

CNC

8060

8065

ProGTL3 language.

(Ref: 1709)



FAGOR AUTOMATION

TRANSLATION OF THE ORIGINAL MANUAL

This manual is a translation of the original manual. This manual, as well as the documents derived from it, have been drafted in Spanish. In the event of any contradictions between the document in Spanish and its translations, the wording in the Spanish version shall prevail. The original manual will be labeled with the text "ORIGINAL MANUAL".

MACHINE SAFETY

It is up to the machine manufacturer to make sure that the safety of the machine is enabled in order to prevent personal injury and damage to the CNC or to the products connected to it. On start-up and while validating CNC parameters, it checks the status of the following safety elements. If any of them is disabled, the CNC shows the following warning message.

- Feedback alarm for analog axes.
- Software limits for analog and sercos linear axes.
- Following error monitoring for analog and sercos axes (except the spindle) both at the CNC and at the drives.
- Tendency test on analog axes.

FAGOR AUTOMATION shall not be held responsible for any personal injuries or physical damage caused or suffered by the CNC resulting from any of the safety elements being disabled.

HARDWARE EXPANSIONS

FAGOR AUTOMATION shall not be held responsible for any personal injuries or physical damage caused or suffered by the CNC resulting from any hardware manipulation by personnel unauthorized by Fagor Automation.

If the CNC hardware is modified by personnel unauthorized by Fagor Automation, it will no longer be under warranty.

COMPUTER VIRUSES

FAGOR AUTOMATION guarantees that the software installed contains no computer viruses. It is up to the user to keep the unit virus free in order to guarantee its proper operation. Computer viruses at the CNC may cause it to malfunction.

FAGOR AUTOMATION shall not be held responsible for any personal injuries or physical damage caused or suffered by the CNC due a computer virus in the system.

If a computer virus is found in the system, the unit will no longer be under warranty.

DUAL-USE PRODUCTS

Products manufactured by FAGOR AUTOMATION since April 1st 2014 will include "-MDU" in their identification if they are included on the list of dual-use products according to regulation UE 428/2009 and require an export license depending on destination.



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.

INDEX

About the product - CNC 8060	5
About the product - CNC 8065	9
Declaration of CE conformity and warranty conditions	13
Version history - CNC 8060	15
Version history - CNC 8065	17
Safety conditions	19
Returning conditions	23
CNC maintenance	25
CHAPTER 1	INTRODUCTION.
<hr/>	
1.1	G functions to expand geometry features. 28
1.2	Summary of G function syntax. 30
CHAPTER 2	PROGTL3 LANGUAGE.
<hr/>	
2.1	G810-G811. Straight line through two points or tangent to two circles. 32
2.2	G813. Definition of a straight line that goes through a point or is tangent to a circle and forms a known angle with the abscissa axis. 35
2.3	G820. Define a circle or a point. 36
2.4	G821. Define a chamfer. 37
2.5	G821. Define a rounding or an arc. 37
2.6	G840. Linear or tangential exit and cancellation of tool radius compensation. 38
2.7	G841/G842. Begin profile with left-hand or right-hand tool radius compensation. 39
2.8	G850. Cancel rotational offset. 40
2.9	G851. Activate rotational offset. 40
CHAPTER 3	ISO FUNCTIONS.
<hr/>	
3.1	G801. Define a ray or segment. 42
3.2	G802/G803. Define a circle of an arc. 44
3.3	G808. Arc tangent to previous path. 46
3.4	G809. Arc going through three points. 47
3.5	G831. Define a ray or segment tangent to the previous element. 48
3.6	G832/G833. Define a circle or arc tangent to the previous element. 50
CHAPTER 4	INCREMENTAL PROGRAMMING WITH THE "I" COMMAND.
<hr/>	
4.1	G801. Ray or segment. 51
4.2	G802/G803. Clockwise/counterclockwise circle or arc. 52
4.3	G808. Arc tangent to previous path. 52
4.4	G809. Arc by two points and beginning at the previous end point. 52
4.5	G810. First point or circle to define a straight line. 53
4.6	G811. Second point or circle to define a straight line. 53
4.7	G813. Straight line that goes through a point or is tangent to a circle and forms a known angle with the abscissa axis. 53
4.8	G820. Define a circle or a point. 54
4.9	G831. Define a rounding or an arc. Define a chamfer. 54
4.10	G832/G833. Clockwise/counterclockwise circle or arc tangent to the previous path. 54
4.11	G851. Rototranslation. 55
4.12	Examples. 55
CHAPTER 5	GEOMETRICAL ENTITIES.
<hr/>	
5.1	Saving the various types of geometrical entities. 58
5.1.1	Saving a point. 58
5.1.2	Saving a straight line. 61
5.1.3	Saving circles. 65
5.1.4	Saving origin changes. 72
5.1.5	Saving a distance. 73
5.2	Using the saved entities inside a profile. 74
5.3	Variables related to the geometrical entities. 75
5.4	Summary of the possible ways to define the geometrical entities. 77



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

CHAPTER 6	PROGRAMMING CYCLES (MACROS).	
		<hr/>
6.1	G8726/ G8727/G8728. Ruled surfaces.	79
6.1.1	Programming example.....	82
6.2	G8734/G8735. Spiral profile milling.....	83
6.2.1	Programming example.....	84
6.3	G8736/G8737/G8738. Milling of a surface defined by a plane profile and one or more section profiles.....	85
6.3.1	Rules for defining section profiles.	89
6.3.2	Programming example (1)	91
6.3.3	Programming example (2) G8736 using D7/D8/D9.	93
6.4	G8754/G8753. Invert the direction of a profile.....	94
6.5	G8777/G8778. Circular polygon pocket defined by a profile.	95
6.5.1	Programming example.....	96
CHAPTER 7	USER SUBROUTINES.	
		<hr/>
7.1	Generic user subroutines (G8000-G8999).	97
7.2	Assistance for subroutines.....	100
7.2.1	Subroutine help files.	100
7.2.2	List of available subroutines.....	102
CHAPTER 8	AUXILIARY FUNCTIONS.	
		<hr/>
CHAPTER 9	PROGRAMMING EXAMPLES.	
		<hr/>



CNC 8060
CNC 8065

(REF: 1709)

ABOUT THE PRODUCT - CNC 8060

BASIC CHARACTERISTICS.

