFT: V01.4x

If the parameter "D" is not programmed, the tool withdraws following the profile up to the previous pass, "C" distance (right figure).

It must be borne in mind that when not programming the parameter D, the cycle execution time is greater but the finishing stock is smaller.

L5.5

Defines the finishing stock along X and it is given in radius.

If not programmed, a value of 0 is assumed.

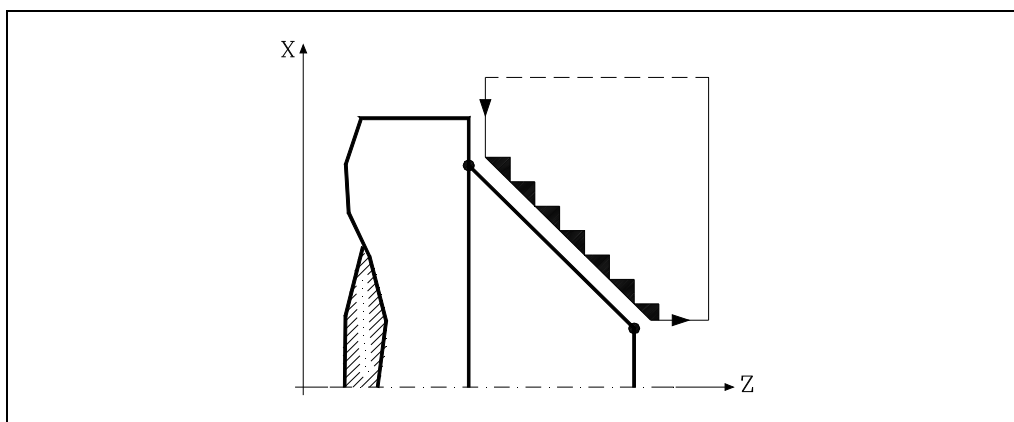
M5.5

Defines the finishing stock along Z.

If not programmed, a value of 0 is assumed.

F5.5

It defines the feedrate for the last roughing pass. If not programmed or programmed with a 0 value, it does not run the last roughing pass.



H5.5

It defines the feedrate during the finishing pass.

If not programmed or programmed with a 0 value, it does not run the finishing pass.

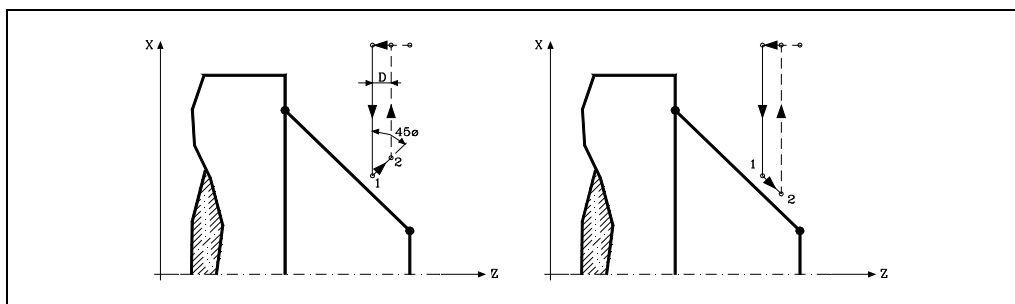
9.

CANNED CYCLES
G82. Facing canned cycle for straight sections

9.5.1 Basic operation

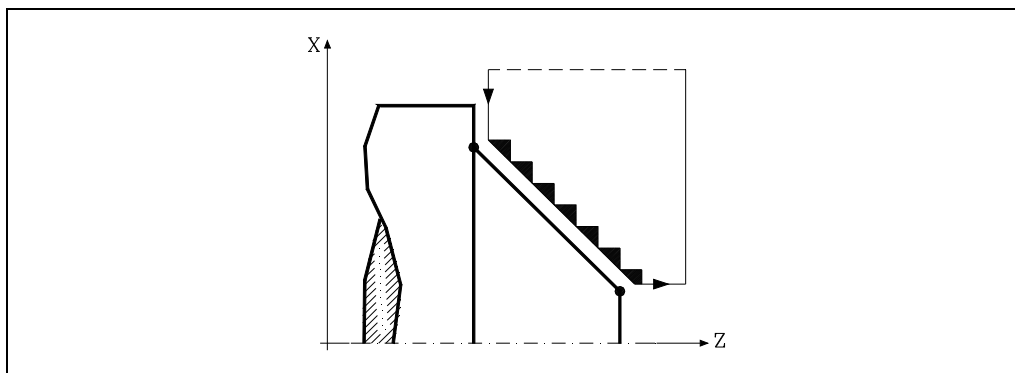
The canned cycle will analyze the programmed profile performing, if necessary, a facing operation until the defined profile is reached. The whole facing operation is carried out with the same pass, this being equal to or smaller than the programmed one (C).

Each facing pass is carried out in the following manner:

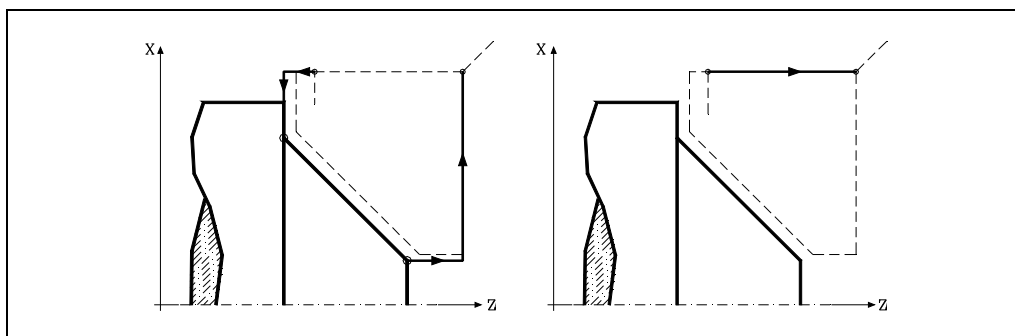


- "1-2" move in rapid (G00).
- Move "2-3" in G01 at programmed feedrate (F).
- If "D" has been programmed, the 3-4 move is done in rapid (G00); but if "D" has not been programmed, the 3-4 move is done following the profile in G01 at the programmed feedrate (F).
- The withdrawal movement "4-5" is made in rapid traverse (G00).

If a final roughing pass has been programmed, the CNC will run a pass parallel to the profile maintaining an "L" and "M" stock at the set feedrate "F". This final roughing pass will eliminate the ridges that were left in the roughing stage.



After the facing operation (with or without finishing pass) the canned cycle will always end at the cycle calling point.



Considerations

The machining conditions (feedrate, spindle speed, etc.) as well as the tool radius compensation (G41, G42) must be programmed before calling the cycle.

Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle.

9.

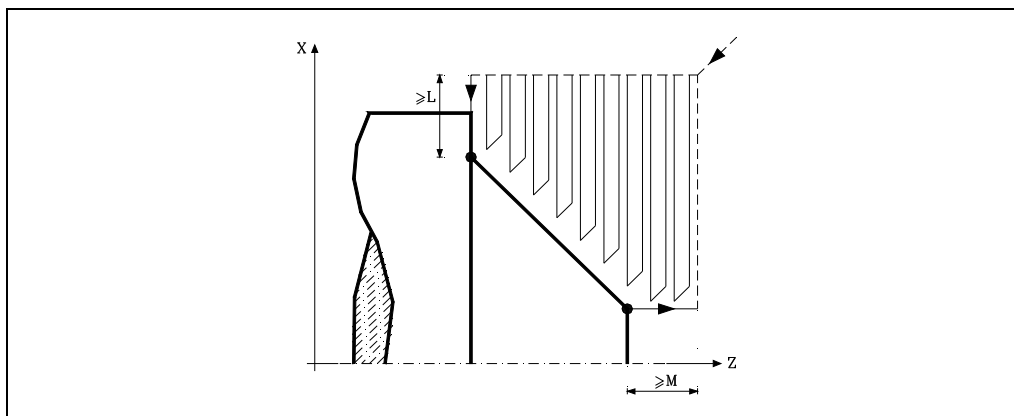
CANNED CYCLES
G82. Facing canned cycle for straight sections



CNC 8037

·T· MODEL
SOFT: V01.4x

The distance between the starting point and the initial point (X, Z) along the X axis must be equal to or greater than L. The distance between the starting point and final point (R,Q) along the Z axis must be equal to or greater than M.



If the tool position is not correct to execute the cycle, the CNC will display the corresponding error.

9.

CANNED CYCLES
G82. Facing canned cycle for straight sections

FAGOR 

CNC 8037

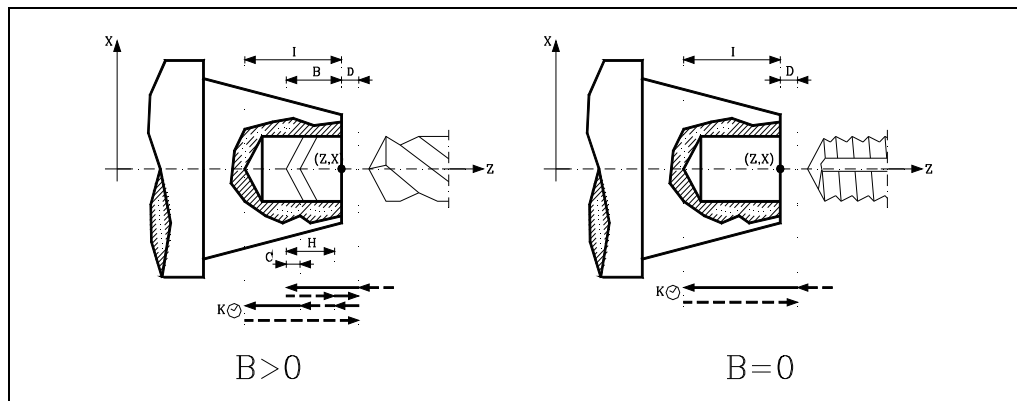
·T· MODEL
SOFT: V01.4x

9.6 G83. Axial drilling and tapping canned cycle

This cycle may be used to carry out axial drilling or axial tapping. Whether one operation or the other will be performed depends on the format being used. For tapping, set parameter "B=0", and for axial drilling, set "B>0".

The basic block structure for each operation is:

Axial drilling	G83 X Z I B D K H C L R
Axial tapping	G83 X Z I B0 D K R



X±5.5

Defines the X coordinate where the cycle is to be run. It must be programmed in absolute coordinates and in the active units, radius or diameter.

Z±5.5

Defines the Z coordinate where the cycle will be run. It must be programmed in absolute coordinates.

I±5.5

Defines the total depth. It is referred to the drilling point (X, Z); thus, it will have a positive value when drilling in the negative Z direction and vice versa.

If programmed with a 0 value, the CNC will display the corresponding error message.

B5.5

Defines the type of operation to be run.

- If B=0, the CNC will carry out an axial tapping.
- If B>0 it will carry out an axial drilling where B indicates the drilling step.

D5.5

Defines the safety distance and it indicates at what distance the tool is positioned in the approach move. If not programmed, a value of 0 is assumed.

K5

Defines the dwell, in hundredths of a second, at the bottom of the hole before starting the withdrawal. If not programmed, a value of 0 is assumed.

H5.5

Defines the withdrawal distance after each drilling step (in G00). If not programmed or programmed a value of 0, it will withdraw to the approach point.

C5.5

Defines the approach distance along Z from the previous drilling step to start the next drilling step (in G00). If not programmed, a value of 1 millimeter.

L5.5

Optional. In the drilling cycle, it defines the minimum drilling peck. It is used with "R" values other than 1.

9.

CANNED CYCLES
G83. Axial drilling and tapping canned cycle



CNC 8037

·T· MODEL
SOFT: V01.4x

If not programmed, a value of 0 is assumed.

R5.5

In the drilling cycle, it indicates the step that reduces the drilling peck "B". If it is not programmed or is programmed with a value of "0", it assumes a value of "1".

- With $R=1$, all the drilling pecks will be the same (the programmed "B" value).
- If R is other than 1, the first drilling peck will be "B", the second one "R B", The third one "R (RB)" and so on. In other words, from the second peck on, the new peck will be the product of the R factor times the previous peck.

In the tapping cycle it defines the type of tapping to be done. If not programmed, it takes the value of "0", regular tapping.

- With R0, regular tapping.
- With R1, rigid tapping. The CNC stops the tool with M19 and orients it to begin tapping.
- With R2, rigid tapping. If the tool is turning in M3 or M4, the CNC does not stop it nor orient it to begin tapping. This option does not allow thread repair, even if the part has not been released because the thread start (entry) will coincide with the one machined earlier.

In order to execute a rigid tapping, the relevant spindle (main or secondary) must be ready to operate in closed loop. In other words, that it must have a servo drive-motor system with rotary encoder.

9.

CANNED CYCLES

G83. Axial drilling and tapping canned cycle

9.6.1 Basic operation

Drilling

1. Rapid move to the approach point located at a "D" distance from the first hole position.
2. First drilling operation. The drilling axis moves in G01 to the programmed incremental depth "D+B".
3. Drilling loop. The following steps will be repeated until the programmed depth "I" is reached.
It first withdraws in rapid (G00) a distance "H" or to the approach point. Rapid approach (G00) to a "C" distance from the next peck.
New drilling step. G1 move to the next incremental depth "B and R".
4. Dwell "K" in hundredths of a second at the bottom of the hole, if programmed.
5. Withdrawal in rapid (G00) to the approach point.

Tapping

1. Rapid movement to the approach point located at a safety distance "D" from the tap point.
2. Tapping. The drilling axis moves in G01 to the programmed incremental depth "D+B".
3. Reverse the spindle turning direction.
If "K" has been programmed, the spindle stops and after the programmed dwell has elapsed, the spindle starts turning in the other way.
4. Withdrawal in work feedrate to the approach point.

Rigid tapping

1. Tapping takes place at the center of the part (X0). Rapid movement to the approach point located at a safety distance "D" from the tap point.
2. Tapping. Tapping to the incremental depth programmed in "D+B".
It is carried out by interpolating the main spindle (that is turning) with the Z axis. Rigid tapping cannot be interrupted, nor the machining conditions be changed. It is done at 100% of the programmed "S" and "F".
3. Reverse the spindle turning direction.
If "K" has been programmed, the spindle stops and after the programmed dwell has elapsed, the spindle starts turning in the other way.
4. Withdrawal in work feedrate to the approach point.

Rigid tapping is drawn with the color used for "uncompensated tool path". When the cycle is completed, the spindle stops (M5).

Considerations

The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the cycle.

When tapping (regular or rigid), the general logic output "TAPPING" (M5517) stays active during the execution of the cycle.

Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle. Only the tool radius compensation will be canceled (G40) if it was active.

9.

CANNED CYCLES
G83. Axial drilling and tapping canned cycle



CNC 8037

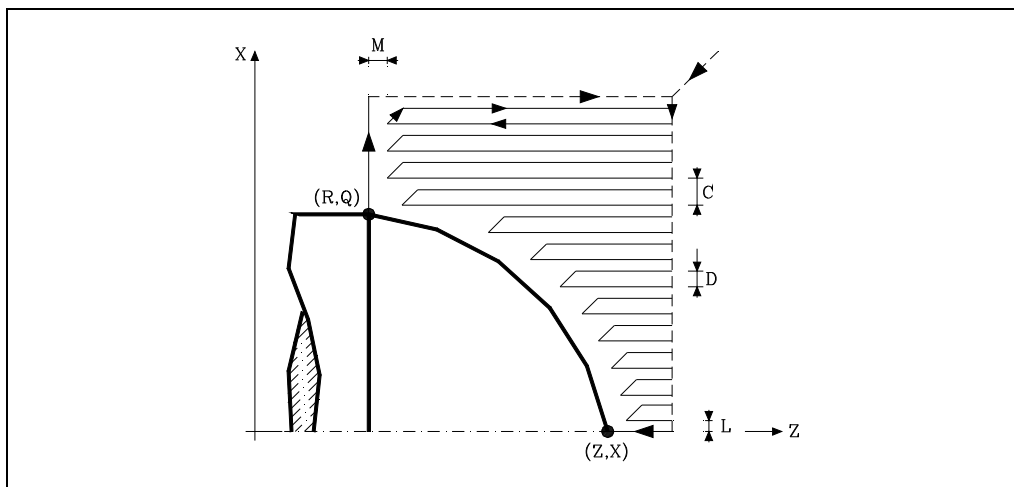
·T· MODEL
SOFT: V01.4x

9.7 G84. Turning canned cycle for curved sections

This cycle turns the programmed section maintaining the indicated pass between the successive turning passes. The cycle allows to select whether or not it will run a finishing pass after completing the programmed turning operation.

The basic structure of the block is:

G84 X Z Q R C D L M F H I K



X±5.5

It sets the X coordinate of the profile's starting point. It must be programmed in absolute coordinates and in the active units, radius or diameter.

Z±5.5

It sets the Z coordinate of the profile's starting point. It must be programmed in absolute coordinates.

Q±5.5

Defines the X coordinate of the final point of the profile. It must be programmed in absolute coordinates and in the active units, radius or diameter.

R±5.5

Defines the Z coordinate of the final point of the profile.

C5.5

Defines the turning pass. It is given by a positive radius value. If programmed with a 0 value, the CNC will display the corresponding error message.

The whole turning operation is carried out with the same pass which is the same or smaller than the programmed one "C".

D5.5

It sets the safety distance the tool withdraws after each pass.

- When programming D with a value other than 0, the cutter withdraws at 45° until reaching the safety distance (left figure).
- When programming D with a 0 value, the exit path is the same as the entry path.
- If the parameter "D" is not programmed, the tool withdraws following the profile up to the previous pass, "C" distance (right figure).

9.

CANNED CYCLES

G84. Turning canned cycle for curved sections

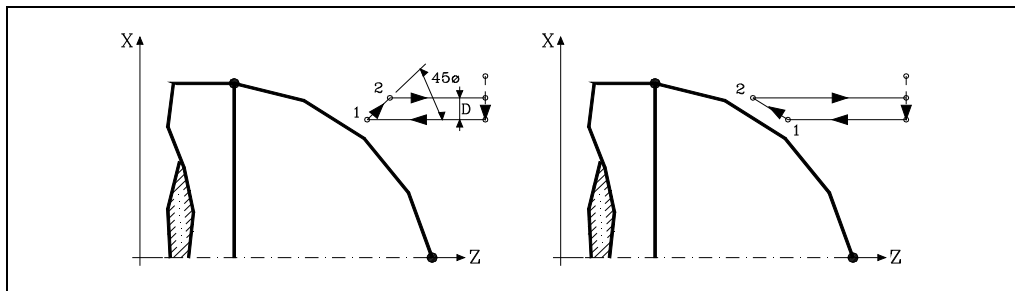
FAGOR 

CNC 8037

·T· MODEL
SOFT: V01.4x

9.

CANNED CYCLES
G84. Turning canned cycle for curved sections



It must be borne in mind that when not programming the parameter D, the cycle execution time is greater but the finishing stock is smaller.

L5.5

Defines the finishing stock along X and it is given in radius.

If not programmed, a value of 0 is assumed.

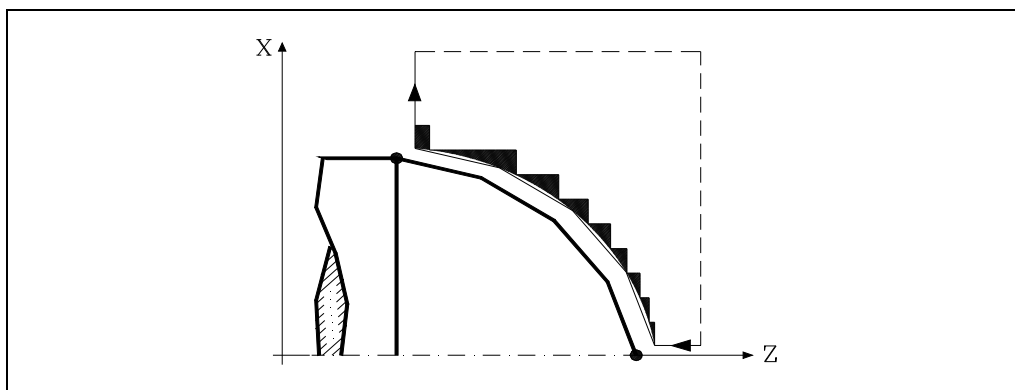
M5.5

Defines the finishing stock along Z.

If not programmed, a value of 0 is assumed.

F5.5

It defines the feedrate for the last roughing pass. If not programmed or programmed with a 0 value, it does not run the last roughing pass.



H5.5

It defines the feedrate during the finishing pass.

If not programmed or programmed with a 0 value, it does not run the finishing pass.

I±5.5

Defines the distance, in radius, from the initial point (X, Z) to the arc's center along the X axis. It is programmed in incremental values with respect to the initial point like the "I" for circular interpolations (G02, G03).

K±5.5

Defines the distance from the initial point (X, Z) to the arc's center along the Z axis. It is programmed in incremental values with respect to the initial point like the "K" for circular interpolations (G02, G03).



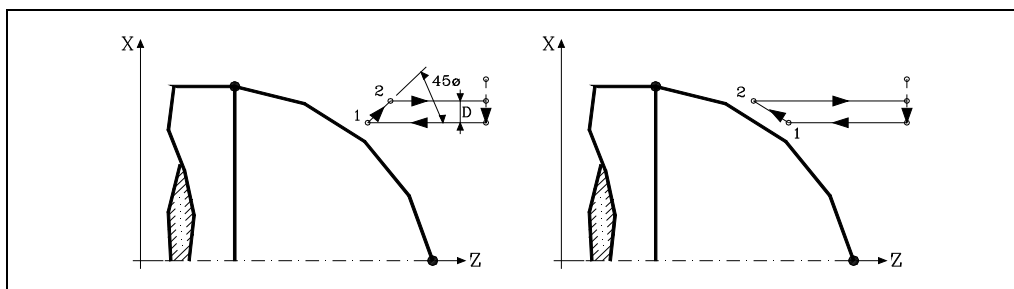
CNC 8037

·T· MODEL
SOFT: V01.4x

9.7.1 Basic operation

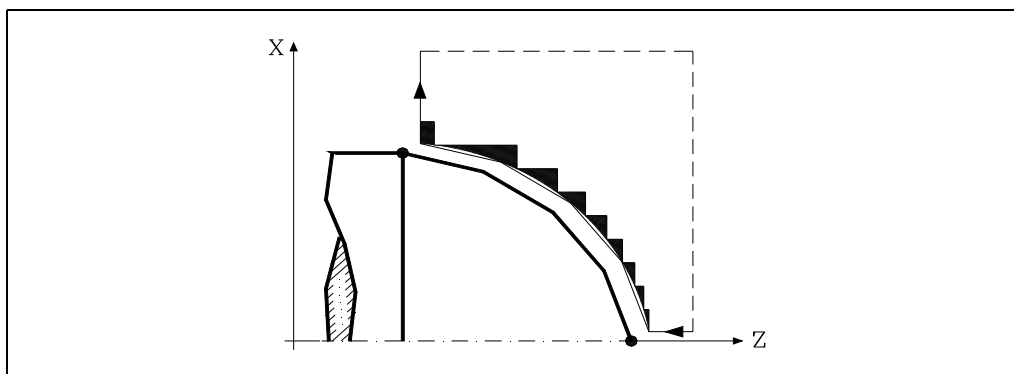
The canned cycle will analyze the programmed profile performing, if necessary, a horizontal turning operation until the defined profile is reached.

The whole turning operation is carried out with the same pass which is the same or smaller than the programmed one "C". Each turning pass is carried out as follows:

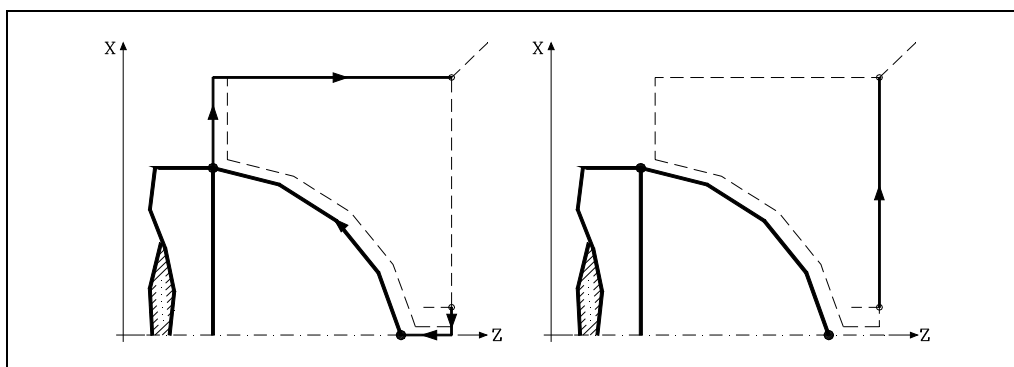


- "1-2" move in rapid (G00).
- Move "2-3" in G01 at programmed feedrate (F).
- If "D" has been programmed, the 3-4 move is done in rapid (G00); but if "D" has not been programmed, the 3-4 move is done following the profile in G01 at the programmed feedrate (F).
- The withdrawal movement "4-5" is made in rapid traverse (G00).

If a final roughing pass has been programmed, the CNC will run a pass parallel to the profile maintaining an "L" and "M" stock at the set feedrate "F". This final roughing pass will eliminate the ridges that were left in the roughing stage.



After the turning operation (with or without finishing pass) the canned cycle will always end at the cycle calling point.



Considerations

The machining conditions (feedrate, spindle speed, etc.) as well as the tool radius compensation (G41, G42) must be programmed before calling the cycle.

Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle.

9.

CANNED CYCLES

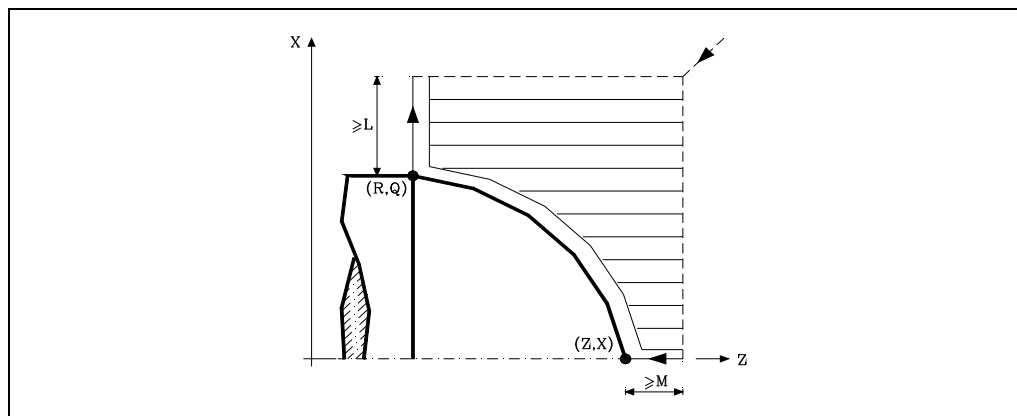
G84. Turning canned cycle for curved sections

FAGOR

CNC 8037

·T· MODEL
SOFT: V01.4x

The distance between the starting point and final point (R,Q) along the X axis must be equal to or greater than L. The distance between the starting point and the initial point (X, Z) along the Z axis must be equal to or greater than M.



If the tool position is not correct to execute the cycle, the CNC will display the corresponding error.

9.