Basic characteristics.	8060 M FL	8060 M Power	8060 T FL	8060 T Power	8060 L
Number of axes.	3 to 4	3 to 6	3 to 4	3 to 6	3 to 6
Number of spindles.	1	1 to 2	1 to 2	1 to 3	1
Maximum number of axes and spindles.	5	7	5	7	7
Interpolated axes.	4	4	4	4	4
Number of tool magazines.	1	1	1	1 to 2	1
Number of execution channels.	1	1	1	1 to 2	1
Number of handwheels.	1 to 3				
Type of servo system.	Analog / Sercos Digital				
Communications.	RS485 / RS422 / RS232 Ethernet				
Integrated PLC. PLC execution time. Digital inputs / Digital outputs. Marks / Registers. Timers / Counters. Symbols.	< 1ms/K 1024 / 1024 8192 / 1024 512 / 256 Unlimited				
Block processing time.	< 2.0 ms	< 1.5 ms	< 2.0 ms	< 1.5 ms	< 1 ms

Remote modules.	RIOW	RIO5	RIO70	RIOR	RCS-S
Valid for CNC.	8070 8065 8060	8070 8065 8060	8070 8065 ---	8070 8065 8060	8070 8065 8060
Communication with the remote modules.	CANopen	CANopen	CANfagor	CANopen	Sercos
Digital inputs per module.	8	24 / 48	16	48	---
Digital outputs per module.	8	16 / 32	16	32	---
Analog inputs per module.	4	4	8	---	---
Analog outputs per module.	4	4	4	---	4
Inputs for PT100 temperature sensors.	2	2	---	---	---
Feedback inputs.	---	---	4 (*)	---	4 (**)

(*) Differential TTL / Sinusoidal 1 Vpp (**) TTL / Differential TTL / Sinusoidal 1 Vpp / SSI protocol / FeeDat / EnDat

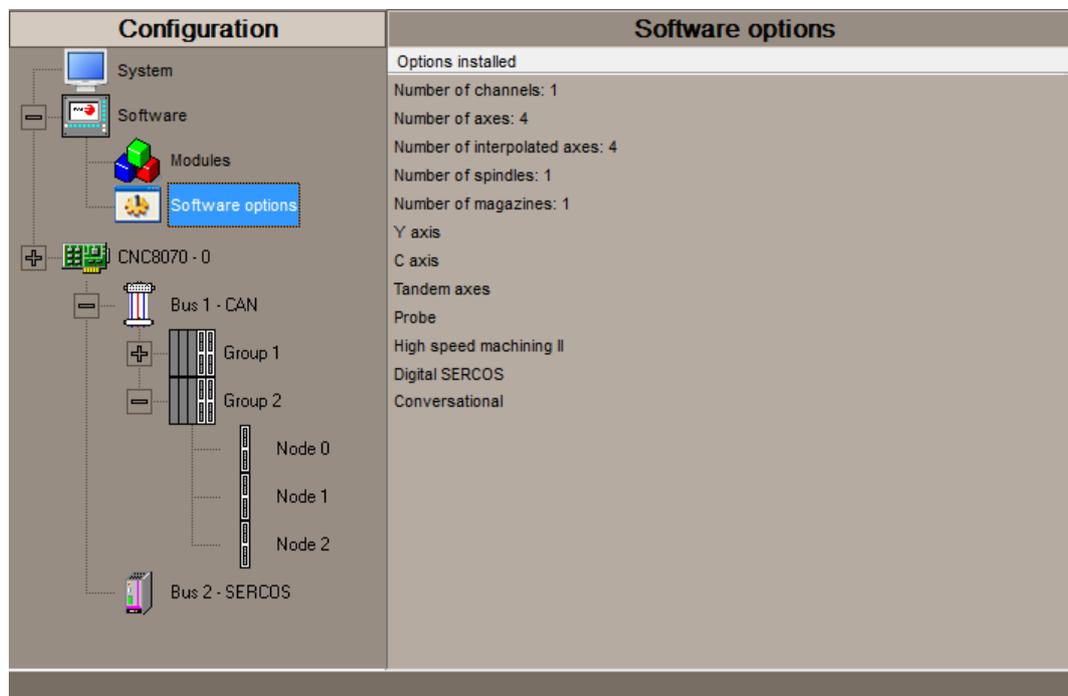


CNC 8060
CNC 8065

(REF: 1709)

SOFTWARE OPTIONS.

Some of the features described in this manual are dependent on the acquired software options. The active software options for the CNC can be consulted in the diagnostics mode (accessible from the task window by pressing [CTRL] [A]), under software options.



Consult the ordering handbook for information on the software options available for your model.

SOFT 8060 ADDIT AXES

Additional shaft.

Add axes to the default configuration.

SOFT 8060 ADDIT SPINDLES

Additional spindle.

Add spindles to the default configuration.

SOFT 8060 ADDIT TOOL MAGAZ

Additional tool magazine.

Add tool magazines to the default configuration.

SOFT 8060 ADDIT CHANNELS

Additional channel.

Add channels to the default configuration.

SOFT DIGITAL SERCOS

Sercos digital bus.

Sercos digital bus.

SOFT EDIT/SIMUL

EDISIMU mode (editing and simulation).

It allows for the editing, modification and simulation of a part-program.

SOFT TOOL RADIUS COMP

Compensación de radio.

Tool compensation allows programming the contour to be machined based on part dimensions of the and without taking into account the dimensions of the tool that will be used later on. This avoids having to calculate and define the tool path based on the tool radius.

SOFT PROFILE EDITOR

Profile editor.

Allows for the part profiles to be edited graphically and to import dxf files.

SOFT 60 F3D GRAPHICS

F3D graphics.

High definition solid 3D graphics for the execution and simulation of part-programs and canned cycles of the editor.

During machining, the F3D graphics display the tool removing the material from the part in real time, allowing for the condition of the part to be seen at all times. F3D graphics can display up to 4 views of the part, where each can be rotated, zoomed in or zoomed out. Measurements can also be made on the part and even sections on the piece from any angle.

SOFT 60 IIP CONVERSATIONAL

Interactive Icon-based Pages (conversational mode).

IIP or conversational mode is specifically designed for people without any prior programming knowledge or for those that are not familiarized with Fagor CNC's.

Working in conversational mode is easier than in ISO mode, as it ensures proper data entry and minimizes the number of operations to be defined. There is no need to work with part-programs.

SOFT 60 RTCP

Dynamic RTCP (Rotating Tool Center Point).

The dynamic RTCP option is required for interpolation machining with 4, 5 or 6 axis.



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

SOFT 60 C AXIS

C axis.

It activates the kinematics for working with the C axis and the associated canned cycles. The CNC can control several C axes. The parameters of each axis indicate if it will function as a C axis or not, where it will not be necessary to activate another axis for the machine parameters.

SOFT 60 Y AXIS

Y axis for lathe.

It activates the kinematics for working with the Y axis and the associated canned cycles.

SOFT 60 TANDEM AXES

Tandem axes.

A tandem axis consists in two motors mechanically coupled (slaved) and making up a single transmission system (axis or spindle). A tandem axis helps provide the necessary torque to move an axis when a single motor is not capable of supplying enough torque to do it.

When activating this feature, it should be kept in mind that for each tandem axis of the machine, another axis must be added to the entire configuration. For example, on a large 3-axis lathe (X Z and tailstock), if the tailstock is a tandem axis, the final purchase order for the machine must indicate 4 axes.

SOFT 60 SYNCHRONISM

Synchronization of axes and spindles.

The axes and ballscrews may be synchronized in two ways: in terms of speed or position. The CNC configuration takes into consideration the synchronization of 2 axes or 2 spindles. Once synchronized, only the master displays and programs the element.

SOFT 60 HSSA I MACHINING SYSTEM

High Speed Surface Accuracy.