CANNED CYCLES

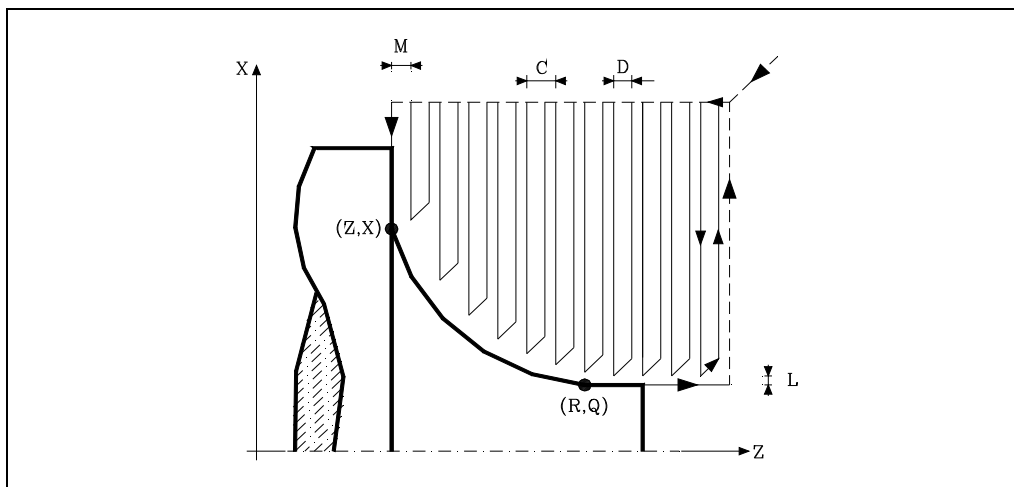
G84. Turning canned cycle for curved sections

9.8 G85. Facing canned cycle for curved sections

This cycle faces the programmed section maintaining the indicated pass between the successive facing passes. The cycle allows to select whether or not it will run a finishing pass after completing the programmed facing operation.

The basic structure of the block is:

G85 X Z Q R C D L M F H I K



X±5.5

It sets the X coordinate of the profile's starting point. It must be programmed in absolute coordinates and in the active units, radius or diameter.

Z±5.5

It sets the Z coordinate of the profile's starting point. It must be programmed in absolute coordinates.

Q±5.5

Defines the X coordinate of the final point of the profile. It must be programmed in absolute coordinates and in the active units, radius or diameter.

R±5.5

Defines the Z coordinate of the final point of the profile.

C5.5

Defines the facing pass. The whole facing operation is carried out with the same pass, this being equal to or smaller than the programmed one (C).

If programmed with a 0 value, the CNC will display the corresponding error message.

D5.5

It sets the safety distance the tool withdraws after each pass.

- When programming D with a value other than 0, the cutter withdraws at 45° until reaching the safety distance (left figure).
- When programming D with a 0 value, the exit path is the same as the entry path.
- If the parameter "D" is not programmed, the tool withdraws following the profile up to the previous pass, "C" distance (right figure).

9.

CANNED CYCLES

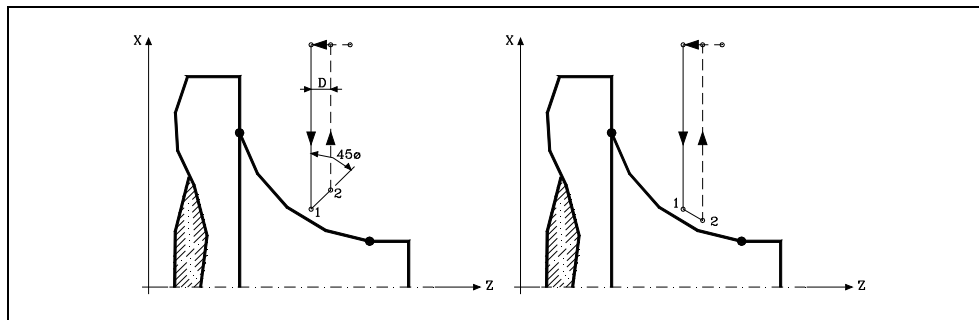
G85. Facing canned cycle for curved sections

FAGOR 

CNC 8037

· T· MODEL
SOFT: V01.4x

9.

CANNED CYCLES
G85. Facing canned cycle for curved sections

It must be borne in mind that when not programming the parameter D, the cycle execution time is greater but the finishing stock is smaller.

L5.5

Defines the finishing stock along X and it is given in radius.

If not programmed, a value of 0 is assumed.

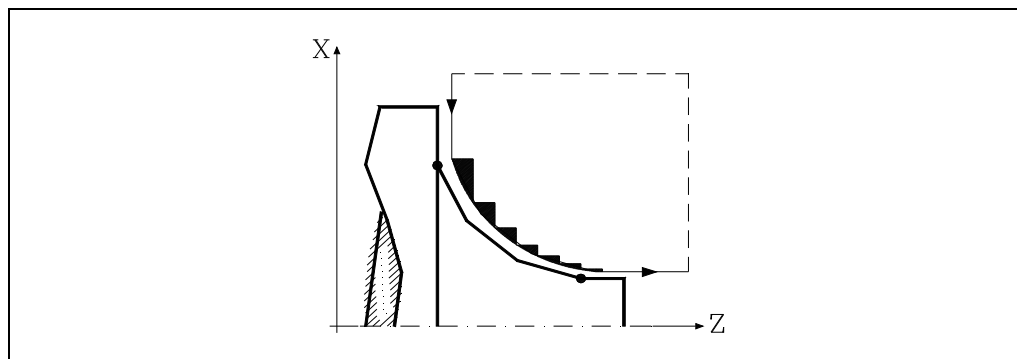
M5.5

Defines the finishing stock along Z.

If not programmed, a value of 0 is assumed.

F5.5

It defines the feedrate for the last roughing pass. If not programmed or programmed with a 0 value, it does not run the last roughing pass.

**H5.5**

It defines the feedrate during the finishing pass.

If not programmed or programmed with a 0 value, it does not run the finishing pass.

I±5.5

Defines the distance, in radius, from the initial point (X, Z) to the arc's center along the X axis. It is programmed in incremental values with respect to the initial point like the "I" for circular interpolations (G02, G03).

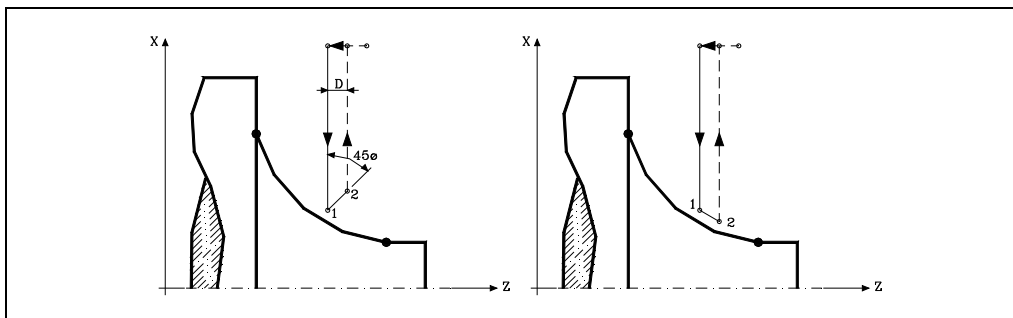
K±5.5

Defines the distance from the initial point (X, Z) to the arc's center along the Z axis. It is programmed in incremental values with respect to the initial point like the "K" for circular interpolations (G02, G03).

9.8.1 Basic operation

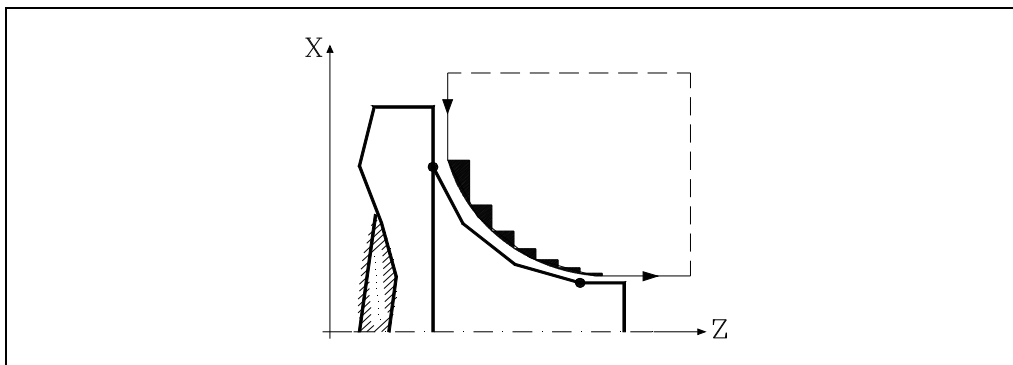
The canned cycle will analyze the programmed profile performing, if necessary, a facing operation until the defined profile is reached.

The whole facing operation is carried out with the same pass, this being equal to or smaller than the programmed one (C). Each facing pass is carried out in the following manner:

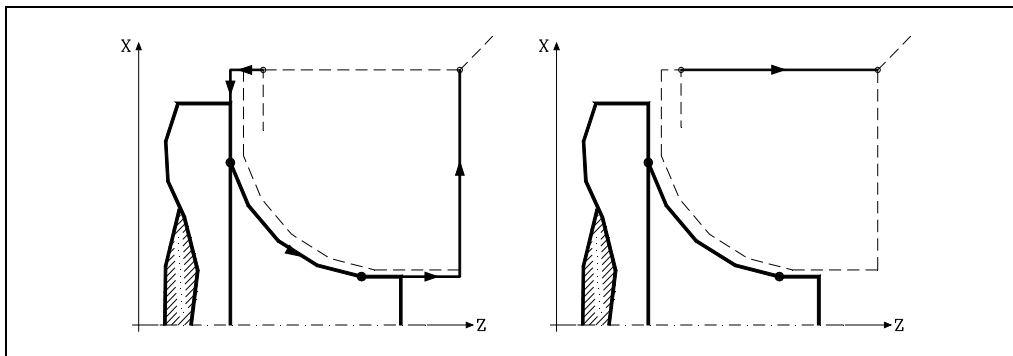


- "1-2" move in rapid (G00).
- Move "2-3" in G01 at programmed feedrate (F).
- If "D" has been programmed, the 3-4 move is done in rapid (G00); but if "D" has not been programmed, the 3-4 move is done following the profile in G01 at the programmed feedrate (F).
- The withdrawal movement "4-5" is made in rapid traverse (G00).

If a final roughing pass has been programmed, the CNC will run a pass parallel to the profile maintaining an "L" and "M" stock at the set feedrate "F". This final roughing pass will eliminate the ridges that were left in the roughing stage.



After the facing operation (with or without finishing pass) the canned cycle will always end at the cycle calling point.



Considerations

The machining conditions (feedrate, spindle speed, etc.) as well as the tool radius compensation (G41, G42) must be programmed before calling the cycle.

Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle.

9.

CANNED CYCLES

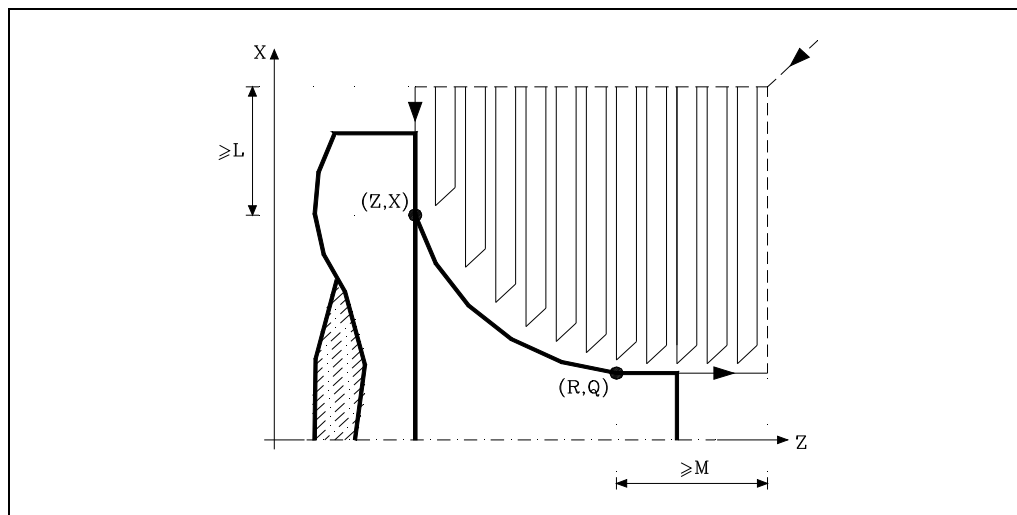
G85. Facing canned cycle for curved sections

FAGOR

CNC 8037

·T· MODEL
SOFT: V01.4x

The distance between the starting point and the initial point (X, Z) along the X axis must be equal to or greater than L. The distance between the starting point and final point (R,Q) along the Z axis must be equal to or greater than M.



If the tool position is not correct to execute the cycle, the CNC will display the corresponding error.

9.

CANNED CYCLES G85. Facing canned cycle for curved sections

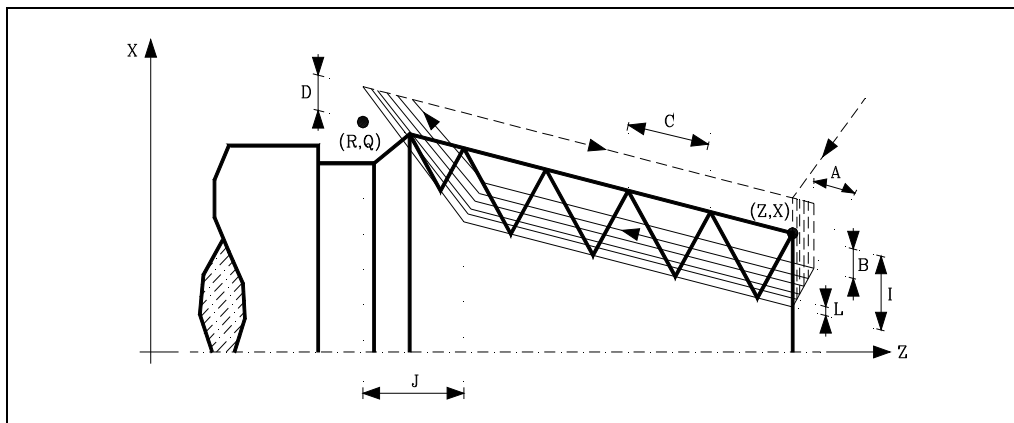
9.9 G86. Longitudinal threading canned cycle

With this cycle it is possible to cut outside, inside and tapered threads.

The right-hand and left-hand threads will be programmed by indicating the direction of the spindle rotation M03 or M04.

The basic structure of the block is:

G86 X Z Q R K I B E D L C J A W V M H U



X±5.5

Defines the X coordinate of the initial point of the thread. It must be programmed in absolute coordinates and in the active units, radius or diameter.

Z±5.5

Defines the Z coordinate of the initial point of the thread. It must be programmed in absolute coordinates.

Q±5.5

Defines the X coordinate of the final point of the thread. It must be programmed in absolute coordinates and in the active units, radius or diameter.

R±5.5

Defines the Z coordinate of the final point of the thread.

K±5.5

Optional. It is used with parameter "W" for thread repair.

Defines the Z coordinate of the point where the thread is to be measured. Usually it is an intermediate point of the thread.

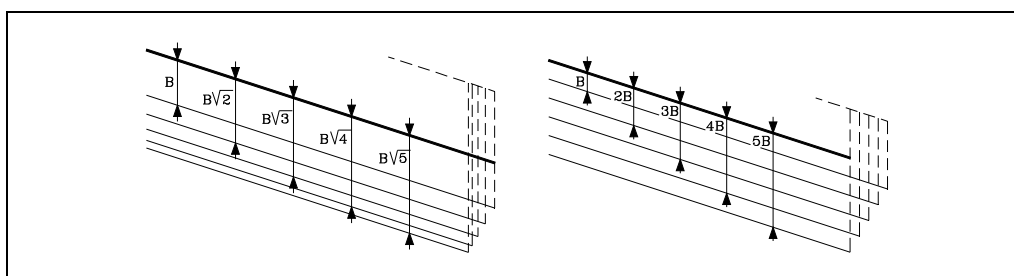
I±5.5

Defines the depth of the thread in radius. Its value must be positive for outside threads and negative for inside ones.

If programmed with a 0 value, the CNC will display the corresponding error message.

B±5.5

Defines the depth of the threading passes and it is given in radius.



9.

CANNED CYCLES
G86. Longitudinal threading canned cycle

FAGOR 

CNC 8037

·T· MODEL
SOFT· V01.4x

- If a positive value is programmed, the depth of each pass will depend on the number of the corresponding pass.

This way, the penetrations along the X axis are:

$$B, B\sqrt{2}, B\sqrt{3}, B\sqrt{4}, \dots, B\sqrt{n}$$

- If a negative value is programmed, the passes will have the same depth with a value equal to the one programmed (B).

This way, the penetrations along the X axis are:

$$B, 2B, 3B, 4B, \dots, nB$$

- If programmed with a 0 value, the CNC will display the corresponding error message.

Regardless of the sign assigned to "B", when the last roughing pass (before finishing) is smaller than the programmed amount, the canned cycle will run a pass equal to the residual stock.

E±5.5

It is related to parameter "B".

It indicates the minimum value to be reached the penetration pitch when parameter "B" has been programmed with a positive value.

If not programmed, a value of 0 is assumed.

D±5.5

Defines the safety distance, along X, and it indicates at what distance from the initial point must the tool be positioned in the approach move. It is programmed in radius.

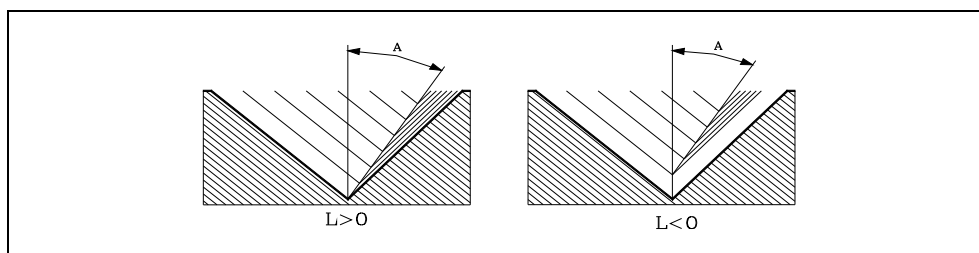
The return to the initial point after each pass will be carried out maintaining this distance (D) from the programmed section.

- If the programmed value is positive, this withdrawal will be performed in round corner (G05) and if negative, in square corner (G07).
- If not programmed, a value of 0 is assumed.

L±5.5

Defines the finishing pass and it is programmed in radius.

- If a positive value is programmed, the finishing pass will be performed maintaining the same penetration angle "A" as for the other passes.

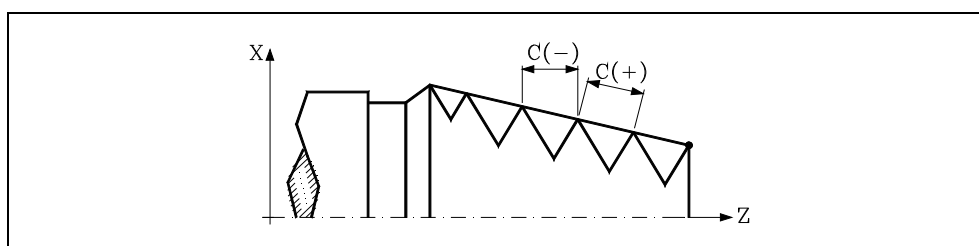


- If a negative value is programmed, the finishing pass will be carried out with radial penetration.
- If a value of 0 is programmed, the previous pass is repeated.

C5.5

Defines the thread's pitch.

- With a positive sign if the pitch is programmed along the taper.
- With a negative sign if the pitch is programmed along the associated axis.



9.

CANNED CYCLES
G86. Longitudinal threading canned cycle



CNC 8037

•T• MODEL
SOFT: V01.4x

If programmed with a 0 value, the CNC will display the corresponding error message.

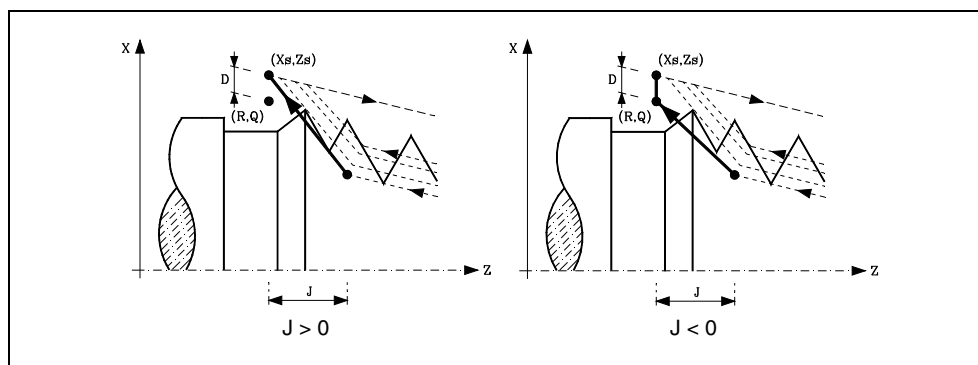
J5.5

Exit from the thread. It defines at what distance from the final point (R, Q), along the Z axis, starts the withdrawal from the thread.

- If programmed with a positive value, the tool moves directly from point "J" to the safety distance X_s, Z_s .
- If programmed with a negative value, the tool moves from point "J" to the end of the thread (R, Q) and then to the safety distance X_s .
- If not programmed, a value of 0 is assumed (blind thread).

The third set of gains and accelerations for the axes and the spindle may be used to improve the adjustment and the machining of the exit from blind threads. If the thread exit is a short distance, any acceleration set may be used or even remove the acceleration without getting the error "insufficient acceleration while threading".

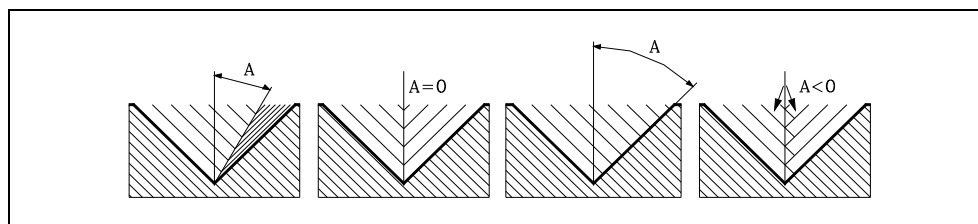
We recommend to use low or no acceleration.



A±5.5

Defines the penetration angle of the tool. It is referred to the X axis and if not programmed, a value of 30° will be taken.

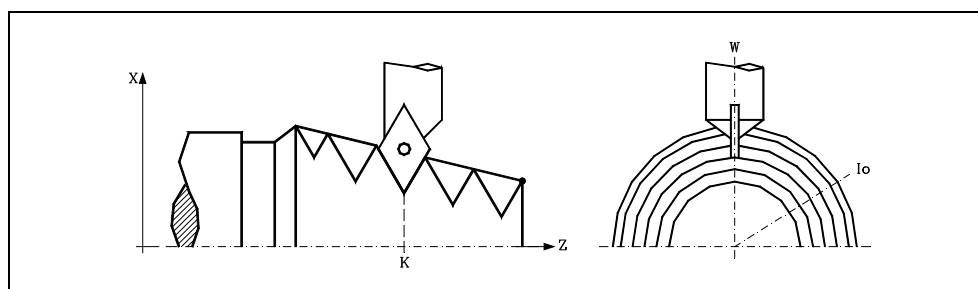
- If $A=0$ is programmed, the thread will be made with radial penetration.
- If the value assigned to parameter "A" is half the tool angle, the penetration will be done rubbing the thread's flank.
- If A is programmed with a negative value, the penetration will be done in zig-zag on the radial axis.



W±5.5

Optional. Its meaning depends on parameter "K".

- If "K" has been defined, it is a thread repair cycle. It indicates the angular spindle position corresponding to the thread measuring point.



- If parameter "K" has not been defined, it indicates the angular position of the spindle corresponding to the thread's starting point. This allows making multi-entry threads without using the parameter "V".

9.

CANNED CYCLES
G86. Longitudinal threading canned cycle

FAGOR

CNC 8037

·T· MODEL
SOFT: V01.4x

The following example shows how to make a 3-entry-point thread. To do that, 3 threading canned cycles will be programmed with the same values except for "W".

```
G86 X Z Q R K I B E D L C J A W0
G86 X Z Q R K I B E D L C J A W120
G86 X Z Q R K I B E D L C J A W240
```

V±5.5

Optional. It sets the desired number of thread entries.

If not programmed or defined with a 0 value, the thread will only have one entry.

When parameter "W" has been set, it is the number of entries to make starting at the spindle angular position defined by that parameter "W".

The following example shows how to make a 3-entry-point thread.

```
G86 X Z Q R K I B E D L C J A W0 V3
```

M±5.5

It defines the thread pitch increment (positive M) or decrement (negative M) per spindle revolution.

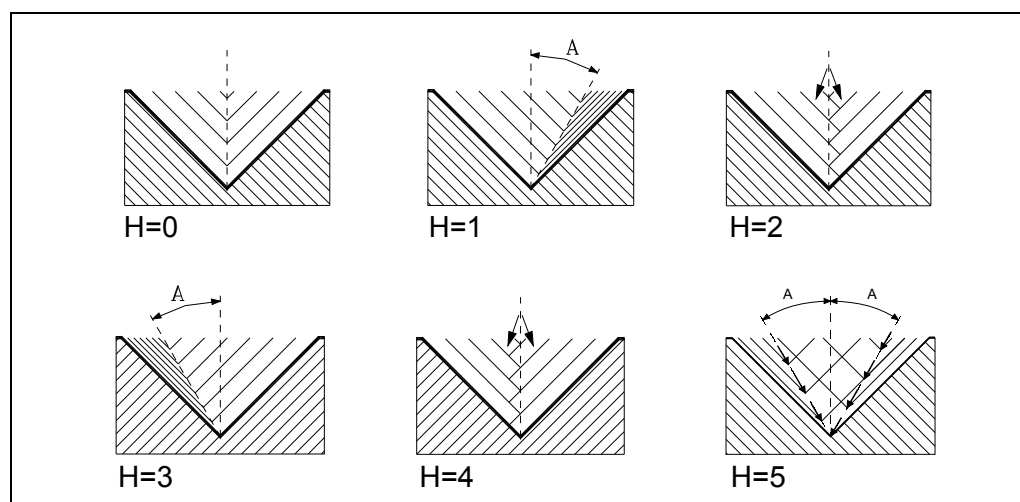
This parameter is incompatible with parameter K (thread repair); therefore, if both parameters are programmed together, the CNC will display the pertinent error message.



It must be borne in mind that if a thread pitch decrement is programmed and the pitch reaches 0 before the end of the thread cutting operation, the CNC will display the corresponding error message.

H1

Optional. Defines thread entry (start) type. When not programming parameter H, the thread entry will be defined with parameter A.



H=0: Radial entry.

The thread entry is radial, even if parameter A has not been set to 0.

H=1: First-flank entry.

The thread entry will be by the first flank and with the absolute angle value given in parameter A.

H=2: Zig-zag radial entry by the first center.

The thread entry will be in radial zig-zag depending on the absolute angle value given in parameter A alternating between radial entry and slightly towards the first flank.

H=3: Last-flank entry.

The thread entry will be by the last flank and with the absolute angle value given in parameter A.

H=4: Zig-zag radial entry by the last center.

The thread entry will be in radial zig-zag depending on the absolute angle value given in parameter A alternating between radial entry and slightly towards the last flank.

H=5: Zig-zag entry by the flanks.

The thread entry will be in zig-zag by the flanks depending on the absolute angle value given in parameter A alternating between the first and last flanks.

Alternating flank-flank, instead of center-flank, extends tool life as compared to radial zig-zag entries (H=2 and H=4).

For this type of entry, the tool being used must have the shape of the thread to be obtained so it fits perfectly at the bottom of the thread. Before executing this cycle, it may be necessary to pre-empty the thread with a regular trapezoidal tool. This way, this cycle will only remove the material left on the flanks of the thread.

U1

Optional. Partial thread repair.

When repairing a lead screw by welding one part of the thread, after that, the lead screw must be repaired using the thread repair cycle.

The thread repair operation is only required in a very small portion of the leadscrew vs its total length. Partial thread repair saves time because it is possible to repair the portion of the lead screw that has been welded.

This option can also be used to machine a thread on a cylinder but entering directly into the cylinder without doing it from the outside.

Using this option requires setting an entry point and an exit point, at intermediate points between the beginning and the exit of the lead screw where the tool goes in and out following a slanted path with respect to the part so as not to damage the thread during the operation.

U=0: When not programmed or programmed with a 0 value, the thread is done beginning in the same direction as its geometry and ending in the same direction or in the direction set for the programmed thread exit.

U=1: The beginning of each pass is done with the necessary angle segment so it reaches the depth of the thread during the first step of the last pass; the end of each pass is done with the necessary angle segment so during it reaches the surface of the thread in the last step of the last pass.

When programming the thread exit, it will be taken into account, otherwise the cycle will calculate the exit path as it does for the entry.

Using the U1 option offers two types of machining:

A partial thread repair, in which case K and W must be programmed.

Make a thread on a cylinder but entering directly into the cylinder without doing it from the outside. In this case, K needs not be programmed.

9.

CANNED CYCLES
G86. Longitudinal threading canned cycle

9.9.1 Basic operation

1. Rapid move (G00) to the approach point located at a safety distance "D" from the initial point (X, Z).
2. Threading loop. The following steps will be repeated until the finishing coordinate is reached; programmed depth "I" minus the finishing stock "L".
 - Rapid move (G00) to the programmed depth indicated by "B". This move will be made according to the penetration angle of the selected tool (A).
 - Performs the threading of the programmed section with the selected exit "J". The feedrate F cannot be varied with the feedrate override switch while threading; it will stay at 100%. At the beginning of the machining operation on large lathes, when cutting long threads, the spindle speed may be overridden in the first passes to prevent the part from bending.
 - Withdrawal in rapid (G00) to the approach point.
3. Thread finishing. Rapid move (G00) to the programmed depth coordinate "I".
This move will be radial or according to the tool penetration angle (A) which depends on the sign assigned to parameter "L".
4. Performs the threading of the programmed section with the selected exit "J".
The feedrate F cannot be varied with the feedrate override switch while threading; it will stay at 100%. It will not be possible to vary the spindle speed override in the last threading pass, it will be set to the value imposed in the previous pass.
5. Withdrawal in rapid (G00) to the approach point.

Thread repair

To do a thread repair, proceed as follows:

1. Home (reference) the spindle.
2. Measure the angle of the root of the thread, K W parameters.
3. Define the G87 cycle for thread repair.
4. Execute the canned cycle.

Considerations

The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the cycle.

Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle. Only the tool radius compensation will be canceled (G40) if it was active.

9.

CANNED CYCLES
G86. Longitudinal threading canned cycle



CNC 8037

•T• MODEL
SOFT: V01.4x

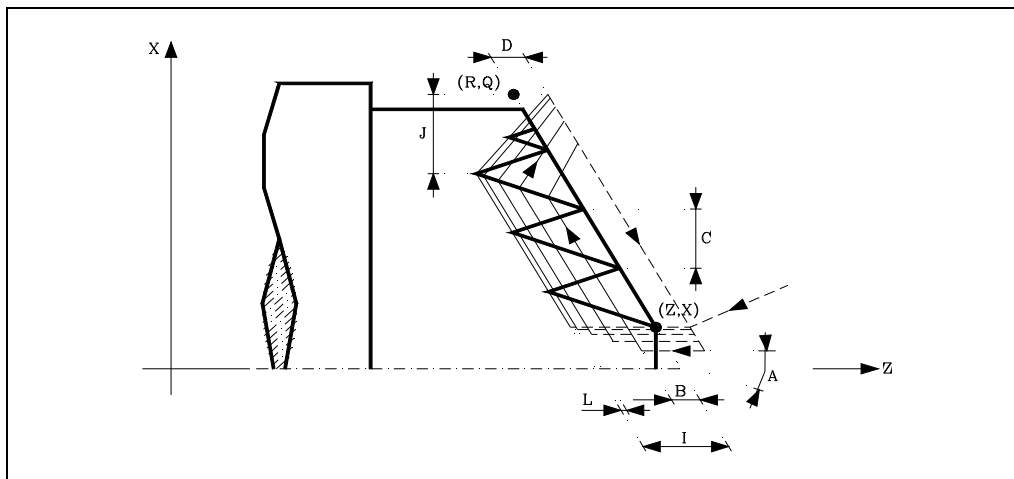
9.10 G87. Face threading canned cycle

With this cycle it is possible to cut outside or inside threads on the face of the part.

The right-hand and left-hand threads will be programmed by indicating the direction of the spindle rotation M03 or M04.

The basic structure of the block is:

G87 X Z Q R K I B E D L C J A W V M H U



X±5.5

Defines the X coordinate of the initial point of the thread. It must be programmed in absolute coordinates and in the active units, radius or diameter.

Z±5.5

Defines the Z coordinate of the initial point of the thread. It must be programmed in absolute coordinates.

Q±5.5

Defines the X coordinate of the final point of the thread. It must be programmed in absolute coordinates and in the active units, radius or diameter.

R±5.5

Defines the Z coordinate of the final point of the thread.

K±5.5

Optional. It is used with parameter "W" for thread repair.

Defines the Z coordinate of the point where the thread is to be measured. Usually it is an intermediate point of the thread.

I±5.5

Defines the depth of the thread. Its value will be positive if threading in the negative Z direction and vice versa.

If programmed with a 0 value, the CNC will display the corresponding error message.

9.

CANNED CYCLES
G87. Face threading canned cycle

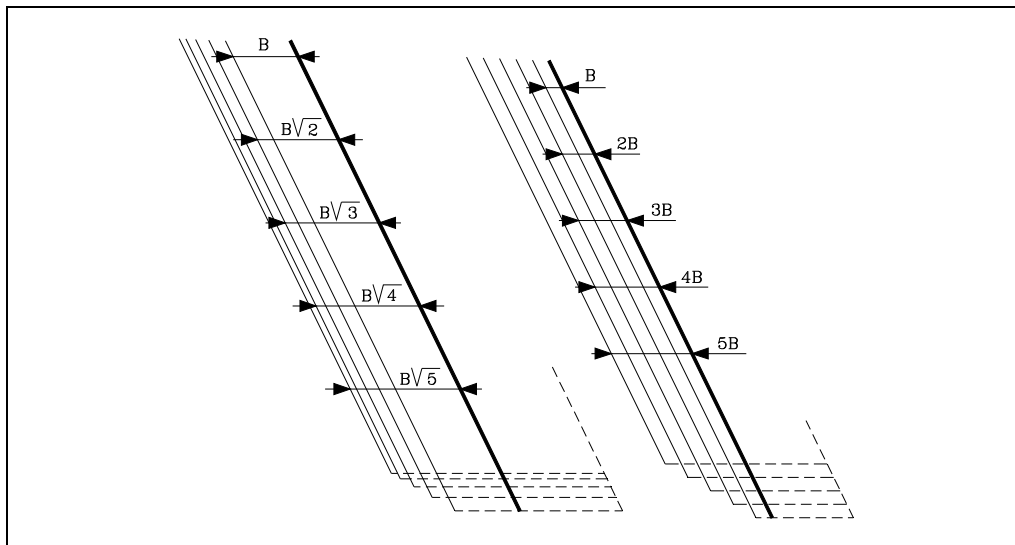
FAGOR 

CNC 8037

·T· MODEL
SOFT: V01.4x

B±5.5

Defines the depth of the threading passes and it is given in radius.



- If a positive value is programmed, the depth of each pass will depend on the number of the corresponding pass.

This way, the penetrations along the Z axis are:

$$B, B/\sqrt{2}, B/\sqrt{3}, B/\sqrt{4}, \dots, B/\sqrt{n}$$

- If a negative value is programmed, the passes will have the same depth with a value equal to the one programmed (B).

This way, the penetrations along the Z axis are:

$$B, 2B, 3B, 4B, \dots, nB$$

- If programmed with a 0 value, the CNC will display the corresponding error message.

Regardless of the sign assigned to "B", when the last roughing pass (before finishing) is smaller than the programmed amount, the canned cycle will run a pass equal to the residual stock.

E±5.5

It is related to parameter "B".

It indicates the minimum value to be reached the penetration pitch when parameter "B" has been programmed with a positive value.

If not programmed, a value of 0 is assumed.

D±5.5

Defines the safety distance and it indicates at what distance, along Z, from the initial point must the tool be positioned in the approach move.

The return to the initial point after each pass will be carried out maintaining this distance (D) from the programmed section.

- If the programmed value is positive, this withdrawal will be performed in round corner (G05) and if negative, in square corner (G07).
- If not programmed, a value of 0 is assumed.

9.

CANNED CYCLES
G87. Face threading canned cycle



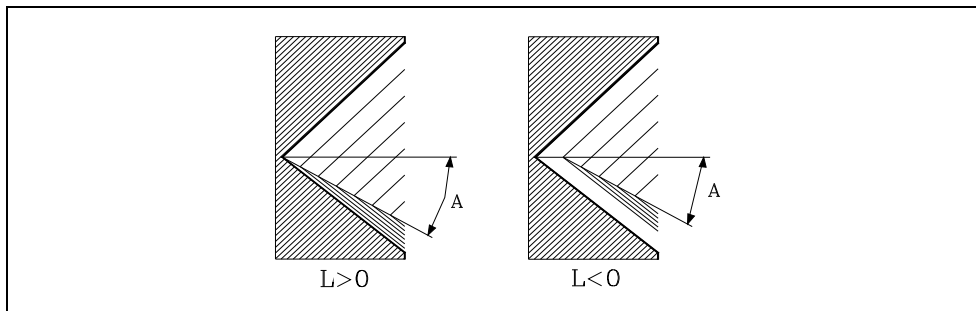
CNC 8037

·T· MODEL
SOFT: V01.4x

L±5.5

Defines the finishing pass

- If a positive value is programmed, the finishing pass will be performed maintaining the same penetration angle "A" as for the other passes.

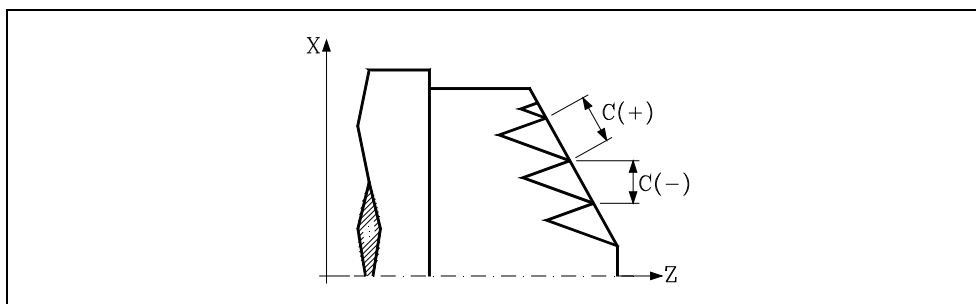


- If a negative value is programmed, the finishing pass will be carried out with radial penetration.
- If a value of 0 is programmed, the previous pass is repeated.

C5.5

Defines the thread's pitch.

- With a positive sign if the pitch is programmed along the taper.
- With a negative sign if the pitch is programmed along the associated axis.



If programmed with a 0 value, the CNC will display the corresponding error message.

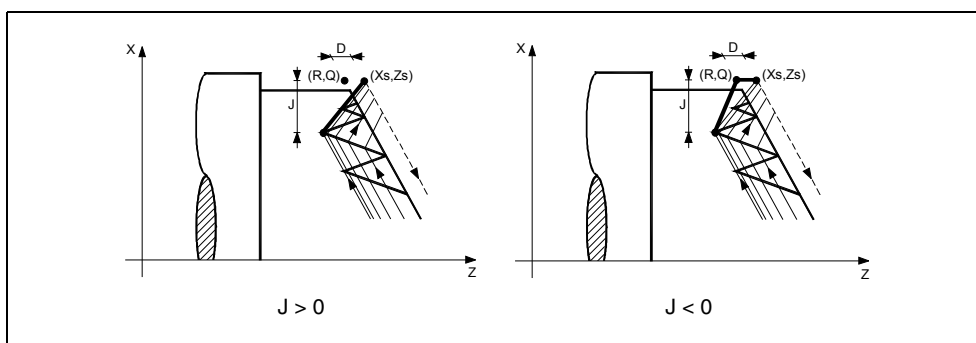
J5.5

Exit from the thread. It defines at what distance from the final point (R, Q), along the Z axis, starts the withdrawal from the thread.

- If programmed with a positive value, the tool moves directly from point "J" to the safety distance X_s, Z_s .
- If programmed with a negative value, the tool moves from point "J" to the end of the thread (R, Q) and then to the safety distance X_s .
- If not programmed, a value of 0 is assumed (blind thread).

The third set of gains and accelerations for the axes and the spindle may be used to improve the adjustment and the machining of the exit from blind threads. If the thread exit is a short distance, any acceleration set may be used or even remove the acceleration without getting the error "insufficient acceleration while threading".

We recommend to use low or no acceleration.



9.

CANNED CYCLES

G87. Face threading canned cycle

FAGOR

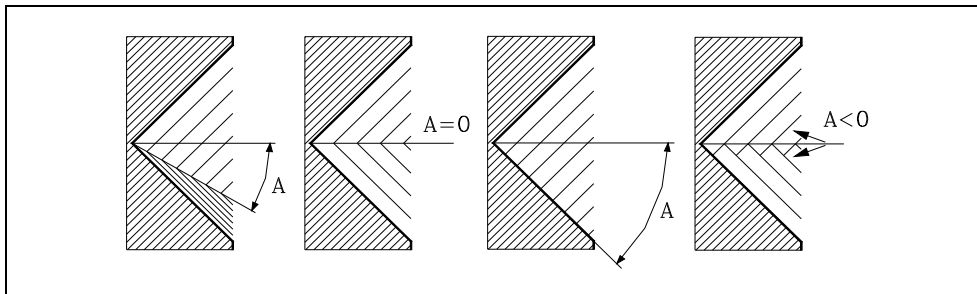
CNC 8037

·T· MODEL
SOFT: V01.4x

A±5.5

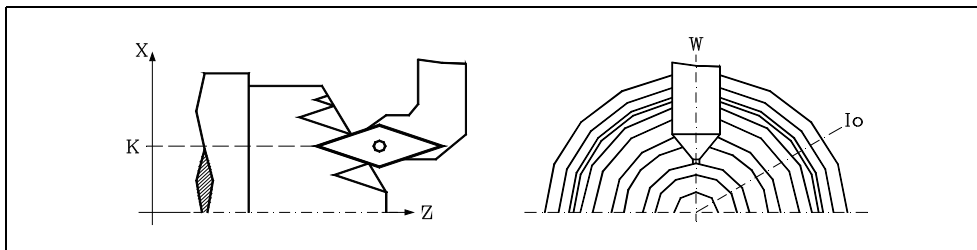
Defines the penetration angle of the tool. It is referred to the X axis and if not programmed, a value of 30° will be taken.

- If A=0 is programmed, the thread will be made with radial penetration.
- If the value assigned to parameter "A" is half the tool angle, the penetration will be done rubbing the thread's flank.
- If A is programmed with a negative value, the penetration will be done in zig-zag on the radial axis.

**W±5.5**

Optional. Its meaning depends on parameter "K".

- If "K" has been defined, it is a thread repair cycle. It indicates the angular spindle position corresponding to the thread measuring point.



- If parameter "K" has not been defined, it indicates the angular position of the spindle corresponding to the thread's starting point. This allows making multi-entry threads without using the parameter "V".

The following example shows how to make a 3-entry-point thread. To do that, 3 threading canned cycles will be programmed with the same values except for "W".

```
G86 X Z Q R K I B E D L C J A W0
G86 X Z Q R K I B E D L C J A W120
G86 X Z Q R K I B E D L C J A W240
```

V±5.5

Optional. It sets the desired number of thread entries.

If not programmed or defined with a 0 value, the thread will only have one entry.

M±5.5

It defines the thread pitch increment (positive M) or decrement (negative M) per spindle revolution.

This parameter is incompatible with parameter K (thread repair); therefore, if both parameters are programmed together, the CNC will display the pertinent error message.



It must be borne in mind that if a thread pitch decrement is programmed and the pitch reaches 0 before the end of the thread cutting operation, the CNC will display the corresponding error message.

9.

CANNED CYCLES
G87. Face threading canned cycle

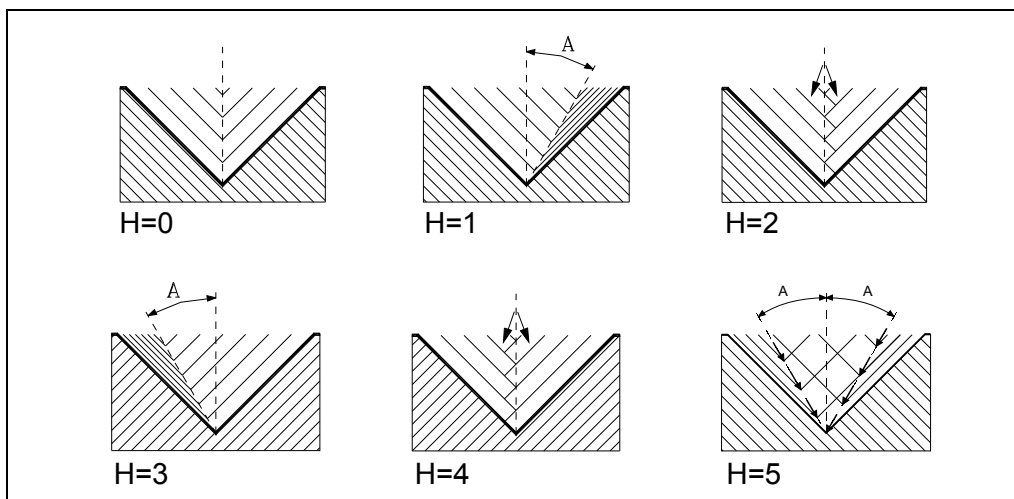


CNC 8037

·T· MODEL
SOFT: V01.4x

H1

Optional. Defines thread entry (start) type. When not programming parameter H, the thread entry will be defined with parameter A.



H=0: Radial entry.

The thread entry is radial, even if parameter A has not been set to 0.

H=1: First-flank entry.

The thread entry will be by the first flank and with the absolute angle value given in parameter A.

H=2: Zig-zag radial entry by the first center.

The thread entry will be in radial zig-zag depending on the absolute angle value given in parameter A alternating between radial entry and slightly towards the first flank.

H=3: Last-flank entry.

The thread entry will be by the last flank and with the absolute angle value given in parameter A.

H=4: Zig-zag radial entry by the last center.

The thread entry will be in radial zig-zag depending on the absolute angle value given in parameter A alternating between radial entry and slightly towards the last flank.

H=5: Zig-zag entry by the flanks.

The thread entry will be in zig-zag by the flanks depending on the absolute angle value given in parameter A alternating between the first and last flanks.

Alternating flank-flank, instead of center-flank, extends tool life as compared to radial zig-zag entries (H=2 and H=4).

For this type of entry, the tool being used must have the shape of the thread to be obtained so it fits perfectly at the bottom of the thread. Before executing this cycle, it may be necessary to pre-empty the thread with a regular trapezoidal tool. This way, this cycle will only remove the material left on the flanks of the thread.

U1

Optional. Partial thread repair.

When repairing a lead screw by welding one part of the thread, after that, the lead screw must be repaired using the thread repair cycle.

The thread repair operation is only required in a very small portion of the leadscrew vs its total length. Partial thread repair saves time because it is possible to repair the portion of the lead screw that has been welded.

This option can also be used to machine a thread on a cylinder but entering directly into the cylinder without doing it from the outside.

Using this option requires setting an entry point and an exit point, at intermediate points between the beginning and the exit of the lead screw where the tool goes in and out following a slanted path with respect to the part so as not to damage the thread during the operation.

9.

CANNED CYCLES

G87. Face threading canned cycle

FAGOR

CNC 8037

T-MODEL
SOFT: V01.4x

9.

CANNED CYCLES

G87. Face threading canned cycle

- U=0: When not programmed or programmed with a 0 value, the thread is done beginning in the same direction as its geometry and ending in the same direction or in the direction set for the programmed thread exit.
- U=1: The beginning of each pass is done with the necessary angle segment so it reaches the depth of the thread during the first step of the last pass; the end of each pass is done with the necessary angle segment so during it reaches the surface of the thread in the last step of the last pass.

When programming the thread exit, it will be taken into account, otherwise the cycle will calculate the exit path as it does for the entry.

Using the U1 option offers two types of machining:

A partial thread repair, in which case K and W must be programmed.

Make a thread on a cylinder but entering directly into the cylinder without doing it from the outside. In this case, K needs not be programmed.

9.10.1 Basic operation

1. Rapid move (G00) to the approach point located at a safety distance "D" from the initial point (X, Z).
2. Threading loop. The following steps will be repeated until the finishing coordinate is reached; programmed depth "I" minus the finishing stock "L".
 - Rapid move (G00) to the programmed depth indicated by "B". This move will be made according to the penetration angle of the selected tool (A).
 - Performs the threading of the programmed section with the selected exit "J". The feedrate F cannot be varied with the feedrate override switch while threading; it will stay at 100%. At the beginning of the machining operation on large lathes, when cutting long threads, the spindle speed may be overridden in the first passes to prevent the part from bending.
 - Withdrawal in rapid (G00) to the approach point.
3. Thread finishing. Rapid move (G00) to the programmed depth coordinate "I".
This move will be radial or according to the tool penetration angle (A) which depends on the sign assigned to parameter "L".
4. Performs the threading of the programmed section with the selected exit "J".
The feedrate F cannot be varied with the feedrate override switch while threading; it will stay at 100%. It will not be possible to vary the spindle speed override in the last threading pass, it will be set to the value imposed in the previous pass.
5. Withdrawal in rapid (G00) to the approach point.

Thread repair

To do a thread repair, proceed as follows:

1. Home (reference) the spindle.
2. Measure the angle of the root of the thread, K W parameters.
3. Define the G87 cycle for thread repair.
4. Execute the canned cycle.

Considerations

The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the cycle.

Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle. Only the tool radius compensation will be canceled (G40) if it was active.

9.

CANNED CYCLES

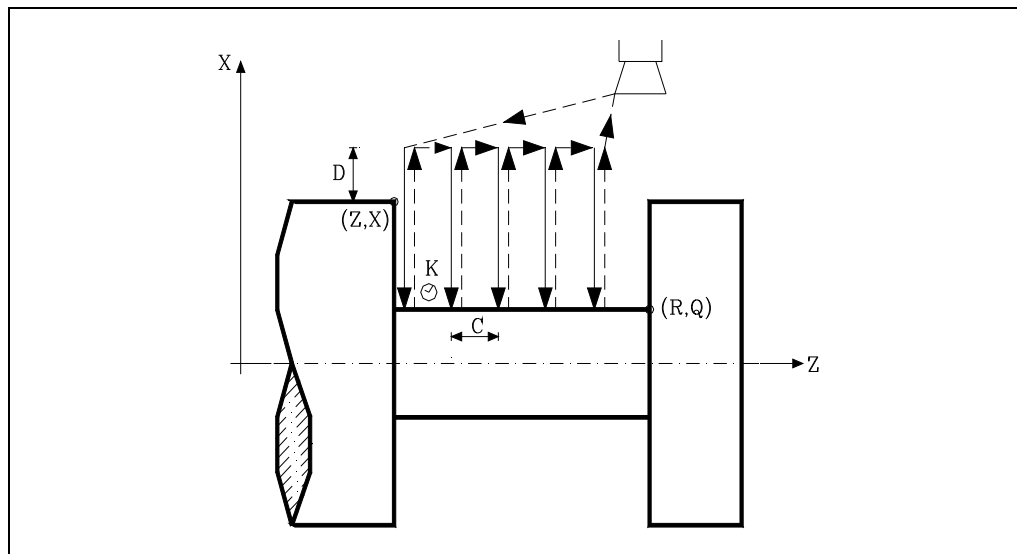
G87. Face threading canned cycle

9.11 G88. X axis grooving canned cycle

This cycle performs a grooving operation along the X axis maintaining the same pass between the successive passes being this equal to or smaller than the one programmed.

The basic structure of the block is:

G88 X Z Q R C D K



X±5.5

Defines the X coordinate of the initial point of the groove. It must be programmed in absolute coordinates and in the active units, radius or diameter.

Z±5.5

Defines the Z coordinate of the initial point of the groove. It must be programmed in absolute coordinates.

Q±5.5

Defines the X coordinate of the final point of the groove. It must be programmed in absolute coordinates and in the active units, radius or diameter.

R±5.5

Defines the Z coordinate of the final point of the groove.

C5.5

Defines the grooving pass.

If not programmed, the CNC will assume the width of the cutter (NOSEW) of the active tool and if programmed with a zero value, the CNC will issue the corresponding error message.

D5.5

Defines the safety distance and it must have a positive value in radius.

K5

Defines the dwell, in hundredths of a second, after each penetration until the withdrawal begins.

If not programmed, a value of 0 is assumed.

9.11.1 Basic operation

The whole grooving operation is done with the same pass being this equal to or smaller than "C". Each grooving pass is carried out in the following manner:

- The penetration is done at programmed feedrate (F).
- The withdrawal and the move to the next penetration point are done in rapid (G00).

After completing the grooving operation, the canned cycle will always end at the cycle calling point.

Considerations

The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the cycle.

Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle. Only the tool radius compensation will be canceled (G40) if it was active.

The tool must be located at a distance, along X, equal to or greater than the one indicated by canned cycle parameter "D" (safety distance) with respect to the part.

If the depth of the groove is 0, the CNC will display the corresponding error message.

If the width of the groove is smaller than the width of the cutter (NOSEW), the CNC will display the corresponding error message.