This is the new version of algorithms for high speed machining (HSC). This new HSSA algorithm allows for high speed machining optimization, where higher cutting speeds, smoother contours, a better surface finishing and greater precision are achieved.

SOFT 60 HSSA II MACHINING SYSTEM

HSSA-II machining system.

This is the new version of algorithms for high speed machining (HSC). This new HSSA algorithm allows for high speed machining optimization, where higher cutting speeds, smoother contours, a better surface finishing and greater precision are achieved.

SOFT 60 PROBE

Probing canned cycles.

The CNC may have two probes; usually a tabletop probe to calibrate tools and a measuring probe to measure the part.

This option activates the functions G100, G103 and G104 (for probe movements) and probe canned cycles (which help to measure part surfaces and to calibrate tools).

For the laser model, it only activates the non-cycle function G100.

SOFT 60 CONV USER CYCLES

Conversational user cycles.

Incorporation of user cycles in conversational mode.

SOFT 60 PROGTL3

ProGTL3 programming language

Another language apart from ISO for the programming of profiles using a geometric language without the need to use external CAD systems. This language allows for program functions to define lines and circles which define the points of intersection on a profile, in addition to macros for the creation of solids defined by a flat profile and one or more section profiles.

SOFT 60 PPTRANS

Part-program translator.

The program translator may be used to convert programs written in other languages into Fagor ISO codes.

SOFT THIRD PARTY CANOPEN

Third-party CANopen.

Enables the use of non-Fagor CANopen modules.

SOFT MAB SYSTEM.

MAB drives.

Sercos connection with MAB drives.

SOFT 60 PWM CONTROL

Pulse-Width Modulation.

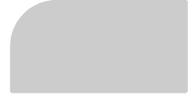
This function is only available for Sercos bus controlled systems. It is mostly oriented toward laser machines for the cutting of very thick sheets, where the CNC generates a series of PWM pulses to control the power of the laser when drilling the starting point.

This feature is essential for cutting very thick sheets and it requires two quick digital outputs located on the central unit. With this new feature, the OEM does not need to install or program any external device, which reduces machine costs and installation times. The end user also benefits, since the "Cutting with PWM" feature is much easier to use and program.

SOFT 60 GAP CONTROL

Gap control.

This is mostly oriented toward laser machines. Gap control makes it possible to maintain a set distance between the laser nozzle and the surface of the sheet. This distance is calculated by a sensor connected to the CNC, so that the CNC offsets the sensor variations on the distance programmed with additional movements in the axis programmed for the gap.



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

BLANK PAGE

ABOUT THE PRODUCT - CNC 8065

BASIC CHARACTERISTICS.

Basic characteristics.	8065 M		8065 M Power	
	Basic	Pack 1	Basic	Pack 1
Number of execution channels.	1	1	1	1 to 4
Number of axes.	3 to 6	5 to 8	5 to 12	8 to 28
Number of spindles.	1	1 to 2	1 to 4	1 to 4
Maximum number of axes and spindles.	7	10	16	32
Number of tool magazines.	1	1	1 to 2	1 to 4
Limited to 4 interpolated axes.	Option	Option	Option	Option

Basic characteristics.	8065 T		8065 T Power	
	Basic	Pack 1	Basic	Pack 1
Number of execution channels.	1	1 to 2	1 to 2	1 to 4
Number of axes.	3 to 5	5 to 7	5 to 12	8 to 28
Number of spindles.	2	2	3 to 4	3 to 4
Maximum number of axes and spindles.	7	9	16	32
Number of tool magazines.	1	1 to 2	1 to 2	1 to 4
Limited to 4 interpolated axes.	Option	Option	Option	Option

Basic characteristics.	8065 M	8065 M Power	8065 T	8065 T Power
Number of handwheels.	1 to 12			
Type of servo system.	Analog / Digital Sercos / Digital Mechatrolink			
Communications.	RS485 / RS422 / RS232 Ethernet			
Integrated PLC.				
PLC execution time.	< 1ms/K			
Digital inputs / Digital outputs.	1024 / 1024			
Marks / Registers.	8192 / 1024			
Timers / Counters.	512 / 256			
Symbols.	Unlimited			
Block processing time.	< 1 ms			



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

Remote modules.	RIOW	RIO5	RIO70	RIOR	RCS-S
Valid for CNC.	8070	8070	8070	8070	8070
	8065	8065	8065	8065	8065
	8060	8060	---	8060	8060
Communication with the remote modules.	CANopen	CANopen	CANfagor	CANopen	Sercos
Digital inputs per module.	8	24 / 48	16	48	---
Digital outputs per module.	8	16 / 32	16	32	---
Analog inputs per module.	4	4	8	---	---
Analog outputs per module.	4	4	4	---	4
Inputs for PT100 temperature sensors.	2	2	---	---	---
Feedback inputs.	---	---	4 (*)	---	4 (**)

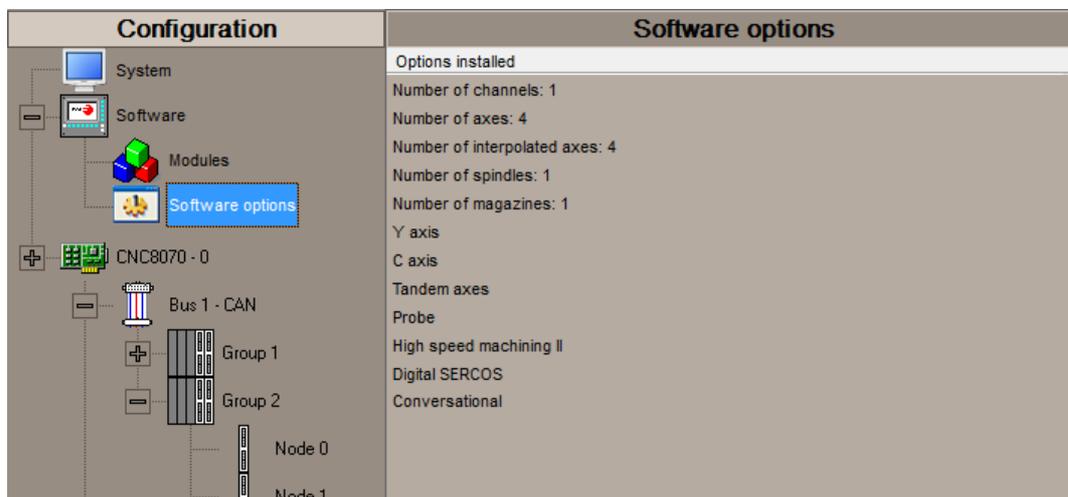
(*) Differential TTL / Sinusoidal 1 Vpp (**) TTL / Differential TTL / Sinusoidal 1 Vpp / SSI protocol / FeeDat / EnDat

Customizing (on an open system only).

PC-based open system, fully customizable.
 INI configuration files.
 Tool for display configuration FGUIM.
 Visual Basic®, Visual C++®, etc.
 Internal databases in Microsoft® Access.
 OPC compatible interface

SOFTWARE OPTIONS.

Some of the features described in this manual are dependent on the acquired software options. The active software options for the CNC can be consulted in the diagnostics mode (accessible from the task window by pressing [CTRL] [A]), under software options.



Consult the ordering handbook for information on the software options available for your model.

SOFT ADDIT AXES

Additional shaft.

Add axes to the default configuration.

SOFT ADDIT SPINDLES

Additional spindle.

Add spindles to the default configuration.

SOFT ADDIT TOOL MAGAZ

Additional tool magazine.

Add tool magazines to the default configuration.

SOFT ADDIT CHANNELS

Additional channel.

Add channels to the default configuration.

SOFT 4 AXES INTERPOLATION LIMIT

Limited to 4 interpolated axes.

It limits the number of axes to 4, where the CNC can also interpolate these at the same time.

SOFT OPEN SYSTEM

Open system.

The CNC is a closed system that offers all the features needed to machine parts. Nevertheless, at times there are some customers who use third-party applications to take measurements, perform statistics or other tasks apart from machining a part.

This feature must be active when installing this type of application, even if they are Office files. Once the application has been installed, it is recommended to close the CNC in order to prevent the operators from installing other kinds of applications that could slow the system down and affect the machining operations.

SOFT DIGITAL SERCOS

Sercos digital bus.

Sercos digital bus.

SOFT EDIT/SIMUL

EDISIMU mode (editing and simulation).

It allows for the editing, modification and simulation of a part-program.

SOFT DUAL-PURPOSE (M-T)

Dual-purpose machine.

A dual-purpose machine allows for both milling and turning cycles. On lathes with a Y axis, it allows making pockets, bosses and even irregular pockets with islands during milling cycles. Turning cycles can be used by milling machines which have a rotary axis that functions as a C axis.

SOFT IEC 61131 LANGUAGE

IEC 61131 language

IEC 61131 is a PLC programming language that is very popular in alternative markets, which is slowly entering into the machine-tool market. With this feature, the PLC may be programmed either in the usual Fagor language or in IEC 61131 format.

This feature requires the MP-PLUS (83700201) processor.

SOFT TOOL RADIUS COMP

Compensación de radio.

Tool compensation allows programming the contour to be machined based on part dimensions of the and without taking into account the dimensions of the tool that will be used later on. This avoids having to calculate and define the tool path based on the tool radius.

SOFT PROFILE EDITOR

Profile editor.

Allows for the part profiles to be edited graphically and to import dxf files.

FAGOR 

FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

SOFT IIP CONVERSATIONAL**Interactive Icon-based Pages (conversational mode).**

IIP or conversational mode is specifically designed for people without any prior programming knowledge or for those that are not familiarized with Fagor CNC's.

Working in conversational mode is easier than in ISO mode, as it ensures proper data entry and minimizes the number of operations to be defined. There is no need to work with part-programs.

SOFT HD GRAPHICS**HD graphics.**

High definition solid 3D graphics for the execution and simulation of part-programs and canned cycles of the editor.

During machining, the HD graphics display, in real time, the tool removing the material from the part, allowing the condition of the part to be seen at all times. HD graphics can display up to 4 views of the part, where each can be rotated, zoomed in or zoomed out. Measurements can also be made on the part and even sections on the piece from any angle.

In a multi-channel system, this feature requires the MP-PLUS (83700201) processor.

SOFT RTCP**Dynamic RTCP (Rotating Tool Center Point).**

The dynamic RTCP option is required for interpolation machining with 4, 5 or 6 axis.

This feature requires the MP-PLUS (83700201) processor.

SOFT C AXIS**C axis.**

It activates the kinematics for working with the C axis and the associated canned cycles. The CNC can control several C axes. The parameters of each axis indicate if it will function as a C axis or not, where it will not be necessary to activate another axis for the machine parameters.

SOFT Y AXIS**Y axis for lathe.**

It activates the kinematics for working with the Y axis and the associated canned cycles.

SOFT TANDEM AXES**Tandem axes.**

A tandem axis consists in two motors mechanically coupled (slaved) and making up a single transmission system (axis or spindle). A tandem axis helps provide the necessary torque to move an axis when a single motor is not capable of supplying enough torque to do it.

When activating this feature, it should be kept in mind that for each tandem axis of the machine, another axis must be added to the entire configuration. For example, on a large 3-axis lathe (X Z and tailstock), if the tailstock is a tandem axis, the final purchase order for the machine must indicate 4 axes.

SOFT SYNCHRONISM**Synchronization of axes and spindles.**

The axes and ballscrews may be synchronized in two ways: in terms of speed or position. The CNC configuration takes into consideration the synchronization of 2 axes or 2 spindles. Once synchronized, only the master displays and programs the element.

SOFT KINEMATIC CALIBRATION**Kinematics calibration.**

This work mode may be used to calibrate kinematics for the first time and to re-calibrate it every so often to correct any possible deviations originated by the day-to-day machine work.

SOFT HSSA II MACHINING SYSTEM**HSSA-II machining system.**

This is the new version of algorithms for high speed machining (HSC). This new HSSA algorithm allows for high speed machining optimization, where higher cutting speeds, smoother contours, a better surface finishing and greater precision are achieved.

SOFT TANGENTIAL CONTROL**Tangential control.**

"Tangential Control" maintains a rotary axis always in the same orientation with respect to the programmed tool path. The machining path is defined on the axes of the active plane and the CNC maintains the orientation of the rotary axis along the entire tool path.

SOFT PROBE**Probing canned cycles.**

The CNC may have two probes; usually a tabletop probe to calibrate tools and a measuring probe to measure the part.

This option activates the functions G100, G103 and G104 (for probe movements) and probe canned cycles (which help to measure part surfaces and to calibrate tools).

SOFT CONV USER CYCLES**Conversational user cycles.**

Incorporation of user cycles in conversational mode.

SOFT 70 PROGTL3**ProGTL3 programming language**

Another language apart from ISO for the programming of profiles using a geometric language without the need to use external CAD systems. This language allows for program functions to define lines and circles which define the points of intersection on a profile, in addition to macros for the creation of solids defined by a flat profile and one or more section profiles.

SOFT PPTRANS**Part-program translator.**

The program translator may be used to convert programs written in other languages into Fagor ISO codes.

SOFT THIRD PARTY CANOPEN**Third-party CANopen.**

Enables the use of non-Fagor CANopen modules.

SOFT FVC UP TO 10m3**SOFT FVC MORE TO 10m3****Medium and large volumetric compensation.**

5-axis machines are generally used during the manufacturing of large parts. The accuracy of the parts is limited by the machine manufacturing tolerances and is effected by temperature variations during machining.

In sectors such as the aerospace industry, machining demands mean that classic compensation tools are becoming suboptimal. Volumetric compensation FVC comes in to complement the machine adjusting tools. When mapping the total work volume of the machine, the CNC knows the exact position of the tool at all times. After applying the required compensation, the resulting part is made with the desired precision and tolerance.

There are 2 choices, which depend on the size of the machine, being up to 10 m³ and over 10 m³.