9.**CANNED CYCLES**

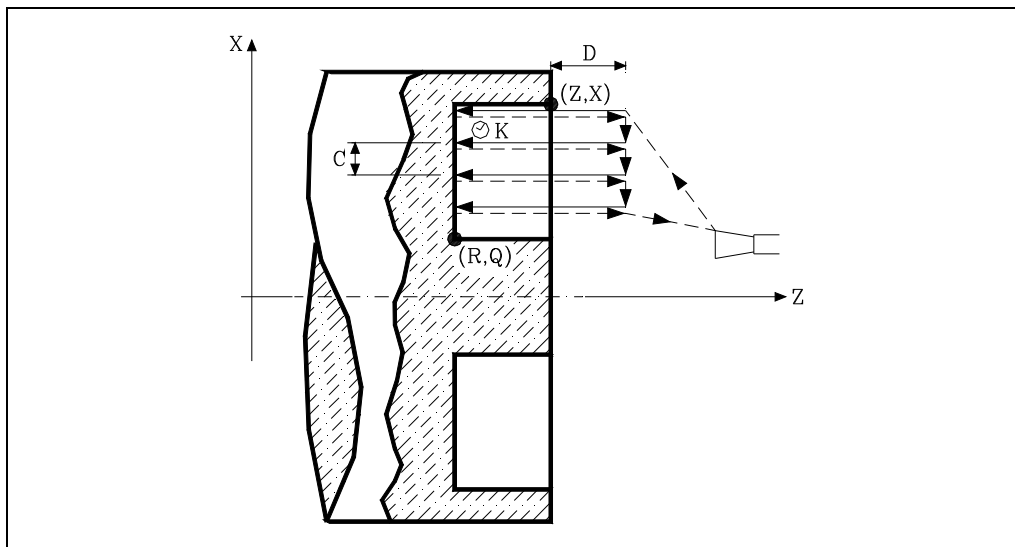
G88. X axis grooving canned cycle

9.12 G89. Z axis grooving canned cycle

This cycle performs a grooving operation along the Z axis maintaining the same pass between the successive passes being this equal to or smaller than the one programmed.

The basic structure of the block is:

G89 X Z Q R C D K



X±5.5

Defines the X coordinate of the initial point of the groove. It must be programmed in absolute coordinates and in the active units, radius or diameter.

Z±5.5

Defines the Z coordinate of the initial point of the groove. It must be programmed in absolute coordinates.

Q±5.5

Defines the X coordinate of the final point of the groove. It must be programmed in absolute coordinates and in the active units, radius or diameter.

R±5.5

Defines the Z coordinate of the final point of the groove.

C5.5

Defines the grooving pass. It is programmed in radius.

If not programmed, the CNC will assume the width of the cutter (NOSEW) of the active tool and if programmed with a zero value, the CNC will issue the corresponding error message.

D5.5

Defines the safety distance.

If not programmed, a value of 0 is assumed.

K5

Defines the dwell, in hundredths of a second, after each penetration until the withdrawal begins.

If not programmed, a value of 0 is assumed.

9.12.1 Basic operation

The whole grooving operation is done with the same pass being this equal to or smaller than "C". Each grooving pass is carried out in the following manner:

- The penetration is done at programmed feedrate (F).
- The withdrawal and the move to the next penetration point are done in rapid (G00).

After completing the grooving operation, the canned cycle will always end at the cycle calling point.

Considerations

The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the cycle.

Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle. Only the tool radius compensation will be canceled (G40) if it was active.

The tool must be located at a distance, along Z, equal to or greater than the one indicated by canned cycle parameter "D" (safety distance) with respect to the part.

If the depth of the groove is 0, the CNC will display the corresponding error message.

If the width of the groove is smaller than the width of the cutter (NOSEW), the CNC will display the corresponding error message.

9.**CANNED CYCLES**

G89. Z axis grooving canned cycle

9.

CANNED CYCLES

G89. Z axis grooving canned cycle



CNC 8037

.T. MODEL
SOFT: V01.4x

The CNC has two probe inputs, one for TTL-type 5Vdc signals and another for 24 Vdc signals.

The connection of the different types of probes to these inputs are explained in the appendix to the Installation manual.

10.1 Probing (G75, G76)

The G75 function allows movements to be programmed that will end after the CNC receives the signal from the measuring probe used.

The G76 function allows movements to be programmed that will end after the CNC no longer receives the signal from the measuring probe used.

Their definition format is:

G75 X..C ± 5.5

G76 X..C ± 5.5

After G75 or G76, the required axis or axes will be programmed, as well as the coordinates of these axes that will define the end point of the programmed movement.

The machine will move according to the programmed path until it receives the signal from the probe (G75) or until it no longer receives the probe signal (G76). At this time, the CNC will consider the block finished, taking as the theoretical position of the axes the real position they have at that time.

If the axes reach the programmed position before receiving or while receiving the external signal from the probe, the CNC will stop the movement of the axes.

This type of movement with probing blocks are very useful when it is required to generate measurement or verification programs for tools and parts.

Functions G75 and G76 are not modal and, therefore, must be programmed whenever it is wished to probe.

Functions G75 and G76 are incompatible with each other and with G00, G02, G03, G33, G34, G41 and G42 functions. In addition, once this has been performed, the CNC will assume functions G01 and G40.

During G75 or G76 moves, the operation of the feedrate override switch depends on the setting of OEM machine parameter FOVRG75.

10.**PROBING**
Probing (G75, G76)**CNC 8037**

·T· MODEL
SOFT: V01.4x

11.1 Lexical description

All the words that make up the high-level language of the numerical control must be written in capital letters except for associated texts which may be written in upper and lower case letters.

The following elements are available for high-level programming:

- Reserved words.
- Numerical constants.
- Symbols.

Reserved words

Reserved words are those that the CNC uses in high-level programming for naming system variables, operators, control instructions, etc.

All the letters of the alphabet A-Z are also reserved words, as they can make up a high-level language word when used alone.

Numerical constants

Blocks programmed in high-level language admit numbers in decimal format and in hexadecimal format.

- The numbers in decimal format must not exceed the ± 6.5 format (6 digits to the left of the decimal point and 5 decimals).
- The numbers in hexadecimal format must be preceded by the \$ symbol and they must not have more than 8 digits.

A constant higher than the format ± 6.5 must be assigned to a variable by means of arithmetic parameters by means of arithmetic expressions or by means of constants in hexadecimal format.

To assign the value 100000000 to the variable "TIMER", It can be done in one of the following ways:

```
(TIMER = $5F5E100)
(TIMER = 10000 * 10000)
(P100 = 10000 * 10000)
(TIMER = P100)
```

If the CNC operates in the metric system (millimeters), the resolution is a tenth of a micron and the figures will be programmed in the ± 5.4 format (positive or negative with 5 digits to the left of the decimal point and 4 decimals).

If the CNC operates in inches, the resolution is a hundred-thousandths of an inch (ten millionths of an inch) and the figures will be programmed in the ± 4.5 format (positive or negative with 4 digits to the left of the decimal point and 5 decimals).

For the convenience of the programmer, this control always allows the format ± 5.5 (positive or negative, with 5 integers and 5 decimals), adjusting each number appropriately to the working units every time they are used.

Symbols

The symbols used in high-level language are:

() " = + - * / ,

11.**HIGH-LEVEL LANGUAGE PROGRAMMING**

Lexical description

**CNC 8037**

.T. MODEL
SOFT: V01.4x

11.2 Variables

The CNC has a number of internal variables that may be accessed from the user program, from the PLC program or via DNC. Depending on how they are used, these variables may be read-only or read-write.

These variables may be accessed from the user program using high-level commands. Each one of these variables is referred to by its mnemonic that must be written in upper-case (capital) letters.

- Mnemonics ending in (X-C) indicate a set of 9 elements formed by the corresponding root followed by X, Y, Z, U, V, W, A, B and C.

ORG(X-C) -> ORGX	ORGY	ORGZ
ORGU	ORGV	ORGW
ORGA	ORGB	ORGC

- Mnemonics ending in *n* indicate that the variables are grouped in tables. To access an element of any of these tables, indicate the field of the desired table using the relevant mnemonic followed by the desired element.

TORn ->	TOR1	TOR3	TOR11
---------	------	------	-------

The variables and block preparation

The variables that access the real values of the CNC interrupt block preparation. The CNC waits for that command to be executed before resuming block preparation. Thus, precaution must be taken when using this type of variable, because if they are inserted between machining blocks that are working with compensation, undesired profiles may be obtained.

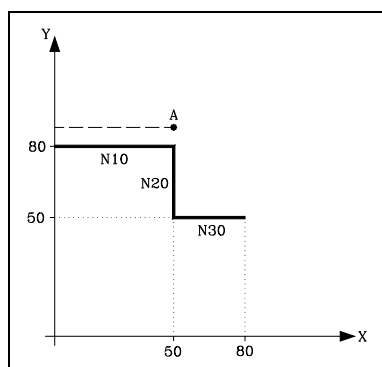
Example: Reading a variable that interrupts block preparation.

The following program blocks are performed in a section with G41 compensation.

```

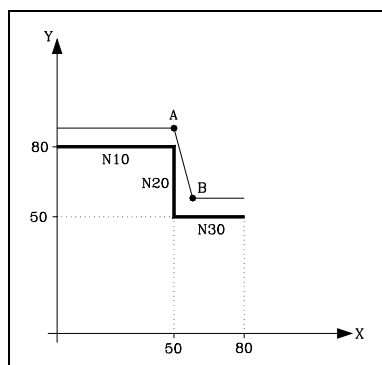
...
N10 X80 Z50
N15 (P100 = POSX); Assigns the value of the real coordinate in X to parameter P100
N20 X50 Z50
N30 X50 Z80
...

```



Block N15 interrupts block preparation and the execution of block N10 will finish at point A.

Once the execution of block N15 has ended, the CNC will resume block preparation from block N20 on.



As the next point corresponding to the compensated path is point "B", the CNC will move the tool to this point, executing path "A-B".

As can be observed, the resulting path is not the desired one, and therefore it is recommended to avoid the use of this type of variable in sections having tool compensation active.

11.

HIGH-LEVEL LANGUAGE PROGRAMMING
Variables

FAGOR 

CNC 8037

·T· MODEL
SOFT: V01.4x

11.2.1 General purpose parameters or variables

General purpose variables are referred to with the letter "P" followed by an integer number. The CNC has four types of general purpose variables.

Parameter type	Range
Local parameters	P0-P25
Global parameters	P100-P299
User parameters	P1000-P1255
OEM (manufacturer's) parameters	P2000-P2255

Blocks programmed in ISO code allow associating parameters to the G F S T D M fields and coordinates of the axes. The block label number must be defined with a numeric value. If parameters are used in blocks programmed in high-level language, they can be programmed within any expression.

Programmers may use general purpose variables when editing their own programs. Later on, during execution, the CNC will replace these variables with the values assigned to them at the time.

When programming...	When executing...
GP0 XP1 Z100 (IF (P100 * P101 EQ P102) GOTO N100)	G1 X-12.5 Z100 (IF (2 * 5 EQ 12) GOTO N100)

Using these general purpose variables will depend on the type of block in which they are programmed and the channel of execution. Programs that are executed in the user channel may contain any global, OEM or user parameter, but may not use local parameters.

Types of arithmetic parameters

Local parameters

Local parameters can only be accessed from the program or subroutine where they have been programmed. There are seven groups of parameters.

Local parameters used in high-level language may be defined either using the above format or by using the letter A-Z, except for N, so that A is equal to P0 and Z to P25.

The following example shows these two methods of definition:

```
(IF ((P0+P1)* P2/P3 EQ P4) GOTO N100)
(IF ((A+B)* C/D EQ E) GOTO N100)
```

When using a parameter name (letter) for assigning a value to it (A instead of P0, for example), if the arithmetic expression is a constant, the instruction can be abbreviated as follows:

```
(P0=13.7) ==> (A=13.7) ==> (A13.7)
```

Be careful when using parenthesis since M30 is not the same as (M30). The CNC interprets (M30) as a high level instruction meaning (P12 = 30) and not the execution of the miscellaneous M30 function.

Global parameters

Global parameters can be accessed from any program and subroutine called from a program.

Global parameters may be used by the user, by the OEM or by the CNC cycles.

User parameters

These parameters are an expansion of the global parameters but they are not used by the CNC cycles.

OEM (manufacturer's) parameters

OEM parameters and subroutines with OEM parameters can only be used in OEM programs; those defined with the [O] attribute. Modifying one of these parameters in the tables requires an OEM password.

11.

HIGH-LEVEL LANGUAGE PROGRAMMING
Variables



CNC 8037

.T- MODEL
SOFT: V01.4X

Using arithmetic parameters by the cycles

Multiple machining cycles (G60 through G65) and the machining canned cycles (G69, G81 to G89) use the sixth nesting level of local parameters when they are active.

Machining canned cycles use the global parameter P299 for internal calculations and probing canned cycles use global parameters P294 to P299.

Updating arithmetic parameter tables

The CNC will update the parameter table after processing the operations indicated in the block that is in preparation. This operation is always done before executing the block and for this reason, the values shown in the table do not necessarily have to correspond to the block being executed.

If the execution mode is abandoned after interrupting the execution of the program, the CNC will update the parameter tables with values corresponding to the block that was being executed.

When accessing the local parameter and global parameter table, the value assigned to each parameter may be expressed in decimal notation (4127.423) or in scientific notation (0.23476 E-3).

Arithmetic parameters in the subroutines

The CNC offers high level instructions for defining and using subroutines that may be called upon from the main program (or a subroutine) and another subroutine from this one and so on, the CNC limits the number of these calls to a maximum of 15 nesting levels. The CNC limitation is 20 nesting levels.

Up to 26 (P0-P25) local parameters may be assigned to a subroutine. These parameters, that will be unknown for blocks outside the subroutine, may be referred to by the blocks that make up the subroutine.

Local parameters may be assigned to more than one subroutine up to 6 parameter nesting levels within the 15 subroutine nesting levels.

11.**HIGH-LEVEL LANGUAGE PROGRAMMING**

Variables

**CNC 8037**·T· MODEL
SOFT: V01.4x

11.2.2 Variables associated with tools.

These variables are associated with the tool offset table, tool table and tool magazine table, so the values which are assigned to or read from these fields will comply with the formats established for these tables.

Tool Offset table

The radius (R), length (L) and wear offset (I, K) values of the tool are given in the active units.

If G70, in inches (within ± 3937.00787).

If G71, in millimeters (within ± 99999.9999).

If rotary axis in degrees (within ± 99999.9999).

The location (tool shape) code (F) will be an integer between 0 and 9.

Tool table

The tool offset number is an integer between 0 and 255. The maximum number of tool offsets is limited by g.m.p. NTOFFSET.

The family code is a number between 0 and 255.

0 to 199 if it is a normal tool.

200 to 255 if it is a special tool.

The nominal life is given either in minutes or in operations (0..65535).

The real (actual) life is given either in hundredths of a minute (0..999999) or in operations (0..999999).

The cutter angle is given in ten-thousandths of a degree (0..359999).

The cutter width is given in the active units.

If G70, in inches (within ± 3937.00787).

If G71, in millimeters (within ± 99999.9999).

If rotary axis in degrees (within ± 99999.9999).

The cutting angle is given in ten-thousandths of a degree (0..359999).

Tool magazine table

Each magazine position is represented as follows:

1..255	Tool number.
0	The magazine position is empty.
-1	The magazine position has been canceled.

The tool position in the magazine is represented as follows:

1..255	Position number.
0	The tool is in the spindle.
-1	Tool not found.
-2	The tool is in the change position.

Read-only variables

TOOL

Returns the number of the active tool.

(P100=TOOL)

Assigns the number of the active tool to parameter P100.

TOD

Returns the number of the active tool offset.

11.

HIGH-LEVEL LANGUAGE PROGRAMMING
Variables



CNC 8037

.T- MODEL
SOFT: V01.4x

NXTOOL

Returns the next tool number, which is selected but is awaiting the execution of M06 to be active.

NXTOD

Returns the number of the tool offset corresponding to the next tool, which is selected but is awaiting the execution of M06 to be active.

TMZPn

Returns the position occupied in the tool magazine by the indicated tool (n).

Read-and-write variables

TOXn

This variable allows reading or modifying the length value along the X axis assigned to the indicated tool offset (n).

(P110=TOX3)

Assigns the X value of tool offset ·3· to Parameter P110.

(TOX3=P111)

Assigns the value of parameter P111 to the X value of tool offset ·3·.

TOZn

This variable allows reading or modifying the length value along the Z axis assigned to the indicated tool offset (n).

TOFn

This variable allows reading or modifying the location code (F) of the indicated tool offset (n).

TORn

This variable allows reading or modifying the radius R value of the indicated tool offset (n).

TOIn

This variable allows reading or modifying the length wear value (I) along the X axis of the indicated tool offset (n).

TOKn

This variable allows reading or modifying the length wear value (K) along the Z axis of the indicated tool offset (n).

NOSEAn:

This variable allows reading or modifying the cutter angle assigned to the indicated tool (n) in the tool table.

NOSEWn

This variable allows reading or modifying the cutter width assigned to the indicated tool (n) in the tool table.

CUTAn

This variable allows reading or modifying the cutting angle assigned to the indicated tool (n) in the tool table.

TLFDn

This variable allows the tool offset number of the indicated tool (n) to be read or modified in the tool table.

11.

HIGH-LEVEL LANGUAGE PROGRAMMING
Variables



CNC 8037

·T· MODEL
SOFT: V01.4x

TLFFn

This variable allows the family code of the indicated tool (n) to be read or modified in the tool table.

TLFNn

This variable allows the value assigned as the nominal life of the indicated tool (n) to be read or modified in the tool table.

TLFRn

This variable allows the value corresponding to the real life of the indicated tool (n) to be read or modified in the tool table.

TMZTn

This variable allows the contents of the indicated position (n) to be read or modified in the tool magazine table.

HTOR

The HTOR variable indicates the tool radius being used by the CNC to do the calculations.

Being a variable that can be read and written by the CNC and read-only from the PLC and DNC, its value may be different from the one assigned in the table (TOR).

On power-up, after a T function, after a RESET or after an M30 function, it assumes the value of the table (TOR).

Application example

To machine a profile with a residual stock of 0.5 mm running 0.1mm-passes with a tool whose radius is 10 mm.

Assign to the tool radius the value of:

- 10.5 mm in the table and execute the profile.
- 10.4 mm in the table and execute the profile.
- 10.3 mm in the table and execute the profile.
- 10.2 mm in the table and execute the profile.
- 10.1 mm in the table and execute the profile.
- 10.0 mm in the table and execute the profile.

However, if while machining, the program is interrupted or a reset occurs, the table assumes the radius value assigned in that instant (e.g.: 10.2 mm). Its value has changed.

To avoid this, instead of modifying the tool radius in the (TOR) table, use the variable (HTOR) to change the tool radius value used by the CNC to calculate.

Now, if the program is interrupted, the tool radius value initially assigned in the (TOR) table will be correct because it has not changed.

11.

HIGH-LEVEL LANGUAGE PROGRAMMING
Variables

CNC 8037

·T· MODEL
SOFT: V01.4x

11.2.3 Variables associated with zero offsets.

These variables are associated with the zero offsets and may correspond to the table values or to those currently preset either by means of function G92 or manually in the JOG mode.

The possible zero offsets in addition to the additive offset indicated by the PLC, are G54, G55, G56, G57, G58 and G59.

The values for each axis are given in the active units:

If G70, in inches (within ± 3937.00787).

If G71, in millimeters (within ± 99999.9999).

If rotary axis in degrees (within ± 99999.9999).

Although there are variables which refer to each axis, the CNC only allows those referring to the axes selected at the CNC. Thus, if the CNC controls the X and Z axes, it only allows the variables ORGX and ORGZ in the case of ORG(X-C).

Read-only variables

ORG(X-C)

Returns the value of the active zero offset in the selected axis. The value of the additive offset indicated by the PLC or by the additive handwheel is not included in this value.

(P100=ORGX)

It assigns to P100 the X value of the part zero active for the X axis. This value could have been set either manually, by means of function G92 or by the variable "ORG(X-C)n".

PORGF

Returns the abscissa value of the polar coordinate origin with respect to the Cartesian origin.

This value is given in radius or diameter depending on the setting of axis machine parameter "DFORMAT".

PORGS

Returns the ordinate value of the polar coordinate origin with respect to the Cartesian origin.

This value is given in radius or diameter depending on the setting of axis machine parameter "DFORMAT".

ADIOF(X-C)

It returns the value of the zero offset generated by the additive handwheel in the selected axis.

Read-and-write variables

ORG(X-C)n

This variable allows the value of the selected axis to be read or modified in the table corresponding to the indicated zero offset (n).

(P110=ORGX 55)

Loads parameter P100 with the X value of G55 in the zero offset table.

(ORGZ 54=P111)

It assigns parameter P111 to the Z axis in the table for the G54 offset.

PLCOF(X-C)

This variable allows the value of the selected axis to be read or modified in the table of additive offsets indicated by the PLC.

Accessing any of the PLCOF(X-C) variables interrupts block preparation and the CNC waits for that command to be executed before resuming block preparation.

11.

11.2.4 Variables associated with machine parameters

These variables associated with machine parameters are read-only variables. These variables may be read and written when executed inside an OEM program or subroutine.

Refer to the installation and start-up manual to know the format of the values returned. The values of 1/0 correspond to the parameters that are set as YES/NO, +/- or ON/OFF.

The coordinate and feedrate values are given in the active units:

If G70, in inches (within ± 3937.00787).

If G71, in millimeters (within ± 99999.9999).

If rotary axis in degrees (within ± 99999.9999).

Modify the machine parameters from an OEM program or subroutine.

These variables may be read and written when executed inside an OEM program or subroutine. In this case, these variables can be used to modify the value of certain machine parameters. Refer to the installation manual for the list of machine parameters that may be modified.

In order to be able to modify these parameters via PLC, an OEM subroutine containing the relevant variables must be executed using the CNC EX command.

Read-only variables

MPGn

Returns the value assigned to general machine parameter (n).

(P110=MPG8)

It assigns the value of the general machine parameter P8 "INCHES" to parameter P110, if millimeters P110=0 and if inches P110=1.

MP(X-C)n

Returns the value assigned to the machine parameter (n) of the indicated axis (X-C).

(P110=MPY 1)

Assigns the value of Y axis machine parameter P1 "DFORMAT" to parameter P110.

MPSn

Returns the value assigned to the indicated machine parameter (n) of the main spindle.

MPLCn

Returns the value assigned to the indicated machine parameter (n) of the PLC.

11.2.5 Variables associated with work zones

Variables associated with work zones are read-only variables.

The values of the limits are given in the active units:

If G70, in inches (within ± 3937.00787).

If G71, in millimeters (within ± 99999.9999).

If rotary axis in degrees (within ± 99999.9999).

The status of the work zones are defined according to the following code:

0 = Disabled.

1 = Enabled as no-entry zone.

2 = Enabled as no-exit zone.

Read-only variables

FZONE

It returns the status of work zone 1.

FZLO(X-C)

Lower limit of zone 1 along the selected axis (X-C).

FZUP(X-C)

Upper limit of zone 1 along the selected axis (X-C).

(P100=FZONE)	; It assigns to parameter P100 the status of work zone 1.
(P101=FZLOX)	; It assigns the lower limit of zone 1 to parameter P101.
(P102=FZUPZ)	; It assigns the upper limit of zone 1 to parameter P102.

SZONE

Status of work zone 2.

SZLO(X-C)

Lower limit of zone 2 along the selected axis (X-C).

SZUP(X-C)

Upper limit of zone 2 along the selected axis (X-C).

TZONE

Status of work zone 3.

TZLO(X-C)

Lower limit of zone 3 along the selected axis (X-C).

TZUP(X-C)

Upper limit of zone 3 along the selected axis (X-C).

FOZONE

Status of work zone 4.

FOZLO(X-C)

Lower limit of zone 4 along the selected axis (X-C).

FOZUP(X-C)

Upper limit of zone 4 along the selected axis (X-C).

11.

FIZONE

Status of work zone 5.

FIZLO(X-C)

Lower limit of zone 5 along the selected axis (X-C).

FIZUP(X-C)

Upper limit of zone 5 along the selected axis (X-C).

11.**HIGH-LEVEL LANGUAGE PROGRAMMING**

Variables

**CNC 8037**

.T- MODEL
SOFT: V01.4x

11.2.6 Variables associated with feedrates

Read-only variables associated with the real (actual) feedrate

FREAL

It returns the CNC's real feedrate. In mm/minute or inches/minute.

(P100=FREAL)

It assigns the real feedrate value of the CNC to parameter P100.

FREAL(X-C)

It returns the actual (real) CNC feedrate of the selected axis.

FTEO(X-C)

It returns the theoretical CNC feedrate of the selected axis.

Read-only variables associated with function G94

FEED

It returns the feedrate selected at the CNC by function G94. In mm/minute or inches/minute.

This feedrate may be indicated by program, by PLC or by DNC; the CNC selects one of them, the one indicated by DNC has the highest priority and the one indicated by program has the lowest priority.

DNCF

It returns the feedrate, in mm/minute or inches/minute selected by DNC. If it has a value of 0 it means that it is not selected.

PLCF

It returns the feedrate, in mm/minute or inches/minute selected by PLC. If it has a value of 0 it means that it is not selected.

PRGF

It returns the feedrate, in mm/minute or inches/minute selected by program.

Read-only variables associated with function G95

FPREV

It returns the feedrate selected at the CNC by function G95. In mm/turn or inches/turn.

This feedrate may be indicated by program, by PLC or by DNC; the CNC selects one of them, the one indicated by DNC has the highest priority and the one indicated by program has the lowest priority.

DNCFPR

It returns the feedrate, in mm/turn or inches/turn selected by DNC. If it has a value of 0 it means that it is not selected.

PLCFPR

It returns the feedrate, in mm/turn or inches/turn selected by PLC. If it has a value of 0 it means that it is not selected.

11.

PRGFPR

It returns the feedrate, in mm/turn or inches/turn selected by program.

Read-only variables associated with function G32

PRGFIN

It returns the feedrate selected by program, in 1/min.

Likewise, the CNC variable FEED, associated with G94, indicates the resulting feedrate in mm/min or inches/min.

Read-only variables associated with the override

FRO

It returns the feedrate override (%) currently selected at the CNC. It is given in integer values between 0 and "MAXFOVR" (maximum 255).

This feedrate percentage may be indicated by program, by PLC, by DNC or by the front panel; the CNC selects one of them and the priority (from the highest to the lowest) is: by program, by DNC, by PLC and from the front panel switch.

DNCFRO

It returns the feedrate override % currently selected by the DNC. If it has a value of 0 it means that it is not selected.

PLCFRO

It returns the feedrate override % currently selected by the PLC. If it has a value of 0 it means that it is not selected.

CNCFRO

It returns the feedrate override % currently selected by the switch.

PLCCFR

It returns the feedrate percentage currently selected by the PLC's execution channel.

Read-write variables associated with the override

PRGFRO

This variable may be used to read or modify the feedrate override percentage currently selected by program. It is given in integer values between 0 and "MAXFOVR" (maximum 255). If it has a value of 0 it means that it is not selected.

(P110=PRGFRO)

It assigns to P110 the % of feedrate override selected by program.

(PRGFRO=P111)

It sets the feedrate override % selected by program to the value of P111.

11.

HIGH-LEVEL LANGUAGE PROGRAMMING
Variables



CNC 8037

·T· MODEL
SOFT: V01.4x

11.2.7 Variables associated with coordinates

The coordinate values for each axis are given in the active units:

If G70, in inches (within ± 3937.00787).

If G71, in millimeters (within ± 99999.9999).

If rotary axis in degrees (within ± 99999.9999).

Read-only variables

Accessing any of the variables POS(X-C), TPOS(X-C), APOS(X-C), ATPOS(X-C) or FLWE(X-C) interrupts block preparation and the CNC waits for that command to be executed before resuming block preparation.

The coordinates provided by the variables PPOS(X-C), POS(X-C), TPOS(X-C), APOS(X-C) and ATPOS(X-C) are given in the active units system (radius or diameter). Check the DIAM variable to know the active units system.