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

DECLARATION OF CE CONFORMITY AND WARRANTY CONDITIONS

DECLARATION OF CONFORMITY

The declaration of conformity for the CNC is available in the downloads section of FAGOR'S corporate website. <http://www.fagorautomation.com>. (Type of file: Declaration of conformity).

WARRANTY TERMS

The warranty conditions for the CNC are available in the downloads section of FAGOR's corporate website. <http://www.fagorautomation.com>. (Type of file: General sales-warranty conditions).



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

BLANK PAGE

VERSION HISTORY - CNC 8060

Here is a list of the features added to each manual reference.

Ref. 1402

Software V01.00
First version.

Ref. 1505

Software V01.30
Geometrical entities.
Incremental programming with the "I" command.
Programming cycles (macros). Milling of a surface defined by a plane profile and one or more section profiles. <ul style="list-style-type: none"> • G8736. Define the cycle and its parameters. • G8737. Beginning of the section profiles. • G8738. Activate the cycle.
Programming cycles (macros). Spiral profile milling. <ul style="list-style-type: none"> • G8735. Define the cycle and its parameters. • G8734. Activate the cycle.

Ref. 1512

Software V01.40
Programming cycles (macros). G8736. <ul style="list-style-type: none"> • New value for parameter D0 (D0=1). Surface emptying (pocket). • New parameter D7. Execute automatic corner rounding to join blocks of the section profile. • New parameter D8. Treatment of the starting coordinate of the section profile (only for toric tool). • New parameter D9. Treatment of the final coordinate of the section profile (only for toric tool).
Programming cycles (macros). G8738. <ul style="list-style-type: none"> • New parameter J. Distance between the machining pass sequences.
Programming cycles (macros). Ruled surfaces. <ul style="list-style-type: none"> • G8726. Define the general parameters of the cycle and the starting point of the first profile. • G8727. Second profile. • G8728. Activate the cycle.

Ref. 1709

Software V01.60
Programming cycles (macros). G8736. <ul style="list-style-type: none"> • New value for parameter D0 (D0=3). Machine only the bottom of the pocket.
Programming cycles (macros). Invert the machining direction of the profile. <ul style="list-style-type: none"> • G8754. Start of profile. • G8753. End of the profile.
Programming cycles (macros). Circular polygon pocket defined by a profile. <ul style="list-style-type: none"> • G8777. Define the cycle and its parameters. • G8778. Activate the cycle. • G8701. Profile pocket. Start of the profile and internal island profiles. • G8702. Profile pocket. Profile section of the pocket and islands (optional).
Generic user subroutines (G8000-G8999).



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

BLANK PAGE

VERSION HISTORY - CNC 8065

Here is a list of the features added to each manual reference.

Ref. 1402

Software V05.01

First version.

Ref. 1505

Software V05.20

Geometrical entities.

Incremental programming with the "I" command.

Programming cycles (macros). Milling of a surface defined by a plane profile and one or more section profiles.

- G8736. Define the cycle and its parameters.
- G8737. Beginning of the section profiles.
- G8738. Activate the cycle.

Programming cycles (macros). Spiral profile milling.

- G8735. Define the cycle and its parameters.
- G8734. Activate the cycle.

Ref. 1512

Software V05.40

Programming cycles (macros). G8736.

- New value for parameter D0 (D0=1). Surface emptying (pocket).
- New parameter D7. Execute automatic corner rounding to join blocks of the section profile.
- New parameter D8. Treatment of the starting coordinate of the section profile (only for toric tool).
- New parameter D9. Treatment of the final coordinate of the section profile (only for toric tool).

Programming cycles (macros). G8738.

- New parameter J. Distance between the machining pass sequences.

Programming cycles (macros). Ruled surfaces.

- G8726. Define the general parameters of the cycle and the starting point of the first profile.
- G8727. Second profile.
- G8728. Activate the cycle.

Ref. 1709

Software V05.60

Programming cycles (macros). G8736.

- New value for parameter D0 (D0=3). Machine only the bottom of the pocket.

Programming cycles (macros). Invert the machining direction of the profile

- G8754. Start of profile.
- G8753. End of the profile.

Programming cycles (macros). Circular polygon pocket defined by a profile.

- G8777. Define the cycle and its parameters.
- G8778. Activate the cycle.
- G8701. Profile pocket. Start of the profile and internal island profiles.
- G8702. Profile pocket. Profile section of the pocket and islands (optional).

Generic user subroutines (G8000-G8999).



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

BLANK PAGE

SAFETY CONDITIONS

Read the following safety measures in order to prevent harming people or damage to this product and those products connected to it. Fagor Automation shall not be held responsible of any physical or material damage originated from not complying with these basic safety rules.



Before start-up, verify that the machine that integrates this CNC meets the 2006/42/EC Directive.

PRECAUTIONS BEFORE CLEANING THE UNIT

- Do not get into the inside of the unit.** Only personnel authorized by Fagor Automation may access the interior of this unit.
- Do not handle the connectors with the unit connected to AC power.** Before handling these connectors (I/O, feedback, etc.), make sure that the unit is not powered.

PRECAUTIONS DURING REPAIRS

In case of a malfunction or failure, disconnect it and call the technical service.

- Do not get into the inside of the unit.** Only personnel authorized by Fagor Automation may access the interior of this unit.
- Do not handle the connectors with the unit connected to AC power.** Before handling these connectors (I/O, feedback, etc.), make sure that the unit is not powered.

PRECAUTIONS AGAINST PERSONAL HARM

- Interconnection of modules.** Use the connection cables provided with the unit.
- Use proper cables.** To prevent risks, only use cables and Sercos fiber recommended for this unit.
To prevent a risk of electrical shock at the central unit, use the proper connector (supplied by Fagor); use a three-prong power cable (one of them being ground).
- Avoid electric shocks.** To prevent electrical shock and fire risk, do not apply electrical voltage out of the indicated range.
- Ground connection.** In order to avoid electrical discharges, connect the ground terminals of all the modules to the main ground terminal. Also, before connecting the inputs and outputs of this product, make sure that the ground connection has been done.
In order to avoid electrical shock, before turning the unit on verify that the ground connection is properly made.
- Do not work in humid environments.** In order to avoid electrical discharges, always work with a relative humidity (non-condensing).
- Do not work in explosive environments.** In order to avoid risks, harm or damages, do not work in explosive environments.

FAGOR 

FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

PRECAUTIONS AGAINST DAMAGE TO THE PRODUCT

Work 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 by the CNC when installed in other environments (residential, homes, etc.).
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 meets the European directives on electromagnetic compatibility. Nevertheless, it is recommended to keep it away from sources of electromagnetic disturbance such as: <ul style="list-style-type: none"> Powerful loads connected to the same mains as the unit. Nearby portable transmitters (radio-telephones, Ham radio transmitters). Nearby radio / TC transmitters. Nearby arc welding machines. Nearby high voltage lines.
Enclosures.	It is up to the manufacturer to guarantee that the enclosure where the unit has been installed meets all the relevant directives of the European Union.
Avoid disturbances coming from the machine.	The machine must have all the interference generating elements (relay coils, contactors, motors, etc.) uncoupled.
Use the proper power supply.	Use an external regulated 24 Vdc power supply for the keyboard, operator panel and the remote modules.
Connecting the power supply to ground.	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.	Use shielded cables connecting all their meshes to the corresponding pin.
Ambient conditions.	Maintain the CNC within the recommended temperature range, both when running and not running. See the corresponding chapter in the hardware manual.
Central unit enclosure.	To maintain the right ambient conditions in the enclosure of the central unit, it must meet the requirements indicated by Fagor. See the corresponding chapter in the hardware manual.
Power switch.	This switch must be easy to access and at a distance between 0.7 and 1.7 m (2.3 and 5.6 ft) off the floor.

SAFETY SYMBOLS

Symbols that may appear in the manual.



Danger or prohibition symbol.
This symbol indicates actions or operations that may hurt people or damage products.



Warning or caution symbol.
This symbol indicates situations that certain operations could cause and the suggested actions to prevent them.



Obligation symbol.
This symbol indicates actions and operations that must be carried out.



Information symbol.
This symbol indicates notes, warnings and advises.



Symbol for additional documentation.
This symbol indicates that there is another document with more detailed and specific information.

Symbols that the product may carry.



Ground symbol.

This symbol indicates that that point must be under voltage.



ESD components.

This symbol identifies the cards as ESD components (sensitive to electrostatic discharges).



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

BLANK PAGE

RETURNING CONDITIONS

Pack it in its original package along with its original packaging material. If you do not have the original packaging material, pack it as follows:

- 1 Get a cardboard box whose 3 inside dimensions are at least 15 cm (6 inches) larger than those of the unit itself. The cardboard being used to make the box must have a resistance of 170 Kg (375 lb.).
- 2 Attach a label to the device indicating the owner of the device along with contact information (address, telephone number, email, name of the person to contact, type of device, serial number, etc.). In case of malfunction also indicate symptom and a brief description of the problem.
- 3 Protect the unit wrapping it up with a roll of polyethylene or with similar material. When sending a central unit with monitor, protect especially the screen.
- 4 Pad the unit inside the cardboard box with polyurethane foam on all sides.
- 5 Seal the cardboard box with packaging tape or with industrial staples.



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

BLANK PAGE

CNC 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 unit and, consequently, possible malfunctions. 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. Never use air compressed at high pressure to clean the unit because it could cause the accumulation of electrostatic charges that could result in electrostatic shocks.

The plastics used on the front panel are resistant to grease and mineral oils, bases and bleach, dissolved detergents and alcohol. Avoid the action of solvents such as chlorine hydrocarbons, benzole, esters and ether which can damage the plastics used to make the unit's front panel.

PRECAUTIONS BEFORE CLEANING THE UNIT

Fagor Automation shall not be held responsible for any material or physical damage derived from the violation of these basic safety requirements.

- Do not handle the connectors with the unit supplied with power. Before handling these connectors (I/O, feedback, etc.), make sure that the unit is not powered.
- Do not get into the inside of the unit. Only personnel authorized by Fagor Automation may access the interior of this unit.



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

BLANK PAGE

INTRODUCTION.

1

This document describes the programming syntax, operation and usage of the ProGTL3 (Professional Geometric Technological Language 3) language. The PROGTL3 language consists in several G functions that make editing profiles quick and easy. This language has evolved from previous oriented programming languages because it offers the user the possibility to program ProGTL functions (for defining straight lines and circles that define the intersection points of the profile) and ISO functions (to define segments and arcs of the profile).



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

1.1 G functions to expand geometry features.

1.

INTRODUCTION.
G functions to expand geometry features.

.G-	Meaning.	
G801	Ray or segment.	3.1
G802	Clockwise circle or arc.	3.2
G803	Counterclockwise circle or arc.	3.2
G808	Arc tangent to previous path.	3.3
G809	Arc by two points and beginning at the previous end point.	3.4
G810	First point or circle to define a straight line.	2.1
G811	Second point or circle to define a straight line.	2.1
G813	Definition of a straight line that goes through a point or is tangent to a circle and forms a known angle with the abscissa axis.	2.2
G820	Define a circle or a point.	2.3
G821	Define a rounding or an arc. Define a chamfer.	2.5
G831	Ray or segment tangent to previous path.	3.5
G832	Circle or clockwise arc tangent to previous path.	3.6
G833	Circle or counterclockwise arc tangent to previous path.	3.6
G840	Linear or tangential exit and cancellation of tool radius compensation.	2.6
G841	Begin profile with left-hand tool radius compensation.	2.7
G842	Begin profile with right-hand tool radius compensation.	2.7
G850	Cancel rotational offset.	2.8
G851	Activate rotational offset.	2.9
G8000 .. G8999	Generic user subroutines.	7.1
G8701	Circular polygon pocket defined by a profile. Profile pocket. Start of the profile and internal island profiles.	6.5
G8702	Circular polygon pocket defined by a profile. Profile pocket. Profile section of the pocket and islands (optional).	6.5
G8726	Ruled surfaces. General parameters of the cycle and the starting point of the first profile.	6.1
G8727	Ruled surfaces. Second profile.	6.1
G8728	Ruled surfaces. Activate the cycle.	6.1
G8735	Spiral profile milling. Define the cycle and its parameters.	6.2
G8734	Spiral profile milling. Activate the cycle.	6.2
G8736	Milling of a surface defined by a plane profile and one or more section profiles. Define the cycle and its parameters.	6.3
G8737	Milling of a surface defined by a plane profile and one or more section profiles. Beginning of the section profiles.	6.3
G8738	Milling of a surface defined by a plane profile and one or more section profiles. Activate the cycle.	6.3
G8753	Invert the direction of a profile. End of the profile.	6.4
G8754	Invert the direction of a profile. Start of profile.	6.4

.G.	Meaning.	
G8777	Circular polygon pocket defined by a profile. Define the cycle and its parameters.	6.5
G8778	Circular polygon pocket defined by a profile. Activate the cycle.	6.5

1.

INTRODUCTION.

G functions to expand geometry features.



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

1.2 Summary of G function syntax.

G function syntax (ProGTL functions).

Function.	Parameters.	Description.
[G810]	[X Y] circle center or support point [I] radius	Circle center or first support point for the straight line.
G811	[X Y] circle center or support point [I] radius [K] Intersection number.	Circle center or second support point for the straight line.
G813	[X Y] circle center or support point [I] radius [J] angle	Straight line.
G820	[X Y] center [I] radius [K] Intersection number.	Circle with known center and radius.
	[X Y] point1 [X Y] point2 [I] radius	Circle by two points and known radius.
	[X Y] point1 [X Y] point2 [X Y] point3	Circle by three points
G821	[I] Rounding radius [J] Chamfer size	Define a rounding or an arc. Define a chamfer.
G840	[X Y] exit point [K] exit type [I] radius	Linear or tangential exit and cancellation of tool radius compensation.
G841 G842	[X Y] joining point [K] Type of joint [I] radius	Begin profile with left-hand or right-hand tool radius compensation.
G850	Cancel G851.	Cancel rotational offset.
G851	[X Y] zero offset/rotation point [J] angle [K] Type of rotation/offset	Activate rotational offset.