PPOS(X-C)

Returns the programmed theoretical coordinate of the selected axis.

(P110=PPOSX)

It assigns to P100 the programmed theoretical position of the X axis.

POS(X-C)

It returns the real tool base position value referred to machine reference zero (home).

On limit-less rotary axes, this variable takes into account the value of the active zero offset. The values of the variable are between the active zero offset and $\pm 360^\circ$ ($ORG^* \pm 360^\circ$).

If $ORG^* = 20^\circ$ it displays between 20° and 380° / displays between -340° and 20° .

If $ORG^* = -60^\circ$ it displays between -60° and 300° / displays between -420 and -60°

TPOS(X-C)

It returns the theoretical position value (real coordinate + following error) of the tool base referred to machine reference zero (home).

On limit-less rotary axes, this variable takes into account the value of the active zero offset. The values of the variable are between the active zero offset and $\pm 360^\circ$ ($ORG^* \pm 360^\circ$).

If $ORG^* = 20^\circ$ it displays between 20° and 380° / displays between -340° and 20° .

If $ORG^* = -60^\circ$ it displays between -60° and 300° / displays between -420 and -60°

APOS(X-C)

It returns the real tool base position value, referred to part zero, of the selected axis.

ATPOS(X-C)

It returns the theoretical position value (real coordinate + following error) of the tool base referred to part zero.

FLWE(X-C)

It returns the following error of the selected axis.

DPLY(X-C)

It returns the position value (coordinate) shown on the screen for the selected axis.

GPOS(X-C)n p

Programmed coordinate for a particular axis in the indicated (n) block of the (p) program.

11.

(P80=GPOSX N99 P100)

It assigns to parameter P88 the value of the coordinate programmed for the X axis in the block having the label N99 and located in program P100.

Only programs located in the CNC's RAM memory may be consulted.

If the defined program or block does not exist, it shows the relevant error message. If the indicated block does not contain the requested axis, it returns the value 100000.0000.

11.

HIGH-LEVEL LANGUAGE PROGRAMMING Variables

Read-and-write variables

DIST(X-C)

These variables may be used to read or modify the distance traveled by the selected axis. This value is accumulative and is very useful when it is required to perform an operation which depends on the distance traveled by the axes, their lubrication for example.

(P110=DISTX)

It assigns to P110 the distance traveled by the X axis

(DISTX=P111)

It presets the variable indicating the distance traveled by the Z axis with the value of arithmetic parameter P111.

Accessing any of the DIST(X-C) variables interrupts block preparation and the CNC waits for that command to be executed before resuming block preparation.

LIMPL(X-C)

LIMMI(X-C)

With these variables, it is possible to set a second travel limit for each axis: LIMPL for the upper limit and LIMMI for the lower one.

Since the second limits are activated or deactivated from the PLC, through general logic input ACTLIM2 (M5052), besides setting the limits, an auxiliary M code must be executed to let it know.

It is also recommended to execute function G4 after the change so the CNC executes the following blocks with the new limits.

The second travel limit will be taken into account if the first one has been set using axis machine parameters LIMIT+ (P5) and LIMIT- (P6).

11.2.8 Variables associated with electronic handwheels

Read-only variables

HANPF

HANPS

HANPT

HANPFO

They return the pulses of the first (HANPF), second (HANPS), third (HANPT) or fourth (HANPFO) handwheel received since the CNC was turned on. It is irrelevant to have the handwheel connected to the feedback inputs or to the PLC inputs.

HANDSE

For handwheels with axis selector button, it indicates whether that button has been pressed or not. A value of 0 means that it has not been pressed.

HANFCT

It returns the multiplying factor set by PLC for each handwheel.

It must be used when using several electronic handwheels or when using a single handwheel but different multiplying factors (x1, x10, x100) are to be applied to each axis.

C			B			A			W			V			U			Z			Y			X			lsb
c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	

Once the switch has been turned to one of the handwheel positions, the CNC checks this variable and, depending on the values assigned to each axis bit (c, b, a) it applies the multiplying factor selected for each one of them.

c	b	a	
0	0	0	The value indicated at the front panel or keyboard switch.
0	0	1	x1 factor
0	1	0	x10 factor
1	0	0	x100 factor

If there are more than one bit set to "1" for an axis, the least significant bit will be considered. Thus:

c	b	a	
1	1	1	x1 factor
1	1	0	x10 factor



The screen always shows the value selected at the switch.

HBEVAR

It must be used when having a Fagor HBE handwheel.

It indicates whether the HBE handwheel is enabled or not, the axis to be jogged and the multiplying factor to be applied (x1, x10, x100).

			C			B			A			W			V			U			Z			Y			X			lsb
*	^		c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	

(*) Indicates whether the HBE handwheel pulses will be taken into account or not in jog mode.

0 = They are ignored.

1 = They are taken into account.

11.

HIGH-LEVEL LANGUAGE PROGRAMMING
Variables

FAGOR

CNC 8037

·T· MODEL
SOFT: V01.4x

(^) When the machine has a general handwheel and individual handwheels (associated with an axis), it indicates which handwheel has priority when both are turned at the same time.

0 = The individual handwheel has priority. The relevant axis ignores the pulses from the general handwheel, the rest of the axes don't.

1 = The general handwheel has priority. It ignores the pulses from the individual handwheel.

(a, b, c) Indicate the axis to be moved and the selected multiplying factor.

c	b	a	
0	0	0	The value indicated at the front panel or keyboard switch.
0	0	1	x1 factor
0	1	0	x10 factor
1	0	0	x100 factor

If several axes are selected, the following order of priority is applied: X, Y, Z, U, V, W, A, B, C.

If there are more than one bit set to "1" for an axis, the least significant bit will be considered. Thus:

c	b	a	
1	1	1	x1 factor
1	1	0	x10 factor

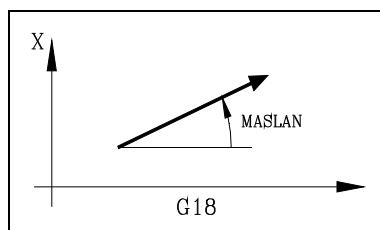
The HBE handwheel has priority. That is, regardless of the mode selected at the CNC switch (continuous or incremental JOG, handwheel), HBEVAR is set to other than "0", the CNC goes into handwheel mode.

It shows the selected axis in reverse video and the multiplying factor selected by the PLC. When the HBEVAR variable is set to "0", it shows the mode selected by the switch again.

Read-and-write variables

MASLAN

It must be used when the path-handwheel or the path-jog is selected.

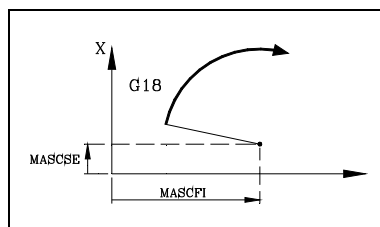


Indicates the angle of the linear path.

MASCFI

MASCSE

They must be used when the path-handwheel or the path-jog is selected.



On circular paths (arcs), they indicate the center coordinates.

11.2.9 Variables associated with feedback

ASIN(X-C)

"A" signal of the CNC's sinusoidal feedback for the X-C axis.

BSIN(X-C)

"B" signal of the CNC's sinusoidal feedback for the X-C axis.

ASINS

"A" signal of the CNC's sinusoidal feedback for the spindle.

BSINS

"B" signal of the CNC's sinusoidal feedback for the spindle.

11.

HIGH-LEVEL LANGUAGE PROGRAMMING Variables



CNC 8037

·T· MODEL
SOFT: V01.4x

11.2.10 Variables associated with the main spindle

In these variables associated with the spindle, their values are given in revolutions per minute and the main spindle override values are given in integers from 0 to 255.

Certain variables interrupt block preparation (it is indicated in each one) and the CNC waits for that command to be executed before resuming block preparation.

Read-only variables

SREAL

Returns the real main spindle turning speed in revolutions per minute. It interrupts block preparation.

(P100=SREAL)

It assigns to P100 the real turning speed of the main spindle.

FTEOS

It returns the theoretical turning speed of the main spindle.

SPEED

Returns, in revolutions per minute, the main spindle speed selected at the CNC.

This turning speed may be indicated by program, by PLC or by DNC; the CNC selects one of them, the one indicated by DNC has the highest priority and the one indicated by program has the lowest priority.

DNCS

Returns the turning speed in revolutions per minute, selected by DNC. If it has a value of 0 it means that it is not selected.

PLCS

Returns the turning speed in revolutions per minute selected by PLC. If it has a value of 0 it means that it is not selected.

PRGS

Returns the turning speed in revolutions per minute, selected by program.

CSS

It returns the constant surface speed selected at the CNC. Its value is given in the active units (ft/min or m/min).

This constant surface speed may be indicated by program, by PLC or by DNC; the CNC selects one of them, the one indicated by DNC has the highest priority and the one indicated by program has the lowest priority.

DNCCSS

It returns the constant surface speed selected via DNC. Its value is given in m/min or ft/min and it is 0 it means that it is not currently selected.

PLCCSS

It returns the constant surface speed selected by PLC. Its value is given in m/min or ft/min.

PRGCCSS

It returns the constant surface speed selected by program. Its value is given in m/min or ft/min.

SSO

It returns the turning speed override (%) of the main spindle currently selected at the CNC. It is given in integer values between 0 and "MAXFOVR" (maximum 255).

11.

This turning speed percentage of the main spindle may be indicated by program, by PLC, by DNC or by the front panel; the CNC selects one of them and the priority (from the highest to the lowest) is: by program, by DNC, by PLC and from the front panel.

DNCSSO

It returns the turning speed override % of the main spindle currently selected via DNC. If it has a value of 0 it means that it is not selected.

PLCSSO

It returns the turning speed override % of the main spindle currently selected by PLC. If it has a value of 0 it means that it is not selected.

CNCSSO

It returns the turning speed override % of the main spindle currently selected from the front panel.

SLIMIT

It returns the value set in rpm at the CNC for the turning speed limit of the main spindle.

This limit may be indicated by program, by PLC or by DNC; the CNC selects one of them, the one indicated by DNC has the highest priority and the one indicated by program has the lowest priority.

DNCSL

It returns the speed limit of the main spindle in rpm currently selected via DNC. If it has a value of 0 it means that it is not selected.

PLCSL

It returns the speed limit of the main spindle in rpm currently selected by PLC. If it has a value of 0 it means that it is not selected.

PRGSL

It returns the speed limit of the main spindle in rpm currently selected by program.

MDISL

Maximum machining spindle speed. This variable is also updated (refreshed) when programming function G92 via MDI.

POSS

It returns the real position of the main spindle. Its value may be within $\pm 99999.9999^\circ$. It interrupts block preparation.

RPOSS

It returns the real position of the main spindle. Its value is given in 0.0001 degree units (between -360° and 360°). It interrupts block preparation.

TPOSS

It returns the theoretical position of the main spindle (real position + lag). Its value may be within $\pm 99999.9999^\circ$. It interrupts block preparation.

RTPOSS

It returns the theoretical position of the main spindle (real position + lag) in 360° module. Its value may be between 0 and 360° . It interrupts block preparation.

PRGSP

Position programmed in M19 via program for the main spindle. This variable may be read from the CNC, from the PLC and from DNC.

FLWES

It returns the main spindle's following error in degrees (within ± 99999.9999). It interrupts block preparation.

11.

HIGH-LEVEL LANGUAGE PROGRAMMING
Variables

FAGOR 

CNC 8037

· T· MODEL
SOFT: V01.4x

Read-and-write variables

PRGSSO

This variable may be used to read or modify the speed override percentage of the main spindle currently selected by program. It is given in integer values between 0 and "MAXFOVR" (maximum 255). If it has a value of 0 it means that it is not selected.

(P110=PRGSSO)

It assigns to P110 the % of the main spindle speed selected by program.

(PRGSSO=P111)

It sets the value indicating the main spindle speed % selected by program to the value of arithmetic parameter P111.

11.

HIGH-LEVEL LANGUAGE PROGRAMMING
Variables



CNC 8037

·T· MODEL
SOFT: V01.4x

11.2.11 PLC related variables

It should be borne in mind that the PLC has the following resources:

(I1 thru I512)	Inputs.
(O1 thru O512)	Outputs.
M1 thru M5957)	Marks.
(R1 thru R499)	32-bit registers.
(T1 thru T512)	Timers with a timing count in 32 bits.
(C1 thru C256)	Counters with a count in 32 bits.

If any variable is accessed which allows the status of a PLC variable to be read or modified (I,O,M,R,T,C), block preparation is interrupted and the CNC waits for this command to be executed in order to restart block preparation.

Read-only variables

PLCMSG

Returns the number of the active PLC message with the highest priority and will coincide with the number displayed on screen (1..128). If there is none, it returns 0.

(P110=PLCMSG)

It assigns to P100 the number of the active PLC message with the highest priority.

Read-and-write variables

PLCIn

This variable allows 32 PLC inputs to be read or modified starting with the one indicated (n).

The value of the inputs which are used by the electrical cabinet cannot be modified as their values are determined by it. Nevertheless, the status of the remaining inputs can be modified.

PLCOn

This variable allows 32 PLC outputs to be read or modified starting from the one indicated (n).

(P110=PLCO 22)

It assigns to parameter P110 the value of outputs O22 through O53 (32 outputs) of the PLC.

(PLCO 22=\$F)

It sets outputs O22 through O25 to "1" and outputs O26 through O53 to "0".

Bit	31	30	29	28	27	26	25	24	23	22	...	5	4	3	2	1	0
	0	0	0	0	0	0	0	0	0	0	...	0	0	1	1	1	1
Output	53	52	51	50	49	48	47	46	45	44	...	27	26	25	24	23	22

PLCMn

This variable allows 32 PLC marks to be read or modified starting from the one indicated (n).

PLCRn

This variable allows the status of 32 register bits to be read or modified starting from the one indicated (n).

PLCTn

This variable allows the timer count to be read or modified starting from the one indicated (n).

11.

PLCCn

This variable allows the counter count to be read or modified starting from the one indicated (n).

PLCMMn

This variable permits reading or modifying the PLC mark (n).

(PLMM4=1)

It sets mark M4 to ·1· and leaves the rest untouched.

(PLCM4=1)

It sets mark M4 to ·1· and the following 31 marks (M5, through M35) to ·0·

11.

HIGH-LEVEL LANGUAGE PROGRAMMING
Variables



CNC 8037

·T· MODEL
SOFT: V01.4x

11.2.12 Variables associated with local parameters

The CNC allows 26 local parameters (P0-P25) to be assigned to a subroutine, by using mnemonics PCALL and MCALL. In addition to performing the required subroutine these mnemonics allow local parameters to be initialized.

Read-only variables

CALLP

Allows us to know which local parameters have been defined and which have not, in the call to the subroutine by means of the PCALL or MCALL mnemonic.

The information will be given in the 26 least significant bits (bits 0..25), each of these corresponding to the local parameter of the same number; thus bit 12 corresponds to P12.

Each bit will indicate whether the corresponding local parameter has been defined (=1) or not (0).

Bit	31	30	29	28	27	26	25	24	23	22	...	5	4	3	2	1	0
	0	0	0	0	0	0	*	*	*	*	...	*	*	*	*	*	*

Example:

```
;Call to subroutine 20.
(PCALL 20, P0=20, P2=3, P3=5)
...
...
;Beginning of subroutine 20.
( SUB 20)
(P100 = CALLP)
...
...
```

In parameter P100 the following will be obtained:

0000	0000	0000	0000	0000	0000	0000	0000	1101	LSB
------	------	------	------	------	------	------	------	------	-----

11.

11.2.13 Operating-mode related variables

Read-only variables related to the standard mode

OPMODE

It returns the code corresponding to the selected operating mode.

0 = Main menu.

10 = Automatic execution.

11 = Single block execution.

12 = MDI in EXECUTION.

13 = Tool inspection.

14 = Repositioning.

15 = Block search executing G.

16 = Block search executing G, M, S, T.

20 = Theoretical path simulation.

21 = G function simulation.

22 = G, M, S and T function simulation.

23 = Simulation with movement in the main plane.

24 = Simulation with rapid movement.

25 = Rapid simulation with S=0.

30 = Normal editing.

31 = User editing.

32 = TEACH-IN editing.

33 = Interactive editor.

40 = Movement in continuous JOG.

41 = Movement in incremental JOG.

42 = Movement with electronic handwheel.

43 = HOME search in JOG.

44 = Position preset in JOG.

45 = Tool calibration.

46 = MDI in JOG.

47 = User JOG operation.

50 = Zero offset table.

51 = Tool offset table.

52 = Tool table.

53 = Tool magazine table.

54 = Global parameter table.

55 = Local parameter table.

56 = User parameter table.

57 = OEM parameter table.

60 = Utilities.

70 = DNC status.

11.

HIGH-LEVEL LANGUAGE PROGRAMMING
Variables

CNC 8037

·T· MODEL
SOFT: V01.4x

71 = CNC status.

80 = PLC file editing.

81 = PLC program compilation.

82 = PLC monitoring.

83 = Active PLC messages.

84 = Active PLC pages.

85 = Save PLC program.

86 = Restore PLC program.

87 = PLC usage maps.

88 = PLC statistics.

90 = Customizing.

100 = General machine parameter table.

101 = Axis machine parameter tables.

102 = Spindle machine parameter table.

103 = Serial line related machine parameter tables.

104 = PLC machine parameter table.

105 = M function table.

106 = Leadscrew error compensation tables.

107 = Machine parameter table for Ethernet.

110 = Diagnosis: configuration.

111 = Diagnosis: hardware test.

112 = Diagnosis: RAM memory test.

113 = Diagnosis: Flash memory test.

114 = User diagnosis.

115 = Hard disk diagnosis (HD).

116 = Circle geometry test.

117 = Oscilloscope.

Read-only variables related to the conversational TC mode.

In these work modes, it is recommended to use variables OPMODA, OPMODB and OPMODC. The OPMODE variable is generic and contains different values to those of the standard mode.

OPMODE

It returns the code corresponding to the selected operating mode.

0 = CNC starting up.

10 = In execution mode.

In execution or waiting for the [CYCLE START] key (drawing of the [CYCLE START] on top).

21 = In graphic simulation mode.

30 = Cycle editing.

40 = In Jog mode (standard screen).

45 = In tool calibration mode.

60 = Managing parts. PPROG mode.

OPMODA

Indicates the operating mode currently selected when working with the main channel.

11.

HIGH-LEVEL LANGUAGE PROGRAMMING
Variables

FAGOR 

CNC 8037

·T· MODEL
SOFT: V01.4x

Use the OPMODE variable to know at any time the selected operating mode (main channel, user channel, PLC channel).

This information is given at the least significant bits with a "1" when active and with a "0" when not active or when it is not available in the current version.

Bit 0	Program in execution.
Bit 1	Program in simulation.
Bit 2	Block in execution via MDI, JOG.
Bit 3	Repositioning in progress.
Bit 4	Program interrupted, by CYCLE STOP.
Bit 5	MDI, JOG Block interrupted.
Bit 6	Repositioning interrupted.
Bit 7	In tool inspection.
Bit 8	Block in execution via CNCEX1.
Bit 9	Block via CNCEX1 interrupted.
Bit 10	CNC ready to accept JOG movements: jog, handwheel, teach-in, inspection.
Bit 11	CNC ready to receive the CYCLE START command: execution, simulation and MDI modes.
Bit 12	The CNC is not ready to execute anything involving axis or spindle movement.
Bit 13	It identifies the block search.

OPMODB

Indicates the type of simulation currently selected. This information is given at the least significant bits indicating with a "1" the one currently selected.

Bit 0	Theoretical path.
Bit 1	G functions.
Bit 2	G M S T functions.
Bit 3	
Bit 4	Rapid.
Bit 5	Rapid (S=0).

OPMODC

Indicates the axes selected by handwheel. This information is given at the least significant bits indicating with a "1" the one currently selected.

Bit 0	Axis 1.
Bit 1	Axis 2.
Bit 2	Axis 3.
Bit 3	Axis 4.
Bit 4	Axis 5.
Bit 5	Axis 6.
Bit 6	Axis 7.
Bit 7	
Bit 8	

The axis name corresponds to the number according to the programming order for them.

Example: If the CNC controls axes X, Y, Z, U, B, C, axis 1=X, axis2=Y, axis3=Z, axis4=U, axis5=B, axis6=C.

11.

HIGH-LEVEL LANGUAGE PROGRAMMING Variables



CNC 8037

.T- MODEL
SOFT: V01.4x

11.2.14 Other variables

Read-only variables**NBTOOL**

Indicates the tool number being managed. This variable can only be used within the tool change subroutine.

Example: There is a manual tool changer. Tool T1 is currently selected and the operator requests tool T5.

The subroutine associated with the tools may contain the following instructions:

```
(P103 = NBTOOL)
(MSG "SELECT T?P103 AND PRESS CYCLE START")
```

Instruction (P103 = NBTOOL) assigns the number of the tool currently being managed to parameter P103. Therefore, P103=5

The message displayed by the CNC will be ""SELECT T5 AND PRESS CYCLE START".

PRGN

Returns the program number being executed. Should none be selected, a value of -1 is returned.

BLKN

It returns the label number of the last executed block.

GSn

Returns the status of the G function indicated (n). 1 if it is active and 0 if not.

```
(P120=GS17)
```

It assigns the value 1 to parameter P120 if the G17 function is active and 0 if not.

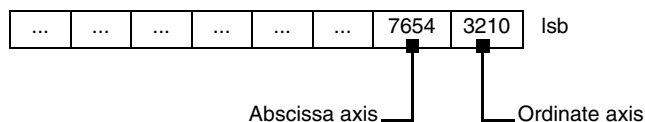
MSn

Returns the status of the M function indicated (n). 1 if it is active and 0 if not.

These functions are M00, M01, M02, M03, M04, M05, M06, M08, M09, M19, M30, M41, M42, M43 and M44.

PLANE

Returns data on the abscissa axis (bits 4 to 7) and the ordinate axis (bits 0 to 3) of the active plane in 32 bits and in binary.



The axes are coded in 4 bits and indicate the axis number according to the programming order.

Example: If the CNC controls the X and Z axes and the ZX plane (G18) is selected.

(P122 = PLANE) assigns the value of \$31 to parameter P122.

0000	0000	0000	0000	0000	0000	0011	0001	LSB
------	------	------	------	------	------	------	------	-----

Abscissa axis = 3 (0011) => Axis

Ordinate axis = 1 (0001) => X axis

11.

MIRROR

Returns in the least significant bits of the 32-bit group, the status of the mirror image of each axis, 1 in the case of being active and 0 if not.

Bit 8	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0	LSB
							Axis 2	Axis 1	

The axis name corresponds to the number according to the programming order for them.

Example: If the CNC controls the X and Z axes, axis1 = X, axis2 = Z.

SCALE

It returns the general scaling factor being applied.

SCALE(X-C)

Returns the specific scaling factor of the indicated axis (X-C).

PRBST

Returns probe status.

0 = the probe is not touching the part.

1 = the probe is touching the part.

If this variable is accessed, block preparation is interrupted and the CNC waits for this command to be executed to resume block preparation.

CLOCK

Returns the time in seconds indicated by the system clock. Possible values 0..4294967295.

If this variable is accessed, block preparation is interrupted and the CNC waits for this command to be executed to resume block preparation.

TIME

Returns the time in hours-minutes-seconds format.

(P150=TIME)

Loads P150 with hh-mm-ss. For example if the time is: 34sec. P150 = 182234.

If this variable is accessed, block preparation is interrupted and the CNC waits for this command to be executed to resume block preparation.

DATE

Returns the date in year-month-day format.

(P151=DATE)

It assigns to P151 the year-month-day. For example if the date is April 25th 1992, P151 = 920425.

If this variable is accessed, block preparation is interrupted and the CNC waits for this command to be executed to resume block preparation.

CYTIME

It returns the amount of time (in hundredths of a second) elapsed executing the part. It ignores the time the execution has been interrupted. Possible values 0..4294967295.

If this variable is accessed, block preparation is interrupted and the CNC waits for this command to be executed to resume block preparation.

11.

HIGH-LEVEL LANGUAGE PROGRAMMING
Variables



CNC 8037

·T· MODEL
SOFT: V01.4x

FIRST

Indicates whether it is the first time that a program has been run or not. It returns a value of 1 if it is the first time and 0 if not.

A first-time execution is considered as being one which is done:

- After turning on the CNC.
- After pressing [SHIFT]+[RESET].
- Every time a new program is selected.

ANAn

It returns the status of the indicated analog input (n). The value given in Volts and in ± 1.4 format. It is possible to select one of the 8 analog inputs (1..8) available. The values returned will be within the ± 5 V range.

If this variable is accessed, block preparation is interrupted and the CNC waits for this command to be executed to resume block preparation.

TIMEG

It shows the timing status of the timer programmed with G4 K in the CNC channel. This variable, returns the time remaining to end the timing block in hundredths of a second.

RIP

Linear theoretical feedrate resulting from the next loop (in mm/min).

The calculation of the resulting feedrate ignores the rotary axes, slave axes (gantry, coupled and synchronized) as well as DRO axes.

Read-and-write variables

TIMER

This variable allows reading or modifying the time, in seconds, indicated by the clock enabled by the PLC. Possible values 0..4294967295.

If this variable is accessed, block preparation is interrupted and the CNC waits for this command to be executed to resume block preparation.

PARTC

The CNC has a part counter whose count increases, in all modes except simulation, every time M30 or M02 is executed and this variable allows its value to be read or modified. This value will be between 0 and 4294967295

If this variable is accessed, block preparation is interrupted and the CNC waits for this command to be executed to resume block preparation.

KEY

Returns the code of the last key accepted.

This variable may be used as a write variable only inside a customizing program (user channel).

If this variable is accessed, block preparation is interrupted and the CNC waits for this command to be executed to resume block preparation.

KEYSRC

This variable allows reading or modifying the source of keystrokes, possible values being:

- 0 = Keyboard.
- 1 = PLC.
- 2 = DNC.

The CNC only allows modification of this variable if this is at 0.

11.

ANAO_n

This variable allows the required analog output (n) to be read or modified. The value assigned will be expressed in volts and in the ± 2.4 format (± 10 V).

The analog outputs which are free among the eight (1 through 8) available at the CNC may be modified, the corresponding error being displayed if an attempt is made to write in one which is occupied.

If this variable is accessed, block preparation is interrupted and the CNC waits for this command to be executed to resume block preparation.

SELPRO

When having two probe inputs, it allows selecting the active input.

On power-up, it assumes the value of $\cdot 1 \cdot$ thus selecting the first probe input. To select the second probe input, set it to a value of $\cdot 2 \cdot$.

Accessing this variable from the CNC interrupts block preparation.

DIAM

It changes the programming mode for X axis coordinates between radius and diameter. When changing the value of this variable, the CNC assumes the new way to program the following blocks.

When the variable is set to $\cdot 1 \cdot$, the programmed coordinates are assumed in diameter; when is set to $\cdot 0 \cdot$, the programmed coordinates are assumed in radius.

This variable affects the display of the real value of the X axis in the coordinate system of the part and the reading of variables PPOSX, TPOSX and POSX.

On power-up, after executing an M02 or M30 and after an emergency or a reset, the variable is initialized according to the value of the DFORMAT parameter of the X axis. If this parameter has a value equal to or greater than 4, the variable takes a value of 1; otherwise, it takes the value of $\cdot 0 \cdot$.