G function syntax (ISO functions).

Function.	Parameters.	Description.
G801	[X Y] end point [J] angle	Ray or segment.
G802 G803	[X Y] end point [I J] center [I1=] radius	Circle or arc.
G808	[X Y] end point	Arc tangent to previous path.
G809	[X Y] end point [I J] intermediate point	Arc by two points and beginning at the previous end point.
G831	[X Y] end point [J] angle	Ray or segment tangent to previous path.
G832 G833	[X Y] end point [I J] center [I1=] radius	Circle or arc tangent to previous path.

1.

INTRODUCTION.
Summary of G function syntax.

The functions of the PROGTL3 language (G810, G811, G813, G820, G821, G840, G841, G842) allow programming tool paths by defining the end point of each segment as an intersection of two segments. These segments as opposed to the segments and arcs of the ISO language, are straight lines and circles. This way, with these functions the profile is defined as the result of the intersections between the straight lines and the circles that make it up.

·G·	Meaning.
G810	First point or circle to define a straight line.
G811	Second point or circle to define a straight line.
G813	Definition of a straight line that goes through a point or is tangent to a circle and forms a known angle with the abscissa axis.
G820	Define a circle or a point.
G821	Define a rounding or an arc. Define a chamfer.
G840	Linear or tangential exit and cancellation of tool radius compensation.
G841	Begin profile with left-hand tool radius compensation.
G842	Begin profile with right-hand tool radius compensation.
G850	Cancel rotational offset.
G851	Activate rotational offset.

The characteristics of these functions are the following.

- The coordinates can be programmed in inches, millimeters or with a scaling factor.
- The data that is left out may be recovered from the last programmed data.
- In the circles, the radius is defined with I (negative for clockwise direction and positive for counterclockwise).
- Programming incremental coordinates (G91) affects the abscissa, ordinate, radius and angle.

2.1 G810-G811. Straight line through two points or tangent to two circles.

Defining a straight line that goes through two points always consists in two program blocks that indicate the two support elements.

- A straight line going through two points.
- A straight line that goes through one point and is tangent to a circle.
- A straight line that is tangent to a circle and goes through a point.
- A straight line tangent to two circles.

The programming format is:

- G810 [X...] [Y...] [Z...] [A...][I...]
First circle or support point.
- G811 [X...] [Y...] [Z...] [A...] ... [I...] [K...]
Second circle or support point.

Firs circle or support point.

G810 First support point or first support circle tangent to the straight line.

Optional; if left out, it will use the point (or circle) of the previous block.

Coordinates of the circle center or of the support point of the straight line.

[X...] [Y...] Cartesian coordinates in G17.
[Z...] [X...] Cartesian coordinates in G18.
[Y...] [Z...] Cartesian coordinates in G19.
[Q...] [R...] Polar coordinates.

Optional. The coordinates are programmed with the name of the axes and the values.

The point can also be indicated in incremental coordinates with G91. When programming one of the two axes of the plane and leaving the other one out, it will use the coordinate programmed last for the axis that is not programmed.

This data may be left out completely if it is identical to those of the last circle programmed in the same profile. In this case, where the radius I will not be programmed, either, if a circle has been defined in the previous block, the circle will be used as support circle for the straight line.

Coordinate axes that do not belong to the reference plane.

[Z...] [A...] Coordinate axes that do not belong to the reference plane.

In this block, the programmed coordinates are useful, in this case Z indicates the total depth to be reached at the end of the block.

Circle radius.

[I...] Circle radius. Optional.

Positive value for counterclockwise and negative for clockwise. If a G91 was programmed earlier, the radius value will be incremental. If the I value is written, the circle is calculated in the current plane; XY plane if in G17, ZX if in G18 and YZ if in G19.

If parameter I is left out, a point will be defined, not a circle. The I value defined in previous blocks will be ignored.

If the point is left out and only G810 I is used, it will use the point of the block defined last as the center of the support circle with an I radius.

2.

PROGTL3 LANGUAGE:
G810-G811. Straight line through two points or tangent to two circles.

Second circle or support point.

G811 Second support point or second support circle tangent to the straight line.

Coordinates of the circle center or of the support point of the straight line.

[X...] [Y...] Cartesian coordinates in G17.
 [Z...] [X...] Cartesian coordinates in G18.
 [Y...] [Z...] Cartesian coordinates in G19.
 [Q...] [R...] Polar coordinates.

Optional. The coordinates are programmed with the name of the axes and the values.

The point can also be indicated in incremental coordinates with G91. When programming one of the two axes of the plane and leaving the other one out, it will use the coordinate programmed last for the axis that is not programmed.

This data may be left out completely if it is identical to those of the last circle programmed in the same profile. In this case, where the radius I will not be programmed, either, if a circle has been defined in the previous block, the circle will be used as support circle for the straight line.

Coordinate axes that do not belong to the reference plane.

[Z...] [A...] Coordinate axes that do not belong to the reference plane.

In this block, the programmed coordinates are useful, in this case Z indicates the total depth to be reached at the end of the block.

Circle radius.

[I...] Circle radius. Optional.

Positive value for counterclockwise and negative for clockwise. If a G91 was programmed earlier, the radius value will be incremental. If the I value is written, the circle is calculated in the current plane; XY plane if in G17, ZX if in G18 and YZ if in G19.

If parameter I is left out, a point will be defined, not a circle. The I value defined in previous blocks will be ignored.

If the point is left out and only G811 I is used, it will use the point of the block defined last as the center of the support circle with an I radius.

Intersection number.

[K...] Choice between the two intersections of the straight line with a circle programmed in the previous block. Optional.

K1 first intersection between the straight line and the circle moving in the direction of the straight line.

K2 second intersection.

If K is not programmed and there are more than one solution, the intersection selected automatically will depend on the previous element.

- If the previous element is a circle, the selected intersection will be the first one in the direction of the straight line.
- If the previous element is a rounding between a circle and the straight line and the straight line being programmed, the selected intersection will depend on the direction of the straight line, on the circle or on the rounding between the straight line and on the circle.

2.

PROGTL3 LANGUAGE:

G810-G811. Straight line through two points or tangent to two circles.

FAGOR 

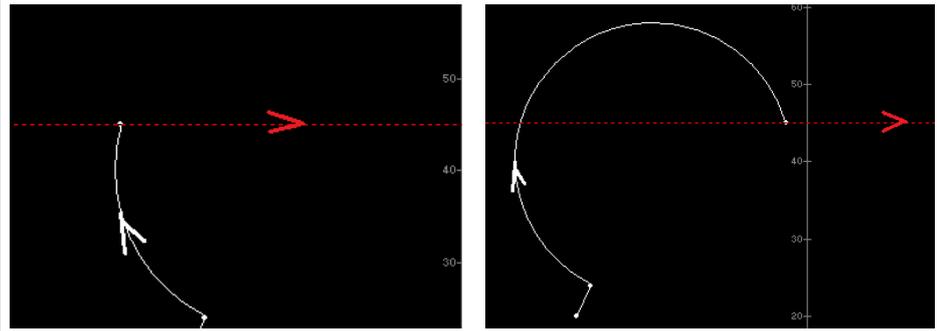
FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

Examples.