PRBMOD

It indicates whether a probing error is to issued or not in the following cases, even if general machine parameter PROBERR (P119) =YES.

- When a G75 probing move finishes before the probe has touched part.
- When a G76 probing move finishes but the probe is still touching the part.

The PRBMOD variable takes the following values.

Value	Meaning
0	An error message is issued.
1	No error message is issued.

Default value 0.

The PRBMOD variable can be read and written from the CNC and the PLC and read from the DNC.

11.

11.3 Constants

Constants are defined as being all those fixed values which cannot be altered by a program. The following are considered as constants:

- Numbers expressed in the decimal system.
- Hexadecimal numbers.
- PI constant.
- Read-only tables and variables as their value cannot be altered with a program.

11.**HIGH-LEVEL LANGUAGE PROGRAMMING**

Constants

**CNC 8037**·T· MODEL
SOFT: V01.4x

11.4 Operators

An operator is a symbol that indicates the mathematical or logic operations to carry out. The CNC has arithmetic, relational, logic, binary, trigonometric operators and special operators.

Arithmetic operators.

+	add.	P1=3 + 4	P1=7
-	subtraction, also a negative value.	P2=5 - 2 P3= -(2 * 3)	P2=3 P3=-6
*	multiplication.	P4=2 * 3	P4=6
/	division.	P5=9 / 2	P5=4.5
MOD	Module or remainder of the division.	P6=7 MOD 4	P6=3
EXP	exponential.	P7=2 EXP 3	P7=8

Relational operators.

EQ	equal.
NE	different.
GT	greater than.
GE	greater than or equal to.
LT	Less than.
LE	Less than or equal to.

Logic and binary operators.

NOT, OR, AND, XOR: The act as logic operators between conditions and as binary operators between variables and constants.

```
IF (FIRST AND GS1 EQ 1) GOTO N100
P5 = (P1 AND (NOT P2 OR P3))
```

Trigonometric functions.

SIN	sine.	P1=SIN 30	P1=0.5
COS	cosine.	P2=COS 30	P2=0.8660
TAN	tangent.	P3=TAN 30	P3=0.5773
ASIN	arc sine.	P4=ASIN 1	P4=90
ACOS	arc cosine.	P5=ACOS 1	P5=0
ATAN	arc tangent.	P6=ATAN 1	P6=45
ARG	ARG(x,y) arctangent y/x.	P7=ARG(-1,-2)	P7=243.4349

There are two functions for calculating the arc tangent ATAN which returns the result between $\pm 90^\circ$ and ARG given between 0 and 360° .

11.

Other functions.

ABS	absolute value.	P1=ABS -8	P1=8
LOG	decimal logarithm.	P2=LOG 100	P2=2
SQRT	square root.	P3=SQRT 16	P3=4
ROUND	rounding up an integer number.	P4=ROUND 5.83	P4=6
FIX	Integer.	P5=FIX 5.423	P5=5
FUP	if integer takes integer. if not, takes entire part + 1.	P6=FUP 7 P6=FUP 5,423	P6=7 P6=6
BCD	converts given number to BCD.	P7=BCD 234	P7=564

0010	0011	0100
------	------	------

BIN	converts given number to binary.	P8=BIN \$AB	P8=171
-----	----------------------------------	-------------	--------

1010	1011
------	------

Conversions to binary and BCD are made in 32 bits, it being possible to represent the number 156 in the following formats:

Decimal	156
Hexadecimal	9C
Binary	0000 0000 0000 0000 0000 0000 1001 1100
BCD	0000 0000 0000 0000 0000 0001 0101 0110

11.

11.5 Expressions

An expression is any valid combination of operators, constants, parameters and variables.

All expressions must be placed between brackets, but if the expression is reduced to an integer, the brackets can be removed.

11.

11.5.1 Arithmetic expressions

These are formed by combining functions and arithmetic, binary and trigonometric operators with the constants and variables of the language.

The priorities of the operators and the way they can be associated determine how these expressions are calculated:

Priority from highest to lowest	To be associated
NOT, functions, - (negative)	from right to left.
EXP, MOD	from left to right.
*, /	from left to right.
+, - (add, subtract)	from left to right.
Relational operators	from left to right.
AND, XOR	from left to right.
OR	from left to right.

Brackets should be used in order to clarify the order in which the expression is to be evaluated.

```
(P3 = P4/P5 - P6 * P7 - P8/P9 )
(P3 = (P4/P5)-(P6 * P7)-(P8/P9))
```

Using redundant or additional brackets will neither cause errors nor slow down the execution.

In functions, brackets must be used except when these are applied to a numerical constant, in which case they are optional.

```
(SIN 45) (SIN (45))    they're both valid and equivalent.
(SIN 10+5)             the same as ((SIN 10)+5).
```

Expressions can be used also to reference parameters and tables:

```
(P100 = P9)
(P100 = P(P7))
(P100 = P(P8 + SIN(P8 * 20)))
(P100 = ORGX 55)
(P100 = ORGX (12+P9))
(PLCM5008 = PLCM5008 OR 1)
; Selects single block execution mode (M5008=1)
(PLCM5010 = PLCM5010 AND $FFFFFFFE)
; Frees feedrate override (M5010=0)
```

11.5.2 Relational expressions

These are arithmetic expressions joined by relational operators.

```
(IF (P8 EQ 12.8)
; It checks if the value of P8 is equal to 12.8.
(IF (ABS(SIN(P24)) GT SPEED)
;Analyzes if the sine is greater than the spindle speed.
(IF (CLOCK LT (P9 * 10.99))
; Analyzes if the clock count is less than (P9 * 10.99)
```

At the same time these conditions can be joined by means of logic operators.

```
(IF ((P8 EQ 12.8) OR (ABS(SIN(P24)) GT SPEED)) AND (CLOCK LT (P9 * 10.99)) ...
```

The result of these expressions is either true or false.

11.

HIGH-LEVEL LANGUAGE PROGRAMMING Expressions

11.

HIGH-LEVEL LANGUAGE PROGRAMMING

Expressions



CNC 8037

·T· MODEL
SOFT: V01.4x

The control instructions available to high-level programming can be grouped as follows:

- Assignment instructions.
- Display instructions.
- Enable-disable instructions.
- Flow control instructions.
- Subroutine instructions.
- Interruption-subroutine instructions.
- Program instructions.
- Screen customizing instructions.

Only one instruction can be programmed in each block, and no other additional information may be programmed in this block.

12.1 Assignment instructions

This is the simplest type of instruction and can be defined as:

(target = arithmetic expression)

A local or global parameter or a read-write variable may be selected as target. The arithmetic expression may be as complex as required or a simple numerical constant.

(P102 = FZLOX)

(ORGX 55 = (ORGX 54 + P100))

In the specific case of designating a local parameter using its name (A instead of P0, for example) and the arithmetic expression being a numerical constant, the instruction can be abbreviated as follows:

(P0=13.7) ==> (A=13.7) ==> (A13.7)

Within a single block, up to 26 assignments can be made to different targets, a single assignment being interpreted as the set of assignments made to the same target.

(P1=P1+P2, P1=P1+P3, P1=P1*P4, P1=P1/P5)

It is the same as saying:

(P1=(P1+P2+P3)*P4/P5).

The different assignments which are made in the same block will be separated by commas ",".

12.

12.2 Display instructions

(ERROR integer, "error text")

This instruction stops the execution of the program and displays the indicated error, it being possible to select this error in the following ways:

(ERROR integer)

This will display the error number indicated and the text associated to this number according to the CNC error code (should there be one).

(ERROR integer, "error text")

This will display the number and the error text indicated, it being necessary to write the text between quote marks "".

(ERROR "error text").

This will display the error text only.

The error number may be defined by means of a numerical constant or an arithmetic parameter. When using a local parameter, its numeric format must be used (P0 thru P25 instead of A thru Z).

Programming examples:

(ERROR 5)

(ERROR P100)

(ERROR "User error")

(ERROR 3, "User error")

(ERROR P120, "User error")

(MSG "message")

This instruction will display the message indicated between quote marks.

The CNC screen is provided with an area for displaying DNC or user program messages, and always displays the last message received irrespective of where it has come from.

Example: (MSG "Check tool")

(DGWZ expression 1, expression 2, expression 3, expression 4)

The DGWZ instruction (Define Graphic Work Zone) defines the graphics area.

Each expression forming the instruction syntax correspond to one of the limits and they must be defined in millimeters or inches.

expression 1	Minimum Z
expression 2	Maximum Z
expression 3	Inside radius or inside diameter.
expression 4	Outside radius or outside diameter.

12.

12.3 Enable-disable instructions

(ESBLK and DSBLK)

After executing the mnemonic ESBLK, the CNC executes all the blocks that come after as if it were dealing with a single block.

This single block treatment is kept active until it is cancelled by executing the mnemonic DSBLK.

In this way, should the program be executed in the SINGLE BLOCK operating mode, the group of blocks which are found between the mnemonics ESBLK and DSBLK will be executed in a continuous cycle, i.e., execution will not be stopped at the end of a block but will continue by executing the following one.

```
G01 X30 Z10 F1000 T1 D1
(ESBLK)                                ; Start of single block
G01 X20 Z10
G01 X20 Z20
G02 X10 Z30 I-10 K0
(DSBLK)                                ; Cancellation of single block
G01 X10 Z40
M30
```

(ESTOP and DSTOP)

After executing the mnemonic DSTOP, the CNC enables the Stop key, as well as the Stop signal from the PLC.

It will remain disabled until it is enabled once again by means of the mnemonic ESTOP.

(EFHOLD and DFHOLD)

After executing the mnemonic DFHOLD, the CNC enables the Feed-Hold input from the PLC.

It will remain disabled until it is enabled once again by means of the mnemonic EFHOLD.

12.

12.4 Flow control instructions

The GOTO and RPT instructions cannot be used in programs that are executed from a PC connected through the serial lines.

(GOTO N(expression))

The mnemonic GOTO causes a jump within the same program, to the block defined by the label N(expression). The execution of the program will continue after the jump, from the indicated block.

The jump label can be addressed by means of a number or by any expression which results in a number.

```

G00 X30 Z10 T2 D4
X30 Z20
(GOTO N22)                ; Jump instruction
X20 Z20                   ; It is not executed
X20 Z10                   ; It is not executed
N22 G01 X10 Z10 F1000      ; Continues execution in this block
G02 X0 Z40 I-105 K0
...
```

(RPT N(expression), N(expression), P(expression))

The mnemonic RPT executes the part of the program between the blocks defined by means of the labels N(expression). The blocks to be executed may be in the execution program or in a RAM memory program.

The label P(expression) indicates the number of the program containing the blocks to be executed. If not defined, the CNC interprets that the portion to be repeated is located in the same program.

All the labels can be indicated by means of a number or by any expression which results in a number. The part of the program selected by means of the two labels must belong to the same program, by first defining the initial block and then the final block.

The execution of the program will continue in the block following the one in which the mnemonic RPT was programmed, once the selected part of the program has been executed.

```

N10 G00 X10
    Z20
    G01 X5
    G00 Z0
N20 X0
N30 (RPT N10, N20) N3
N40 G01 X20
    M30

When reaching block N30, the program will execute section N10-N20 three times.
Once this has been completed, the program will continue execution in block N40.
```



Since the RPT instruction does not interrupt block preparation or tool compensation, it may be used when using the EXEC instruction and while needing to maintain tool compensation active.

(IF condition <action1> ELSE <action2>)

This instruction analyzes the given condition that must be a relational expression. If the condition is true (result equal to 1), <action1> will be executed, otherwise (result equal to 0) <action2> will be executed.

Example:

```

(IF (P8 EQ 12.8) CALL 3 ELSE PCALL 5, A2, B5, D8)
If P8 = 12.8 executes the mnemonic (CALL3)
If P8 <> 12.8 executes the mnemonic (PCALL 5, A2, B5, D8)
```

12.

PROGRAM CONTROL INSTRUCTIONS
Flow control instructions

FAGOR

CNC 8037

·T· MODEL
SOFT: V01.4x

The instruction can lack the ELSE part, i.e., it will be enough to program IF condition <action1>.

Example:

(IF (P8 EQ 12.8) CALL 3)

Both <action1> and <action2> can be expressions or instructions, except for mnemonics IF and SUB.

Due to the fact that in a high level block local parameters can be named by means of letters, expressions of this type can be obtained:

(IF (E EQ 10) M10)

If the condition of parameter P5 (E) having a value of 10 is met, the miscellaneous function M10 will not be executed, since a high level block cannot have ISO code commands. In this case M10 represents the assignment of value 10 to parameter P12, i.e., one can program either:

(IF (E EQ 10) M10) or (IF (P5 EQ 10) P12=10)

12.

PROGRAM CONTROL INSTRUCTIONS

Flow control instructions



CNC 8037

·T· MODEL
SOFT: V01.4x

12.5 Subroutine instructions

A subroutine is a part of a program which, being properly identified, can be called from any position of a program to be executed.

A subroutine can be kept in the memory of the CNC as an independent part of a program and be called one or several times, from different positions of a program or different programs.

Only subroutines stored in the CNC's RAM memory can be executed. Therefore, to execute a subroutine stored in the hard disk (KeyCF) or in a PC connected through the serial line, copy it first into the CNC's RAM memory.

If the subroutine is too large to be copied into RAM, it must be converted into a program and then the EXEC instruction must be used.

(SUB integer)

The SUB instruction defines as subroutine the set of program blocks programmed next until reaching the RET subroutine. The subroutine is identified with an integer which also defines the type of subroutine; either general or OEM.

Range of general subroutines	SUB 0000 - SUB 9999
Range of OEM (manufacturer's) subroutines	SUB 10000 - SUB 20000

The OEM subroutines are treated like the general ones, but with the following restrictions:

- They can only be defined in OEM programs, having the [O] attribute. Otherwise, the CNC will display the corresponding error.

Error 63: Program subroutine number 1 thru 9999.

- To execute an OEM subroutine using CALL, PCALL or MCALL, it must be inside an OEM program. Otherwise, the CNC will display the corresponding error.

Error 1255: Subroutine restricted to an OEM program.

There can not be two subroutines with the same identification number in the CNC memory, even when they belong to different programs.

(RET)

The mnemonic RET indicates that the subroutine which was defined by the mnemonic SUB, finishes in this block.

```
( SUB 12)                                ; Definition of subroutine 12
G91 G01 XP0 F5000
ZP1
XP0
ZP1
(RET)                                    ; End of subroutine
```

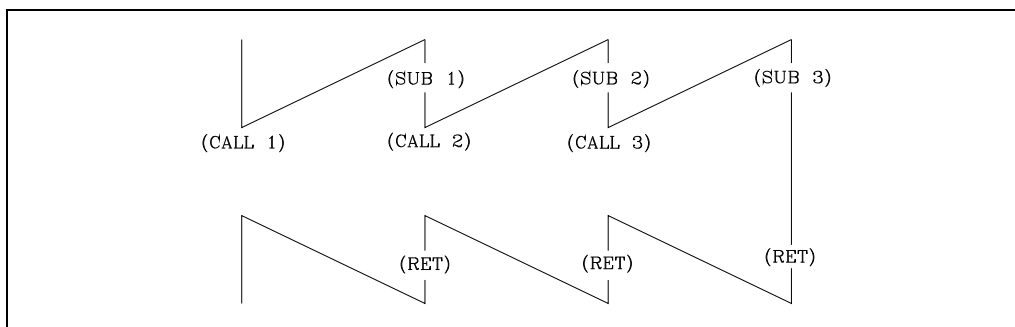
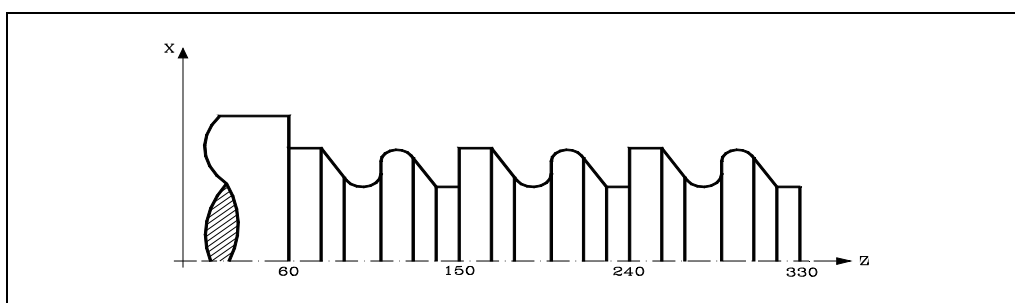
12.

PROGRAM CONTROL INSTRUCTIONS
Subroutine instructions

(CALL (expression))

The mnemonic CALL makes a call to the subroutine indicated by means of a number or by means of any expression that results in a number.

As a subroutine may be called from a main program, or a subroutine, from this subroutine to a second one, from the second to a third, etc..., the CNC limits these calls to a maximum of 15 nesting levels, it being possible to repeat each of the levels 9999 times.

**Programming example.**

```
G90 G01 X100 Z330
(CALL 10)
G90 G01 X100 Z240
(CALL 10)
G90 G01 X100 Z150
M30
```

```
( SUB 10)
G91 G01 Z-10
X40 Z-10
G03 X0 Z-20 I0 K-10
G01 X-20
G02 X0 Z-20 I0 K-10
G01 X40 Z-10
Z-20
(RET)
```

(PCALL (expression), (assignment instruction), (assignment instruction),...)

The mnemonic PCALL calls the subroutine indicated by means of a number or any expression that results in a number. In addition, it allows up to a maximum of 26 local parameters of this subroutine to be initialized.

These local parameters are initialized by means of assignment instructions.

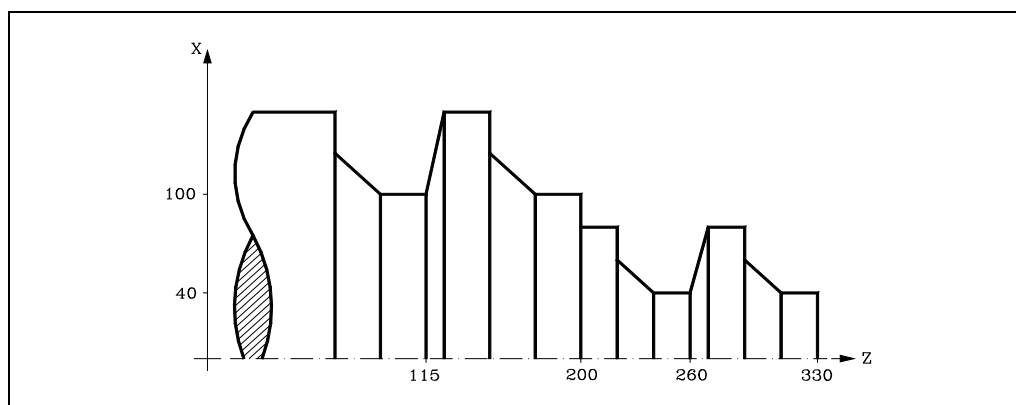
Example: (PCALL 52, A3, B5, C4, P10=20)

In this case, in addition to generating a new subroutine nesting level, a new local parameter nesting level will be generated, there being a maximum of 6 levels of local parameter nesting, within the 15 levels of subroutine nesting.

Both the main program and each subroutine that is found on a parameter nesting level, will have 26 local parameters (P0-P25).

Programming example.

The X axis is programmed in diameter.



```
G90 G01 X80 Z330
(PCALL 10, P0=20, P1=-10)           ; Also (PCALL 10, A20, B-10)
G90 G01 X80 Z260
(PCALL 10, P0=20, P1=-10)           ; Also (PCALL 10, A20, B-10)
G90 G01 X200 Z200
(PCALL 10, P0=30, P1=-15)           ; Also (PCALL 10, A30, B-15)
G90 G01 X200 Z115
(PCALL 10, P0=30, P1=-15)           ; Also (PCALL 10, A30, B-15)
M30

( SUB 10)
G91 G01 ZP1
XP0 ZP1
XP0
ZP1
(RET)
```

12.

PROGRAM CONTROL INSTRUCTIONS
Subroutine instructions

(MCALL (expression), (assignment instruction), (assignment instruction),...)

By means of the mnemonic MCALL, any user-defined subroutine (SUB integer) acquires the category of canned cycle.

The execution of this mnemonic is the same as the mnemonic PCALL, but the call is modal, i.e., if another block with axis movement is programmed at the end of this block, after this movement, the subroutine indicated will be executed and with the same call parameters.

If, when a modal subroutine is selected, a movement block with a number of repetitions is executed, for example X10 N3, the CNC will execute the movement only once (X10) and after the modal subroutine, as many times as the number of repetitions indicates.

Should block repetitions be chosen, the first execution of the modal subroutine will be made with updated call parameters, but not for the remaining times, which will be executed with the values which these parameters have at that time.

If, when a subroutine is selected as modal, a block containing the MCALL mnemonic is executed, the present subroutine will lose its modal quality and the new subroutine selected will be changed to modal.

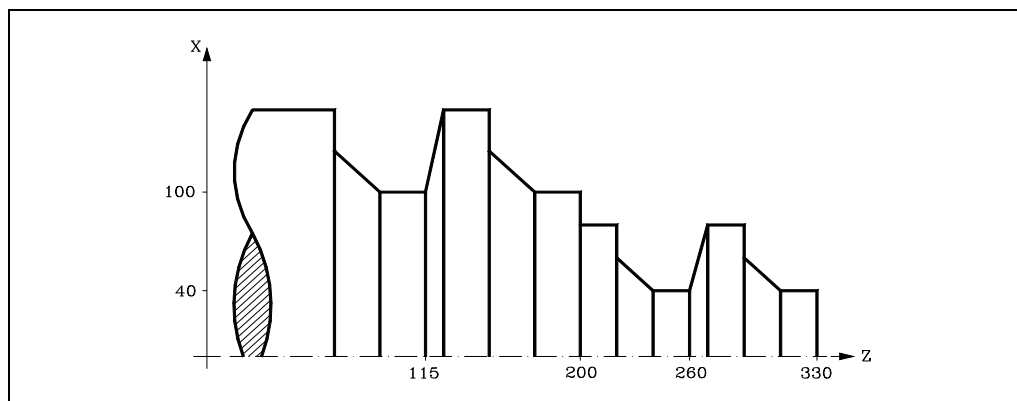
(MDOFF)

The MDOFF instruction indicates that the mode assumed by a subroutine with the MCALL instruction or a part-program with MEXEC ends in that block.

The use of modal subroutines simplifies programming.

Programming example.

The X axis is programmed in diameter.



```
(P100=20, P101=-10)
```

```
G90 G01 X80 Z330
```

```
(MCALL 10)
```

```
G90 G01 X80 Z260
```

```
(P100=30, P101=-15)
```

```
G90 G01 X200 Z200
```

```
G90 G01 X200 Z115
```

```
(MDOFF)
```

```
M30
```

```
( SUB 10)
```

```
G91 G01 ZP101
```

```
XP100 ZP101
```

```
XP100
```

```
ZP101
```

```
(RET)
```


12.6 Interruption-subroutine instructions

Whenever one of the general interruption logic input is activated, "INT1" (M5024), "INT2" (M5025), "INT3" (M5026) or "INT4" (M5027), the CNC temporarily interrupts the execution of the program in progress and starts executing the interruption subroutine whose number is indicated by the corresponding general parameter.

With INT1 (M5024) the one indicated by machine parameter INT1SUB (P35)

With INT2 (M5025) the one indicated by machine parameter INT2SUB (P36)

With INT3 (M5026) the one indicated by machine parameter INT3SUB (P37)

With INT4 (M5027) the one indicated by machine parameter INT4SUB (P38)

The interruption subroutines are defined like any other subroutine by using the instructions: "(SUB integer)" and "(RET)".

These interruption subroutines do not change the nesting level of local parameters, thus only global parameters must be used in them.

Within an interruption subroutine, it is possible to use the "(REPOS X, Y, Z, ...)" instruction described next.

Once the execution of the subroutine is over, the CNC resumes the execution of the program which was interrupted.

(REPOS X, Y, Z, ...)

The REPOS instruction must always be used inside an interruption subroutine and facilitates the repositioning of the machine axes to the point of interruption.

When executing this instruction, the CNC moves the axes to the point where the program was interrupted.

Inside the REPOS instruction, indicate the order the axes must move to the point where the program was interrupted.

- The axes move one by one.
- It is not necessary to define all the axes, only those to be repositioned.
- The axes that make up the main plane of the machine move together. Both axes need not be defined because the CNC moves the first one. The movement is not repeated when defining the second one, it is ignored.

If the REPOS instruction is detected while executing a subroutine not activated by an interruption input, the CNC will issue the corresponding error message.

12.

12.7 Program instructions

With this CNC, from a program in execution, it is possible to:

- Execute another program. Instruction (EXEC P.....)
- Execute another program in modal mode. Instruction (MEXEC P.....)
- Generate a new program. Instruction (OPEN P.....)
- Add blocks to an existing program. Instruction (WRITE P.....)

(EXEC P(expression), (directory))

The EXEC P instruction executes the part-program of the indicated directory

The part-program may be defined by a number or any expression resulting in a number.

By default, the CNC interprets that the part-program is in the CNC's RAM memory. If it is in another device, it must be indicated in (directory).

HD	in the Hard Disk.
DNC2	in a PC connected through the serial line.
DNCE	in a PC connected through Ethernet.

(MEXEC P(expression), (directory))

The MEXEC instruction executes the part-program of the indicated directory and it also becomes modal; i.e. if after this block, another one is programmed with axis movement; after this movement, it will execute the indicated program again.

The part-program may be defined with a number or with an expression whose result is a number.

By default, the CNC interprets that the part-program is in the CNC's RAM memory. If it is in another device, it must be so indicated in (directory):

HD	in the Hard Disk.
DNC2	in a PC connected through the serial line.
DNCE	in a PC connected through Ethernet.

If while the modal part-program is selected, a motion block is executed with a number of repetitions (for example X10 N3), the CNC ignores the number of repetitions and executes the movement and the modal part-program only once.

If while a part-program is selected as modal, a block containing the MEXEC instruction is executed from the main program, the current part-program stops being modal and the part-program called upon with MEXEC will then become modal.

If within the modal part-program, an attempt is made to execute a block using the MEXEC instruction, it will issue the relevant error message.

1064: The program cannot be executed.

(MDOFF)

The MDOFF instruction indicates that the mode assumed by a subroutine with the MCALL instruction or a part-program with MEXEC ends in that block.

(OPEN P(expression), (destination directory), A/D, "program comment")

The OPEN instruction begins editing a part-program. The part-program number may be indicated by a number or any expression resulting in a number.

By default, the new part-program edited will be stored in the CNC's RAM memory. To store it another device, it must be indicated in (destination directory).

HD	in the Hard Disk.
DNC2	in a PC connected through the serial line.
DNCE	in a PC connected through Ethernet.

12.

Parameter A/D is used when the program to be edited already exists.

- A The CNC appends the new blocks after the ones already existing.
- D The CNC deletes the existing program and starts editing a new one.

A program comment may also be associated with it; this comment will later be displayed next to it in the program directory.

The OPEN instruction allows generating a program from a program already in execution. That generated program will depend on the values assumed by the program being executed.

To edit blocks, the WRITE instruction must be used as described next.

Notes:

If the program to be edited already exists and the A/D parameters are not defined, the CNC will display an error message when executing the block.

The program opened with the OPEN instruction is closed when executing an M30, or another OPEN instruction and after an Emergency or Reset.

From a PC, only programs stored in the CNC'S RAM memory or in the hard disk (HD) can be opened.

(WRITE <block text>)

The mnemonic WRITE adds, after the last block of the program which began to be edited by means of the mnemonic OPEN P, the information contained in <block text> as a new program block.

When it is an ISO coded parametric block, all the parameters (global and local) are replaced by the numeric value they have at the time.

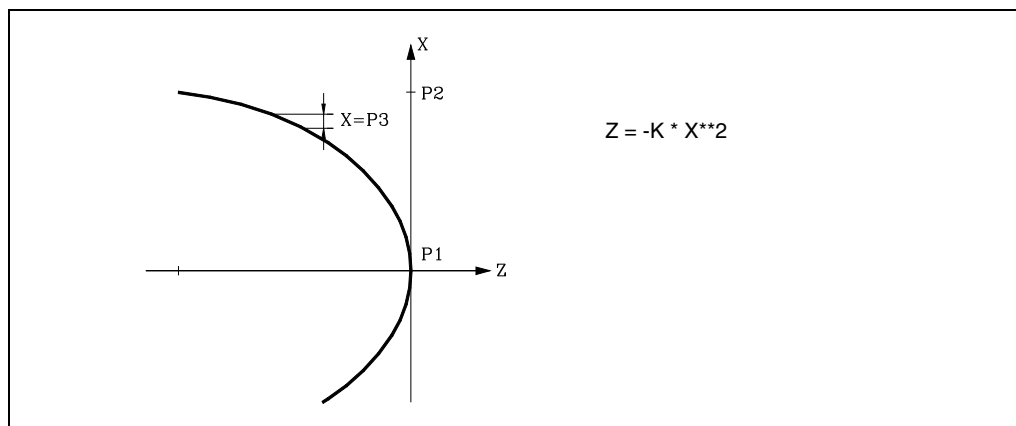
(WRITE G1 XP100 ZP101 F100) => G1 X10 Z20 F100

When it is a parametric block edited in high level, use the "?" character to indicate that the parameter is supposed to be replaced by the numeric value it has at the time.

(WRITE (SUB P102))	=>	(SUB P102)
(WRITE (SUB ?P102))	=>	(SUB 55)
(WRITE (ORGX54=P103))	=>	(ORGX54=P103)
(WRITE (ORGX54=?P103))	=>	(ORGX54=222)
(WRITE (PCALL P104))	=>	(PCALL P104)
(WRITE (PCALL ?P104))	=>	(PCALL 25)

If the mnemonic WRITE is programmed without having programmed the mnemonic OPEN previously, the CNC will display the corresponding error, except when editing a user customized program, in which case a new block is added to the program being edited.

Example of how to create a program that contains several points of a parabolic path.



The X axis is programmed in diameter, it uses subroutine number 2 and its parameters have the following meaning:

Call parameters:

A or P0 K constant value.
 B or P1 Initial X coordinate.
 C or P2 Final X coordinate.
 D or P3 X increment or step.

Calculated parameters:

E or P4 X coordinate.
 F or P5 Z coordinate.

A way to use this example could be:

```
G00 X0 Z0
(PCALL 2, A0.01, B0, C100, D1)
M30
```

Program generating subroutine.

```
( SUB 2)
(OPEN P12345)           ; Starts editing of program P12345
(P4=P1)
N100 (IF (P4+P3 GE P2) P4=P2 ELSE P4=P4+P3)
(P5=-(P0 * P4 * P4))
(WRITE G01 XP4 ZP5)      ; Movement block
(IF (P4 NE P2) GOTO N100)
(WRITE M30)              ; End of program block
(RET)                    ; End of subroutine
```

12.

PROGRAM CONTROL INSTRUCTIONS

Program instructions



CNC 8037

.T- MODEL
 SOFT: V01.4x

12.8 Screen customizing instructions

Customizing instructions may be used only when customizing programs made by the user.

These customizing programs must be stored in the CNC's RAM memory and they may utilize the "Programming Instructions" and they will be executed in the special channel designed for this use; the program selected in each case will be indicated in the following general machine parameters.

In "USERDPLY" the program to be executed in the Execution Mode will be indicated.

In "USEREDIT" the program to be executed in the Editing Mode will be indicated.

In "USERMAN" the program to be executed in the Manual (JOG) Mode will be indicated.

In "USERDIAG" the program to be executed in the Diagnosis Mode will be indicated.

The customizing programs may have up to five nesting levels besides their current one. Also, the customizing instructions do not allow local parameters, nevertheless all global parameters may be used to define them.

(**PAGE** (expression))

The mnemonic PAGE displays the page number indicated by means of a number or by means of any expression resulting in a number.

User-defined pages will be from page 0 to page 255 and will be defined from the CNC keyboard in the Graphic Editor mode and as indicated in the Operating Manual.

System pages will be defined by a number greater than 1000. See the corresponding appendix.

(**SYMBOL** (expression 1), (expression 2), (expression 3))

The mnemonic SYMBOL displays the symbol whose number is indicated by means of the value of expression 1 once this has been evaluated.

Its position on screen is also defined by expression 2 (column) and by expression 3 (row).

Expression 1, expression 2 and expression 3 may contain a number or any expression resulting in a number.

The CNC allows displaying any user-defined symbol (0-255) defined at the CNC keyboard in the Graphic Editor mode such as is indicated in the Operating Manual.

In order to position it within the display area its pixels must be defined, 0-639 for columns (expression 2) and 0-335 for rows (expression 3).

(**IB** (expression) = **INPUT** "text", format)

The CNC has 26 data entry variables (IB0-IB25)

The IB mnemonic displays the text indicated in the data input window and stores the data input by the user in the entry variable indicated by means of a number or by means of any expression resulting in a number.

The wait for data entry will only occur when programming the format of the requested data. This format may have a sign, integer part and decimal part.

If it bears the "-" sign, it will allow positive and negative values, and if it does not have a sign, it will only allow positive values.

The integer part indicates the maximum number of digits (0-6) desired to the left of the decimal point.

The decimal part indicates the maximum number of digits (0-5) desired to the right of the decimal point.

If the numerical format is not programmed; for example: (IB1 =INPUT "text"), the mnemonic will only display the indicated text without waiting for the data to be entered.

12.

(ODW (expression 1), (expression 2), (expression 3))

The mnemonic ODW defines and draws a white window on the screen with fixed dimensions (1 row and 14 columns).

Each mnemonic has an associated number which is indicated by the value of expression 1 once this has been evaluated.

Likewise, its position on screen is defined by expression 2 (row) and by expression 3 (column).

Expression 1, expression 2 and expression 3 may contain a number or any expression resulting in a number.

The CNC allows 26 windows (0-25) to be defined and their positioning within the display area, providing 21 rows (0-20) and 80 columns (0-79).

(DW(expression 1) = (expression 2), DW (expression 3) = (expression 4), ...)

The instruction DW displays in the window indicated by the value for expression 1, expression 3, .. once they have been evaluated, the numerical data indicated by expression 2, expression 4, ...

Expression 1, expression 2, expression 3, may contain a number or any expression which may result in a number.

The following example shows a dynamic variable display:

```
(ODW 1, 6, 33)
    ; Defines data window 1
(ODW 2, 14, 33)
    ; Defines data window 2
N10 (DW1=DATE, DW2=TIME)
    ; Displays the date in window 1 and the time in 2
(GOTO N10)
```

The CNC allows displaying the data in decimal, hexadecimal and binary format; the following instructions are available:

(DW1 = 100)

Decimal format. Value "100" displayed in window 1.

(DWH2 = 100)

Hexadecimal format. Value "64" displayed in window 2.

(DWB3 = 100)

Binary format. Value "01100100" displayed in window 3.

When using the binary format, the display is limited to 8 digits in such a way that a value of "11111111" will be displayed for values greater than 255 and the value of "10000000" for values more negative than -127.

Besides, the CNC allows the number stored in one of the 26 data input variables (IB0-IB25) to be displayed in the requested window.

The following example shows a request and later displays the axis feedrate:

```
(ODW 3, 4, 60)
    ; Defines data window 3.
(IB1=INPUT "Axis feed: ", 5.4)
    ; Axis feedrate request.
(DW3=IB1)
    ; Displays feedrate in window 3.
```

12.**PROGRAM CONTROL INSTRUCTIONS**
Screen customizing instructions**CNC 8037**

·T· MODEL
SOFT: V01.4x

(SK (expression 1) = "text1" (expression 2) = "text 2",)

The instruction SK defines and displays the new softkey menu indicated.

Each of the expressions will indicate the softkey number which it is required to modify (1-7, starting from the left) and the texts which it is required to write in them.

Expression 1, expression 2, expression 3, may contain a number or any expression which may result in a number.

Each text will allow a maximum of 20 characters that will be shown in two lines of 10 characters each. If the text selected has less than 10 characters, the CNC will center it on the top line, but if it has more than 10 characters the programmer will center it.

Examples:

(SK 1="HELP", SK 2="MAXIMUN POINT")

HELP	MAXIMUN POINT
------	---------------

(SK 1="FEED", SK 2=" _ _MAXIMUN _ _POINT")

FEED	MAXIMUN POINT
------	---------------



If while a standard CNC softkey menu is active, one or more softkeys are selected via high level language instruction: "SK", the CNC will clear all existing softkeys and it will only show the selected ones.

If while a user softkey menu is active, one or more softkeys are selected via high level language instruction "SK", the CNC will only replace the selected softkeys leaving the others intact.

(WKEY)

The mnemonic WKEY stops execution of the program until the key is pressed.

The pressed key will be recorded in the KEY variable.

```
...
(WKEY)                ; Wait for key
(IF KEY EQ $FC00 GOTO N1000) ; If key F1 has been pressed, continue in N1000
...
```

(WBUF "text", (expression))

The WBUF instruction can only be used when editing a program in the user channel.

This instruction may be programmed in two ways:

- (WBUF "text", (expression))

It adds the text and value of the expression, once it has been evaluated, to the block that is being edited and within the data entry window.

(Expression) may contain a number or any expression resulting in a number.

It will be optional to program the expression, but it will be required to define the text. If no text is required, "" must be programmed.

Examples for P100=10:

(WBUF "X", P100)	=>	X10
(WBUF "X P100")	=>	X P100

- (WBUF)

Enters into memory, adding to the program being edited and after the cursor position, the block being edited by means of (WBUF "text", (expression)). It also clears the editing buffer in order to edit a new block.

This allows the user to edit a complete program without having to quit the user editing mode after each block and press ENTER to "enter" it into memory.

12.

PROGRAM CONTROL INSTRUCTIONS
Screen customizing instructions

```

(WBUF "(PCALL 25, ")
; Adds "(PCALL 25, " to the block being edited.
(IB1=INPUT "Parameter A: ",-5.4)
; Request of Parameter A.
(WBUF "A=", IB1)
; Adds "A=(value entered)" to the block being edited.
(IB2=INPUT "Parameter B: ", -5.4)
; Request of Parameter B.
(WBUF ", B=", IB2)
; Adds "B=(value entered)" to the block being edited.
(WBUF ")")
; Adds ")" to the block being edited.
(WBUF )
; Enters the edited block into memory.
...

```

After executing this program the block being edited contains:

```
(PCALL 25, A=23.5, B=-2.25)
```

(SYSTEM)

The mnemonic SYSTEM stops execution of the user customized program and returns to the corresponding standard menu of the CNC.

Customizing program example:

The following customizing program must be selected as user program associated to the Editing Mode.

After selecting the Editing Mode and pressing the USER softkey, this program starts executing and it allows assisted editing of 2 user cycles. This editing process is carried out a cycle at a time and as often as desired.

Displays the initial editing page (screen)

```
N0 ( PAGE 10 )
```

Sets the softkeys to access the various modes and requests a choice

```

N5 (SK 1="CYCLE 1",SK 2="CYCLE 2",SK 7="EXIT")
(WKEY ) ; Request a key
(IF KEY EQ $FC00 GOTO N10) ; Cycle 1
(IF KEY EQ $FC01 GOTO N20) ; Cycle 2
(IF KEY EQ $FC06 SYSTEM ELSE GOTO N5) ; Quit or request a key

```



CNC 8037

·T· MODEL
SOFT: V01.4x

CYCLE 1

```

; Displays page 11 and defines 2 data entry windows
N10 (PAGE 11)
    (ODW 1,10,60)
    (ODW 2,15,60)

;Editing
(WBUF "( PCALL 1,") ; Adds "(PCALL 1," to the block being edited.

(IB 1=INPUT "X:",-6.5) ; Requests the value of X.
(DW 1=IB1) ; Data window 1 shows the entered value.
(WBUF "X",IB1) ; Adds X (entered value) to the block being edited.

(WBUF ",") ; Adds "," to the block being edited.

(IB 2=INPUT "Y:",-6.5) ; Requests the value of Y.
(DW 2=IB2) ; Data window 2 shows the entered value.
(WBUF "Y",IB2) ; Adds Y (entered value) to the block being edited.

(WBUF ")") ; Adds ")" to the block being edited.
(WBUF ) ; Enters the edited block into memory.
; For example : (PCALL 1, X2, Y3)

(GOTO N0)

```

CYCLE 2

```

; Displays page 12 and defines 3 data entry windows
N20 (PAGE 12)
    (ODW 1,10,60)
    (ODW 2,13,60)
    (ODW 3,16,60)

; Editing
(WBUF "( PCALL 2,") ; Adds "(PCALL 2," to the block being edited.

(IB 1=INPUT "A:",-6.5) ; Requests the value of A.
(DW 1=IB1) ; Data window 1 shows the entered value.
(WBUF "A",IB1) ; Adds A (entered value) to the block being edited.

(WBUF ",") ; Adds "," to the block being edited.

(IB 2=INPUT "B:",-6.5) ; Requests the value of B.
(DW 2=IB2) ; Data window 2 shows the entered value.
(WBUF "B",IB2) ; Adds B (entered value) to the block being edited.

(WBUF ",") ; Adds "," to the block being edited.
(IB 3=INPUT "C:",-6.5) ; Requests the value of C.
(DW 3=IB3) ; Data window 3 shows the entered value.
(WBUF "C",IB3) ; Adds C (entered value) to the block being edited.

(WBUF ")") ; Adds ")" to the block being edited.

(WBUF ) ; Enters the edited block into memory.
; For example: (PCALL 2, A3, B1, C3).

(GOTO N0)

```

12.

PROGRAM CONTROL INSTRUCTIONS
Screen customizing instructions



CNC 8037

·T· MODEL
SOFT: V01.4x

12.

PROGRAM CONTROL INSTRUCTIONS

Screen customizing instructions



CNC 8037

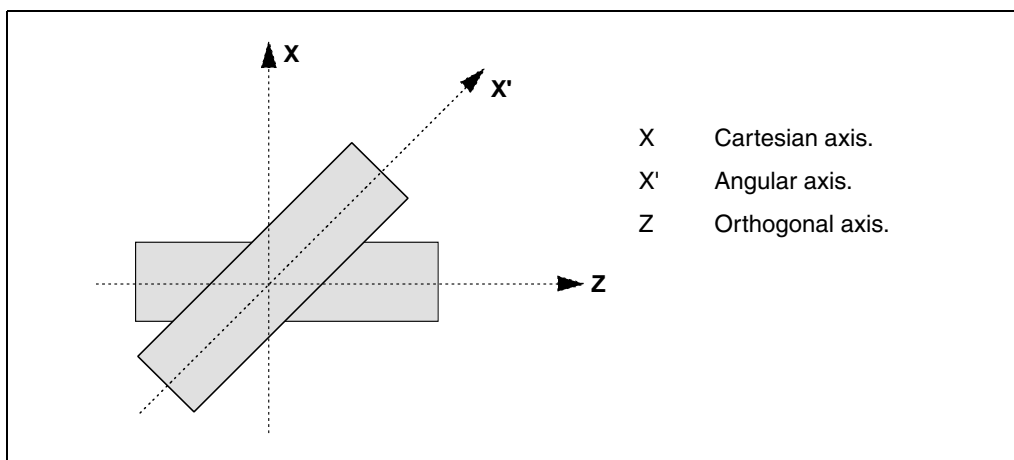
.T. MODEL
SOFT: V01.4x

ANGULAR TRANSFORMATION OF AN INCLINE AXIS

13

With the angular transformation of an incline axis, it is possible to make movements along an axis that is not perpendicular to another. The movements are programmed in the Cartesian system and to make the movements, they are transformed into movements on the real axes.

On certain machines, the axes are configured in a Cartesian way, they are not perpendicular to each other. A typical case is the X axis of a lathe that for sturdiness reasons is not perpendicular to the Z axis.



Programming in the Cartesian system (Z-X) requires activating an angular transformation of an inclined plane that converts the movements of the real (non-perpendicular) axes (Z-X'). This way, a movement programmed on the X axis is transformed into movements on the Z-X' axes; i.e. it then moves along the Z axis and the angular X' axis.

Turning angular transformation on and off.

The CNC assumes no transformation on power-up; the angular transformations are activated via part-program using the instruction G46.

The angular transformations are turned off via part-program using function G46. Optionally, a transformation may be "frozen" (suspended) to move angular axis by programming in Cartesian coordinates.

Influence of the reset, turning the CNC off and of the M30.

The angular transformation of an incline axis stays active after a RESET, M30 and even after turning the CNC off and back on.

Considerations for the angular transformation of an incline axis.

The axes involved in an angular transformation must be linear. Both axes may have Gantry axes associated with them.

If the angular transformation is active, the coordinates displayed will be those of the Cartesian system. Otherwise, it will display the coordinates of the real axes.

The following operations are possible while the transformation is active:

- Zero offsets.
- Coordinate preset.
- Movements in continuous / incremental jog and handwheels.

Movements to hardstop are not possible while the transformation is active.

Home search

Function G46 is canceled when homing an axis that is involved in the angular transformation (machine parameters ANGAXNA and ORTAXNA). When homing the axes that are not involved in the angular axis transformation, function G46 stays active.

While searching home, only the real axes move.

Jogging and handwheel movements.

Either the real or the Cartesian axes may be jogged depending on how they've been set by the manufacturer. It is selected via PLC (MACHMOVE) and it may be available, for example, from a user key.

13.**ANGULAR TRANSFORMATION OF AN INCLINE AXIS****CNC 8037**

·T· MODEL
SOFT: V01.4x

13.1 Turning angular transformation on and off

Turn angular transformation on

When the transformation is on, the movements are programmed in the Cartesian system and to make the movements, the CNC transforms them into movements on the real axes. The coordinates displayed on the screen will be those of the Cartesian system.

The angular transformation is turned on using function G46 whose programming format is:

G46 S1

This instruction turns a "frozen" (suspended) transformation on again. See ["13.2 Freezing the angular transformation"](#) on page 254.

Turning the angular transformation off

If the transformation is off, the movements are programmed and executed in the system of the real axes. The coordinates displayed on the screen will be those of the real axes.

The angular transformation is turned off using function G46 whose programming format is:

G46 S0

G46

The angular transformation of an incline axis stays active after a RESET, M30 and even after turning the CNC off and back on.

13.

ANGULAR TRANSFORMATION OF AN INCLINE AXIS

Turning angular transformation on and off

13.2 Freezing the angular transformation

Freezing the angular transformation is a special way to make movements along the angular axis, but programming it in the Cartesian system. The angular transformation cannot be "frozen" (suspended) while jogging.

The angular transformation is "frozen" (suspended) using function G46 whose programming format is:

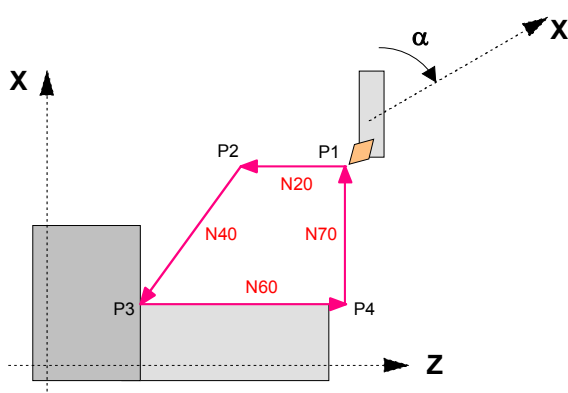
G46 S2

Programming movements after "freezing" the angular transformation.

If an angular transformation is "frozen" (suspended), only the coordinate of the angular axis must be programmed in the motion block. If the coordinate of the orthogonal axis is programmed, the movement is carried out according to the normal angular transformation.

Canceling the freezing of a transformation.

The "freezing" of an angular transformation is canceled after a reset or an M30. Turning the transformation on (G46 S1) also cancels the "freezing".



```

N10 G46 S1
N20 G1 Z(P2)
N30 G46 S2      Freezing the transformation.
N40 X(P3)       Movement programming the coordinate in the Cartesian system ZX.
N50 G46 S1      Activate the normal mode.
N60 Z(P4)
N70 X(P1)
  
```

APPENDIX

A. ISO code programming.....	257
B. Program control instructions.....	259
C. Summary of internal CNC variables.....	261
D. Key code	267
E. Maintenance.....	269

ISO CODE PROGRAMMING

Function	M	D	V	Meaning	Section
G00	*	?	*	Rapid traverse	6.1
G01	*	?	*	Linear interpolation	6.2
G02	*		*	Clockwise circular interpolation	6.3
G03	*		*	Counterclockwise circular interpolation	6.3
G04				Dwell/interruption of block preparation	7.1 / 7.2
G05	*	?	*	Round corner	7.3.2
G06			*	Circle center in absolute coordinates	6.4
G07	*	?		Square corner	7.3.1
G08			*	Arc tangent to previous path.	6.5
G09			*	Arc defined by three points	6.6
G10	*	*		Mirror image cancellation	7.5
G11	*		*	Mirror image on X axis	7.5
G12	*		*	Mirror image on Y axis	7.5
G13	*		*	Mirror image on Z axis	7.5
G14	*		*	Mirror image in the programmed directions	7.5
G16	*		*	Main plane selection by two addresses	3.2
G17	*	?	*	Main plane X-Y and longitudinal Z	3.2
G18	*	?	*	Main plane Z-X and longitudinal Y	3.2
G19	*		*	Main plane Y-Z and longitudinal X	3.2
G20				Definition of lower work zone limits	3.8.1
G21				Definition of upper work zone limits.	3.8.1
G22			*	Enable/disable work zones.	3.8.2
G32	*		*	Feedrate "F" as an inverted function of time.	6.15
G33	*		*	Electronic threading	6.11
G34				Variable-pitch threading	6.13
G36			*	Controlled corner rounding	6.9
G37			*	Tangential entry	6.7
G38			*	Tangential exit	6.8
G39			*	Chamfer	6.10
G40	*	*		Cancellation of tool radius compensation	8.2.6
G41	*		*	Left-hand tool radius compensation	8.2.3
G41 N	*		*	Collision detection	8.3
G42	*		*	Right-hand tool radius compensation	8.2.3
G42 N	*		*	Collision detection	8.3
G50	*		*	Controlled corner rounding	7.3.3
G51	*		*	Look-Ahead	7.4
G52			*	Movement until making contact	6.14
G53			*	Programming with respect to machine zero	4.3
G54	*		*	Absolute zero offset 1	4.4.2
G55	*		*	Absolute zero offset 2	4.4.2
G56	*		*	Absolute zero offset 3	4.4.2
G57	*		*	Absolute zero offset 4	4.4.2
G58	*		*	Additive zero offset 1	4.4.2
G59	*		*	Additive zero offset 2	4.4.2
G66			*	Pattern repeat cycle canned cycle	9.1
G68			*	X axis roughing canned cycle	9.2
G69			*	Z axis roughing canned cycle	9.3
G70	*	?	*	Programming in inches	3.3
G71	*	?		Programming in millimeters	3.3
G72	*		*	General and specific scaling factor	7.6
G74			*	Home search	4.2
G75			*	Probing move until touching	10.1
G76			*	Probing move while touching	10.1
G81			*	Turning canned cycle for straight sections	9.4
G82			*	Facing canned cycle for straight sections	9.5
G83			*	Drilling canned cycle	9.6

A.

ISO code programming

FAGOR 

CNC 8037

 ·T· MODEL
 SOFT: V01.4x

Function	M	D	V	Meaning	Section
G84			*	Turning canned cycle for curved sections	9.7
G85			*	Facing canned cycle for curved sections	9.8
G86			*	Longitudinal threading canned cycle	9.9
G87			*	Face threading canned cycle	9.10
G88			*	X axis grooving canned cycle	9.11
G89			*	Z axis grooving canned cycle	9.12
G90	*	?		Absolute programming:	3.4
G91	*	?	*	Incremental programming	3.4
G92				Coordinate preset / spindle speed limit	4.4.1
G93				Polar origin preset	4.5
G94	*	?		Feedrate in millimeters (inches) per minute	5.2.1
G95	*	?	*	Feedrate in millimeters (inches) per revolution.	5.2.2
G96	*		*	Constant surface speed	5.3.1
G97	*	*		Spindle speed in RPM	5.3.2
G151	*	?		Programming the X axis in diameter.	3.5
G152	*	?		Programming the X axis in radius.	3.5

M means modal, i.e. the G function, once programmed, remains active until another incompatible G function is programmed or until an M02, M30, EMERGENCY or RESET is executed or the CNC is turned off and back on.

D means BY DEFAULT, i.e. they will be assumed by the CNC when it is powered on, after executing M02, M30 or after EMERGENCY or RESET.

In those cases indicated by ?, it should be understood that the DEFAULT of these G functions depends on the setting of the general machine parameters of the CNC.

The letter V means that the G code is displayed next to the current machining conditions in the execution and simulation modes.

PROGRAM CONTROL INSTRUCTIONS

Display instructions.

	(section 12.2)
(ERROR integer, "error text")	Stops the execution of the program and displays the indicated error.
(MSG "message")	Displays the indicated message.
(DGWZ expression 1, expression 4)	Define the graphics area.

Enabling and disabling instructions.

	(section 12.3)
(ESBLK and DSBLK)	The CNC executes all the blocks between ESBLK and DSBLK as if it were a single block.
(ESTOP and DSTOP)	Enabling (ESTOP) and disabling (DSTOP) of the Stop key and the external Stop signal (PLC).
(EFHOLD and DFHOLD)	Enabling (EFHOLD) and disabling (DFHOLD) of the Feed-hold input (PLC).

Flow control instructions.

	(section 12.4)
(GOTO N(expression))	It causes a jump within the same program, to the block defined by the label N(expression).
(RPT N(expression), N(expression), P(expression))	It repeats the execution of the portion of the program between the blocks defined by means of the labels N(expression).
(IF condition <action1> ELSE <action2>)	It analyzes the given condition that must be a relational expression. If the condition is true (result equal to 1), <action1> will be executed, otherwise (result equal to 0) <action2> will be executed.

Subroutine instructions.

	(section 12.5)
(SUB integer)	Subroutine definition.
(RET)	End of subroutine.
(CALL (expression))	Call to a subroutine.
(PCALL (expression), (assignment instruction), (assignment instruction),...)	Call to a subroutine. In addition, using assignment instructions, it is possible to initialize up to a maximum of 26 local parameters of this subroutine.
(MCALL (expression), (assignment instruction), (assignment instruction),...)	Same as the PCALL instruction, but making the indicated subroutine modal.
(MDOFF)	Cancellation of modal subroutine.

B.

Program control instructions

FAGOR 

CNC 8037

 ·T· MODEL
 SOFT: V01.4x

Interruption-subroutine instructions.([section 12.6](#))**(REPOS X, Y, Z,)**

It must always be used inside an interruption subroutine and facilitates the repositioning of the machine axes to the point of interruption.

Program instructions.([section 12.7](#))**(EXEC P(expression), (directory))**

Starts program execution.

(MEXEC P(expression), (directory))

Starts program execution in modal mode.

(OPEN P(expression), (destination directory), A/D, "program comment")

It begins editing a new program being possible to associate a comment with the program.

(WRITE <block text>)

It adds, after the last block of the program which began to be edited by means of the mnemonic OPEN P, the information contained in <block text> as a new program block.

Screen customizing instructions.([section 12.8](#))**(PAGE (expression))**

The screen displays the indicated user page number (0-255) or system page (1000).

(SYMBOL (expression 1), (expression 2), (expression 3))

The screen displays the symbol (0-255) indicated by the expression 1.

Its position on the screen is defined by expression 2 (row, 0-639) and by expression 3 (column 0-335).

(IB (expression) = INPUT "text", format)

It displays the text indicated in the data input window and stores in the input variable (IBn) the data entered by the user.

(ODW (expression 1), (expression 2), (expression 3))

It defines and draws a white window on the screen (1 row and 14 columns).

Its position on screen is defined by expression 2 (row) and by expression 3 (column).

(DW(expression 1) = (expression 2), DW (expression 3) = (expression 4), ...)

It displays in the windows indicated by the value of the expression 1, 3,..., the numerical data indicated by the expression 2,4,..

(SK (expression 1) = "text1" (expression 2) = "text 2",)

It defines and displays the new softkey menu indicated.

(WKEY)

It stops the execution of the program until a key is pressed.

(WBUF "text", (expression))

It adds the text and value of the expression, once it has been evaluated, to the block that is being edited and within the data entry window.

(WBUF)

Enters the block being edited into memory. It can only be used in the screen customizing program to be executed in the Editing mode.

(SYSTEM)

It ends the execution of the user screen customizing program and returns to the corresponding standard menu of the CNC.

B.

Program control instructions

**CNC 8037**

·T· MODEL
SOFT: V01.4x

SUMMARY OF INTERNAL CNC VARIABLES.

- The R symbol indicates that the variable can be read.
- The W symbol indicates that the variable can be modified.

Variables associated with tools.

Variable	CNC	PLC	DNC	(section 11.2.2)
TOOL	R	R	R	Number of the active tool.
TOD	R	R	R	Number of active tool offset.
NXTOOL	R	R	R	Number of the next requested tool waiting for M06.
NXTOD	R	R	R	Number of the next tool's offset.
TMZPn	R	R	-	(n) tool's position in the tool magazine.
TLFDn	R/W	R/W	-	(n) tool's offset number.
TLFFn	R/W	R/W	-	(n) tool's family code.
TLFNn	R/W	R/W	-	Nominal life assigned to tool (n).
TLFRn	R/W	R/W	-	Real life value of tool (n).
TMZTn	R/W	R/W	-	Contents of tool magazine position (n).
HTOR	R/W	R	R	Tool radius being used by the CNC to do the calculations.
TOXn	R/W	R/W	-	Tool length offset (n) along X axis.
TOZn	R/W	R/W	-	Tool length offset (n) along Z axis.
TOFn	R/W	R/W	-	Location code of offset (n).
TORn	R/W	R/W	-	Tool radius value of offset (n).
TOIn	R/W	R/W	-	Tool length wear of offset (n) along X axis.
TOKn	R/W	R/W	-	Tool length wear of offset (n) along Z axis.
NOSEAn:	R/W	R/W	-	Cutter angle of indicated tool.
NOSEWn	R/W	R/W	-	Width of indicated tool.
CUTAn	R/W	R/W	-	Cutting angle of indicated tool.

Variables associated with zero offsets.

Variable	CNC	PLC	DNC	(section 11.2.3)
ORG(X-C)	R	R	-	Active zero offset on the selected axis. The value of the additive offset indicated by the PLC is not included.
PORGF	R	-	R	Abcissa coordinate value of polar origin.
PORGS	R	-	R	Ordinate coordinate value of polar origin.
ORG(X-C)n	R/W	R/W	R	Zero offset (n) value of the selected axis.
PLCOF(X-C)	R/W	R/W	R	Value of the additive zero offset activated via PLC.
ADIOF(X-C)	R	R	R	Value for the selected axis of the zero offset with additive handwheel.

Variables associated with machine parameters.

Variable	CNC	PLC	DNC	(section 11.2.4)
MPGn	R	R	-	Value assigned to general machine parameter (n).
MP(X-C)n	R	R	-	Value assigned to (X-C) axis machine parameter (n).
MPSn	R	R	-	Value assigned to machine parameter (n) of the main spindle.
MPLCn	R	R	-	Value assigned to machine parameter (n) of the PLC.

C.

Summary of internal CNC variables.

FAGOR


CNC 8037

 ·T· MODEL
 SOFT: V01.4x

Variables associated with work zones.

Variable	CNC	PLC	DNC	(section 11.2.5)
FZONE	R	R/W	R	Status of work zone 1.
FZLO(X-C)	R	R/W	R	Work zone 1. Lower limit along the selected axis (X/C).
FZUP(X-C)	R	R/W	R	Work zone 1. Upper limit along the selected axis (X-C).
SZONE	R	R/W	R	Status of work zone 2.
SZLO(X-C)	R	R/W	R	Work zone 2. Lower limit along the selected axis (X/C).
SZUP(X-C)	R	R/W	R	Work zone 2. Upper limit along the selected axis (X-C).
TZONE	R	R/W	R	Status of work zone 3.
TZLO(X-C)	R	R/W	R	Work zone 3. Lower limit along the selected axis (X/C).
TZUP(X-C)	R	R/W	R	Work zone 3. Upper limit along the selected axis (X-C).
FOZONE	R	R/W	R	Status of work zone 4.
FOZLO(X-C)	R	R/W	R	Work zone 4. Lower limit along the selected axis (X/C).
FOZUP(X-C)	R	R/W	R	Work zone 4. Upper limit along the selected axis (X-C).
FIZONE	R	R/W	R	Status of work zone 5.
FIZLO(X-C)	R	R/W	R	Work zone 5. Lower limit along the selected axis (X/C).
FIZUP(X-C)	R	R/W	R	Work zone 5. Upper limit along the selected axis (X-C).

Feedrate related variables.

Variable	CNC	PLC	DNC	(section 11.2.6)
FREAL	R	R	R	Real feedrate of the CNC in mm/min or inch/min.
FREAL(X-C)	R	R	R	Actual (real) CNC feedrate of the selected axis.
FTEO(X-C)	R	R	R	Theoretical CNC feedrate of the selected axis.

Variables associated with function G94.

FEED	R	R	R	Active feedrate at the CNC in mm/min or inch/min.
DNCF	R	R	R/W	Feedrate selected via DNC.
PLCF	R	R/W	R	Feedrate selected via PLC.
PRGF	R	R	R	Feedrate selected by program.

Variables associated with function G94.

FPREV	R	R	R	Active feedrate at CNC, in m/rev or inch/rev.
DNCFPR	R	R	R/W	Feedrate selected via DNC.
PLCFPR	R	R/W	R	Feedrate selected via PLC.
PRGFPR	R	R	R	Feedrate selected by program.

Variables associated with function G94.

PRGFIN	R	R	R	Feedrate selected by program, in 1/min.
--------	---	---	---	---

Variables associated with feedrate override (%)

FRO	R	R	R	Feedrate Override (%) active at the CNC.
PRGFRO	R/W	R	R	Override (%) selected by program.
DNCFRO	R	R	R/W	Override (%) selected via DNC.
PLCFRO	R	R/W	R	Override (%) selected via PLC.
CNCFRO	R	R	R	Override (%) selected from the front panel knob.
PLCCFR	R	R/W	R	Override (%) of the PLC execution channel.

Summary of internal CNC variables.

**CNC 8037**

·T· MODEL
SOFT: V01.4x

Coordinate related variables.

Variable	CNC	PLC	DNC	(section 11.2.7)
PPOS(X-C)	R	-	-	Programmed theoretical position value (coordinate).
POS(X-C)	R	R	R	Machine coordinates. Real coordinates of the tool base.
TPOS(X-C)	R	R	R	Machine coordinates. Theoretical coordinates of the tool base.
APOS(X-C)	R	R	R	Part coordinates. Real coordinates of the tool base.
ATPOS(X-C)	R	R	R	Part coordinates. Theoretical coordinates of the tool base.
FLWE(X-C)	R	R	R	Following error of the indicated axis.
DIST(X-C)	R/W	R/W	R	Distance traveled by the indicated axis.
LIMPL(X-C)	R/W	R/W	R	Second upper travel limit.
LIMMI(X-C)	R/W	R/W	R	Second lower travel limit.
DPLY(X-C)	R	R	R	Coordinate of the selected axis displayed on the screen.
GPOS(X-C)n p	R	-	-	Coordinate of the selected axis, programmed in the (n) block of the program (p).

Variables associated with electronic handwheels.

Variable	CNC	PLC	DNC	(section 11.2.8)
HANPF	R	R	-	Pulses received from 1st handwheel since the CNC was turned on.
HANPS	R	R	-	Pulses received from 2nd handwheel since the CNC was turned on.
HANPT	R	R	-	Pulses received from 3rd handwheel since the CNC was turned on.
HANPFO	R	R	-	Pulses received from 4th handwheel since the CNC was turned on.
HANDSE	R	R		For handwheels with a selector button, it indicates whether that button has been pressed or not.
HANFCT	R	R/W	R	Multiplying factor different for each handwheel (when having several).
HBEVAR	R	R/W	R	HBE handwheel. Reading enabled, axis being jogged and multiplying factor (x1, x10, x100).
MASLAN	R/W	R/W	R/W	Linear path angle for "Path handwheel" or "Path Jog" mode.
MASCFI	R/W	R/W	R/W	Arc center coordinates for "Path handwheel mode" or "Path jog".
MASCSE	R/W	R/W	R/W	Arc center coordinates for "Path handwheel mode" or "Path jog".

Feedback related variables.

Variable	CNC	PLC	DNC	(section 11.2.9)
ASIN(X-C)	R	R	R	A signal of the CNC's sinusoidal feedback for the selected axis.
BSIN(X-C)	R	R	R	B signal of the CNC's sinusoidal feedback for the selected axis.
ASINS	R	R	R	"A" signal of the CNC's sinusoidal feedback for the spindle.
BSINS	R	R	R	"B" signal of the CNC's sinusoidal feedback for the spindle.

Variables associated with the main spindle.

Variable	CNC	PLC	DNC	(section 11.2.10)
SREAL	R	R	R	Real spindle speed
FTEOS	R	R	R	Theoretical spindle speed.

Variables associated with spindle speed.

SPEED	R	R	R	Active spindle speed at the CNC.
DNCS	R	R	R/W	Spindle speed selected via DNC.
PLCS	R	R/W	R	Spindle speed selected via PLC.
PRGS	R	R	R	Spindle speed selected by program.

Summary of internal CNC variables.

FAGOR **CNC 8037**.T- MODEL
SOFT: V01.4x

Variables associated with constant surface speed.

CSS	R	R	R	Constant surface speed active at the CNC.
DNCCSS	R	R	R/W	Constant surface speed selected via DNC.
PLCCSS	R	R/W	R	Constant surface speed selected via PLC.
PRGCSS	R	R	R	Constant surface speed selected by program.

Variables associated with the spindle override.

SSO	R	R	R	Spindle Speed Override (%) active at the CNC.
PRGSSO	R/W	R	R	Override (%) selected by program.
DNCSSO	R	R	R/W	Override (%) selected via DNC.
PLCSSO	R	R/W	R	Override (%) selected via PLC.
CNCSSO	R	R	R	Spindle Speed Override (%) selected from front panel.

Speed limit related variables.

SLIMIT	R	R	R	Spindle speed limit active at the CNC.
DNCSL	R	R	R/W	Spindle speed limit selected via DNC.
PLCSL	R	R/W	R	Spindle speed limit selected via PLC.
PRGSL	R	R	R	Spindle speed limit selected by program.
MDISL	R	R/W	R	Maximum machining spindle speed.

Position related variables.

POSS	R	R	R	Real Spindle position. Reading from the PLC in ten-thousandths of a degree (within ± 999999999) and from the CNC in degrees (within ± 99999.9999).
RPOSS	R	R	R	Real Spindle position. Reading from the PLC in ten-thousandths of a degree (between -3600000 and 3600000) and from the CNC in degrees (between -360 and 360).
TPOSS	R	R	R	Theoretical spindle position. Reading from the PLC in ten-thousandths of a degree (within ± 999999999) and from the CNC in degrees (within ± 99999.9999).
RTPOSS	R	R	R	Theoretical spindle position. Reading from the PLC in ten-thousandths of a degree (between 0 and 3600000) and from the CNC in degrees (between 0 and 360).
PRGSP	R	R	R	Position programmed in M19 via program for the main spindle.

Variables related to the following error.

FLWES	R	R	R	Spindle following error.
-------	---	---	---	--------------------------

PLC related variables.

Variable	CNC	PLC	DNC	(section 11.2.11)
PLCMSG	R	-	R	Number of the active PLC message with the highest priority.
PLCIn	R/W	-	-	32 PLC inputs starting from (n).
PLCO _n	R/W	-	-	32 PLC outputs starting from (n).
PLCM _n	R/W	-	-	32 PLC marks starting from (n).
PLCR _n	R/W	-	-	(n) Register.
PLCT _n	R/W	-	-	Indicated (n) Timer's count.
PLCC _n	R/W	-	-	Indicated (n) Counter's count.
PLCMM _n	R/W	-	-	Modifies the (n) mark of the PLC.

Variables associated with local and global parameters.

Variable	CNC	PLC	DNC	(section 11.2.12)
GUP n	-	R/W	-	Global parameter (P100-P299) (n).
LUP (a,b)	-	R/W	-	Indicated local (P0-P25) parameter (b) of the nesting level (a).
CALLP	R	-	-	Indicates which local parameters have been defined by means of a PCALL or MCALL instruction (calling a subroutine).

Operating-mode related variables.

Variable	CNC	PLC	DNC	(section 11.2.13)
OPMODE	R	R	R	Operating mode.
OPMODA	R	R	R	Operating mode when working in the main channel.
OPMODB	R	R	R	Type of simulation.
OPMODC	R	R	R	Axes selected by handwheel.

Other variables.

Variable	CNC	PLC	DNC	(section 11.2.14)
NBTOOL	R	-	R	Number of the tool being managed..
PRGN	R	R	R	Number of the program in execution.
BLKN	R	R	R	Label number of the last executed block.
GSn	R	-	-	Status of the indicated G function (n).
GGSA	-	R	R	Status of functions G00 thru G24.
GGSB	-	R	R	Status of functions G25 thru G49.
GGSC	-	R	R	Status of functions G50 thru G74.
GGSD	-	R	R	Status of functions G75 thru G99.
MSn	R	-	-	Status of the indicated M function (n)
GMS	-	-	R	Status of M functions: M (0..6, 8, 9, 19, 30, 41..44).
PLANE	R	R	R	Abscissa and ordinate axes of the active plane.
LONGAX	R	R	R	Axis affected by the tool length compensation (G15).
MIRROR	R	R	R	Active mirror images.
SCALE	R	R	R	General scaling factor applied. Reading from the PLC in ten-thousandths.
SCALE(X-C)	R	R	R	Scaling Factor applied only to the indicated axis. Reading from the PLC in ten-thousandths.
ORGROT	R	R	R	Rotation angle (G73) of the coordinate system.
PRBST	R	R	R	Returns probe status.
CLOCK	R	R	R	System clock in seconds.
TIME	R	R	R/W	Time in Hours, minutes and seconds.
DATE	R	R	R/W	Date in Year-Month-Day format
TIMER	R/W	R/W	R/W	Clock activated by PLC, in seconds.
CYTIME	R	R	R	Time to execute a part in hundredths of a second.
PARTC	R/W	R/W	R/W	Parts counter of the CNC.
FIRST	R	R	R	First time a program is executed.
KEY	R/W	R/W	R/W	keystroke code.
KEYSRC	R/W	R/W	R/W	Source of the keys.
ANAIn	R	R	R	Voltage (in volts) of the indicated analog input (n).
ANAO n	R/W	R/W	R/W	Voltage (in volts) to apply to the indicated output (n).
CNCERR	-	R	R	Active CNC error number.

C.

Summary of internal CNC variables.

FAGOR **CNC 8037**·T· MODEL
SOFT: V01.4x

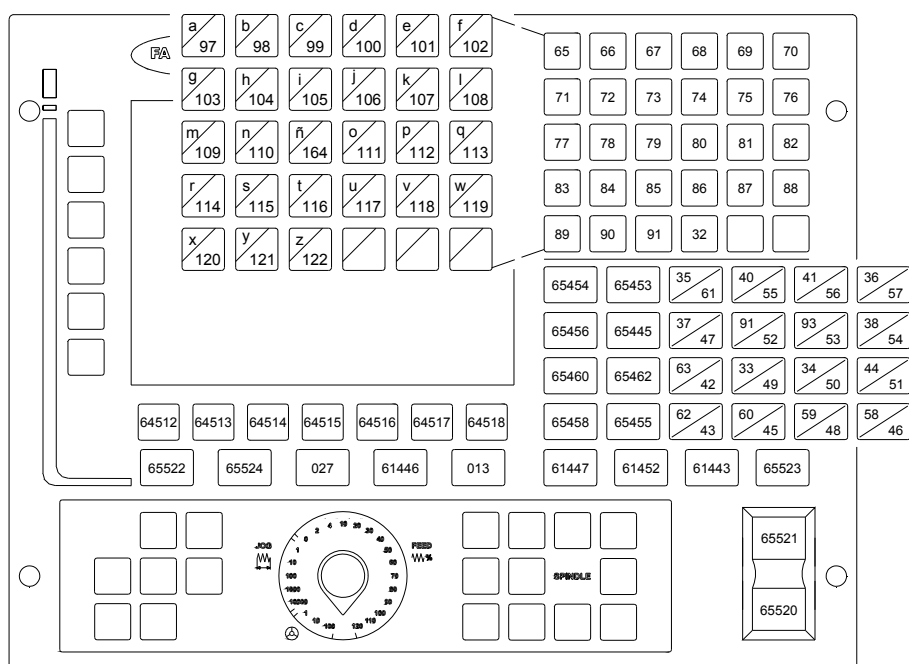
Variable	CNC	PLC	DNC	(section 11.2.14)
PLCERR	-	-	R	Active PLC error number.
DNCERR	-	R	-	Number of the error generated during DNC communications.
DNCSTA	-	R	-	DNC transmission status.
TIMEG	R	R	R	Remaining time to finish the dwell block (in hundredths of a second)..
SELPRO	R/W	R/W	R	When having two probe inputs, it selects the active input.
DIAM	R/W	R/W	R	It changes the programming mode for X axis coordinates between radius and diameter.
PRBMOD	R/W	R/W	R	Indicates whether a probing error must be displayed or not.
RIP	R	R	R	Linear theoretical feedrate resulting from the next loop (in mm/min).
FBDIF(X-C)	R	R	R	For monitoring the difference between the coordinates (position values) of the first and second feedback on the oscilloscope
CYCLEV	R	R	R	It indicates on the conversational model, the tab number being displayed at a particular moment.
CYCEDI	R	R	R	It indicates on the conversational model, the cycle or screen number being displayed at a particular moment.
DISBLO	R	R	R	It indicates the value of the total programmed distance in blocks with look-ahead.
MIXPO(X..C)	R	R	R	Variable that indicates the axis position with the combined feedback.
FLWAC(X..C)	R	R	R	Variable that indicates the real error considering the second feedback.
RADIOC	R	-	-	It indicates the programmed radius value when executing a G15 R.



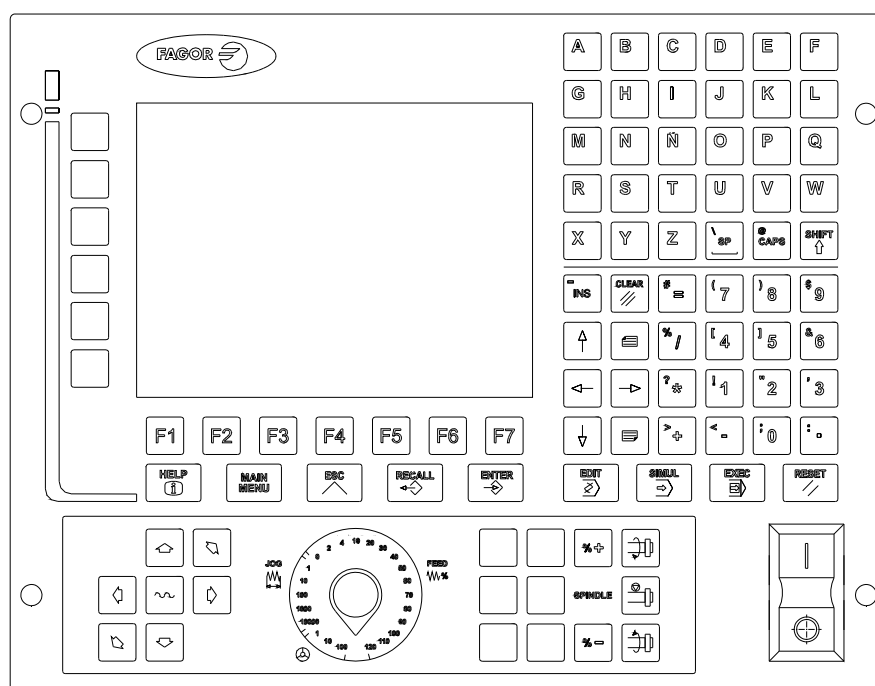
*The "KEY" variable can be "written" at the CNC only via the user channel.
The "NBTOOL" variable can only be used within the tool change subroutine.*

KEY CODE

Alphanumeric operator panel (M-T models)


D.

Key code


FAGOR

CNC 8037

 ·T· MODEL
 SOFT: V01.4x

D.

Key code



CNC 8037

·T· MODEL
SOFT: V01.4x

MAINTENANCE

Cleaning

The accumulated dirt inside the unit may act as a screen preventing the proper dissipation of the heat generated by the internal circuitry which could result in a harmful overheating of the CNC and, consequently, possible malfunctions.

On the other hand, accumulated dirt can sometimes act as an electrical conductor and short-circuit the internal circuitry, especially under high humidity conditions.

To clean the operator panel and the monitor, a smooth cloth should be used which has been dipped into de-ionized water and /or non abrasive dish-washer soap (liquid, never powder) or 75° alcohol.

Do not use highly compressed air to clean the unit because it could generate electrostatic discharges.

The plastics used on the front panel are resistant to :

- Grease and mineral oils.
- Bases and bleach.
- Dissolved detergents.
- Alcohol.



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

To check the fuses, first unplug the unit from mains. If the CNC does not turn on when flipping the power switch, check that the fuses are the right ones and they are in good condition.

Avoid solvents. The action of solvents such as chlorine hydrocarbons, benzole, esters and ether may damage the plastics used to make the front panel of the unit.

Do not get into the inside of the unit. Only personnel authorized by Fagor Automation may manipulate the inside of this unit.

Do not handle the connectors with the unit connected to main AC power. Before handling these connectors (I/O, feedback, etc.), make sure that the unit is not connected to main AC power.

E.

Maintenance

FAGOR **CNC 8037**·T· MODEL
SOFT: V01.4x

E.

Maintenance



CNC 8037

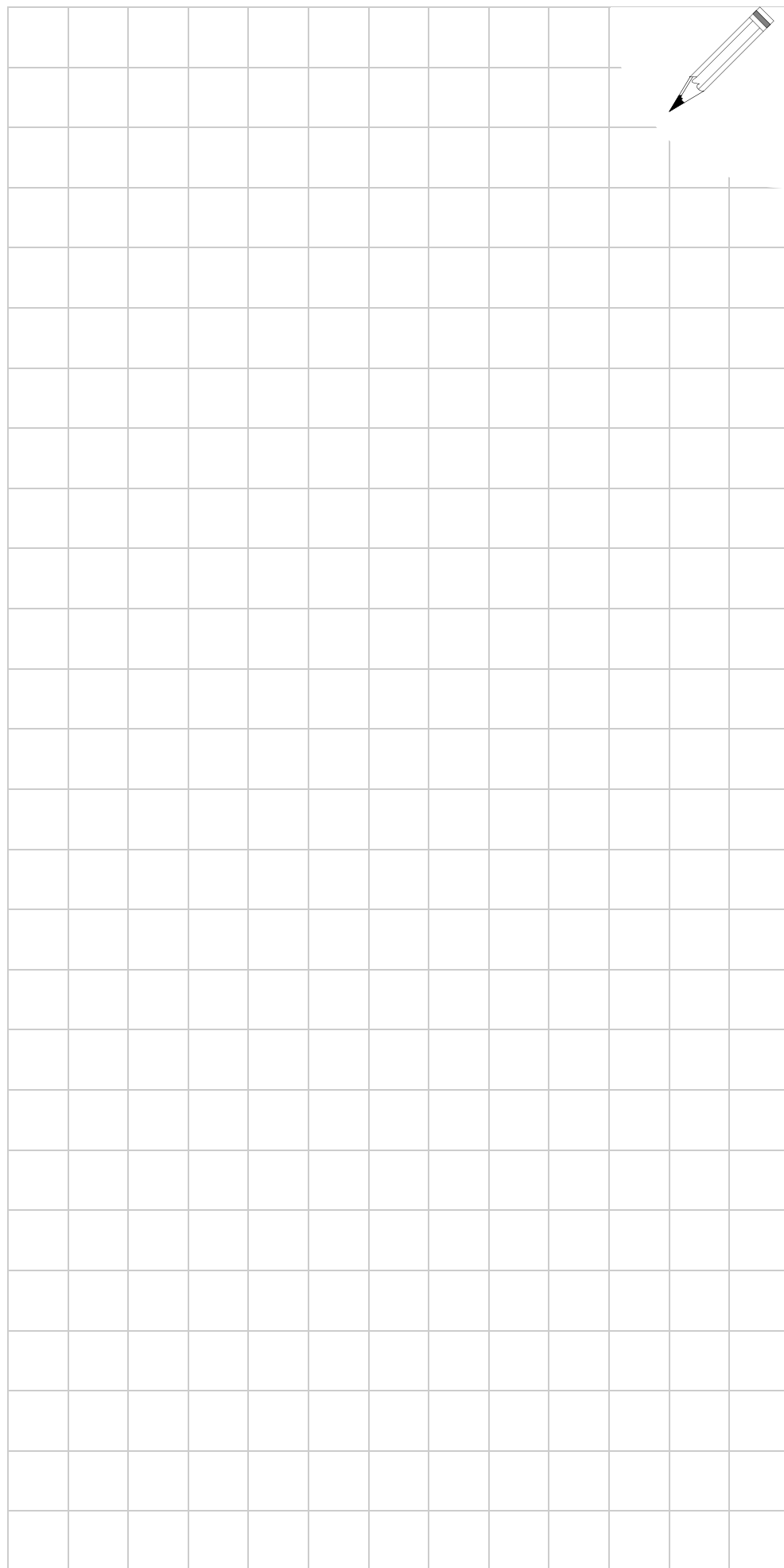
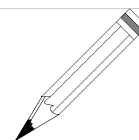
·T· MODEL
SOFT: V01.4x



CNC 8037

SOFT: V01.4x

E.



CNC 8037

SOFT: V01.4x