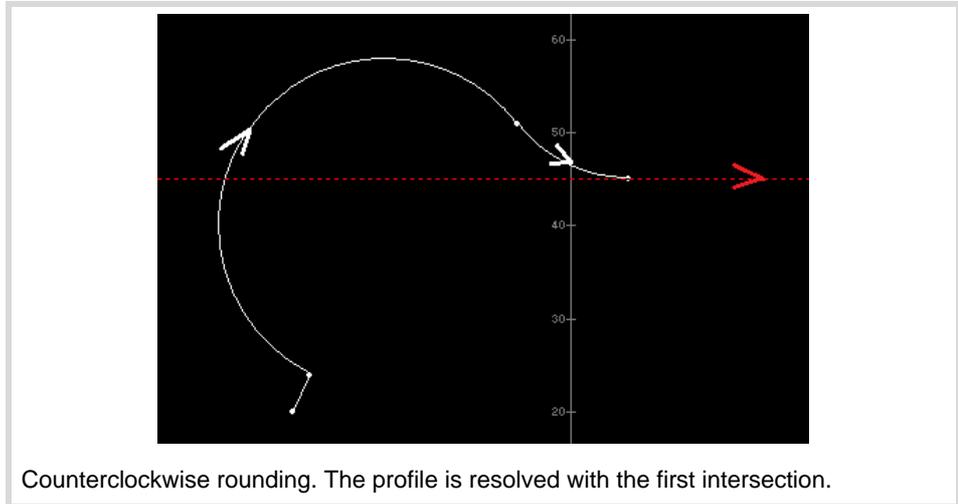
Intersection of straight line with previous circle, there are two possible solutions, if K is not indicated, it will be equal to K1.



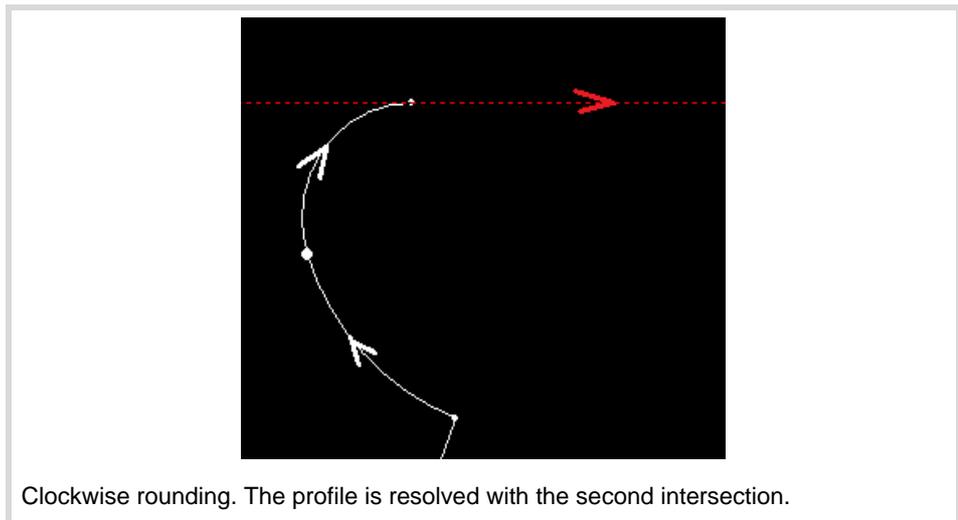
Intersection of a straight line with a clockwise circle if K=1.

Intersection of a straight line with a clockwise circle if K=2.

Intersection of a straight line with a previous circle and rounding. There only one possible solution following the orientation of the profile.



Counterclockwise rounding. The profile is resolved with the first intersection.



Clockwise rounding. The profile is resolved with the second intersection.

2.

PROGTL3 LANGUAGE.

G810-G811. Straight line through two points or tangent to two circles.



CNC 8060
CNC 8065

(REF: 1709)

2.2 G813. Definition of a straight line that goes through a point or is tangent to a circle and forms a known angle with the abscissa axis.

A straight line is defined as follows:

- G813 [X...] [Y...] [Z...] [A...][I...] J... [K...]

Definition of a straight line that goes through a point or is tangent to a circle and forms a known angle with the abscissa axis.

Coordinates of the circle center or of the support point of the straight line.

[X...] [Y...] Cartesian coordinates in G17.

[Z...] [X...] Cartesian coordinates in G18.

[Y...] [Z...] Cartesian coordinates in G19.

[Q...] [R...] Polar coordinates.

Optional. The coordinates are programmed with the name of the axes and the values.

The point can also be indicated in incremental coordinates with G91. When programming one of the two axes of the plane and leaving the other one out, it will use the coordinate programmed last for the axis that is not programmed.

This data may be left out completely if it is identical to those of the last circle programmed in the same profile. In this case, where the radius I will not be programmed, either, if a circle has been defined in the previous block, the circle will be used as support circle for the straight line.

Coordinate axes that do not belong to the reference plane.

[Z...] [A...] Coordinate axes that do not belong to the reference plane.

In this type of block, the names of all the axes of the machine may be used. In this case, one must indicate the final depth of the axes to be reached at the end of the block.

Circle radius.

[I...] Circle radius. Optional.

Positive value for counterclockwise and negative for clockwise. If a G91 was programmed earlier, the radius value will be incremental. If the I value is written, the circle is calculated in the current plane; XY plane if in G17, ZX if in G18 and YZ if in G19.

If parameter I is left out, a point will be defined, not a circle. The I value defined in previous blocks will be ignored.

If the point is left out and only G813 I is used, it will use the point of the block defined last as the center of the support circle with an I radius.

Angle of the straight line.

J... Angle, in degrees, between the straight line and the abscissa axis.

The angle value will be incremental with respect to the programmed previous angle if a G91 has been programmed earlier.

Intersection number.

[K...] Choice between the two intersections of the straight line with a circle programmed in the previous block. Optional.

K1 first intersection between the straight line and the circle moving in the direction of the straight line.

K2 second intersection.

If K is not programmed and there are more than one solution, the intersection selected automatically will depend on the previous element.

- If the previous element is a circle, the selected intersection will be the first one in the direction of the straight line.
- If the previous element is a rounding between a circle and the straight line and the straight line being programmed, the selected intersection will depend on the direction of the straight line, on the circle or on the rounding between the straight line and on the circle.

2.

PROGTL3 LANGUAGE.

G813. Definition of a straight line that goes through a point or is tangent to a circle and forms a known angle with the abscissa axis.

FAGOR 

FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

2.3 G820. Define a circle or a point.

A circle or a point may be defined as follows.

- G820 [X...] [Y...] [I...] [Z...] [A...]... [K...]
Circle with known center and radius.
- G820 [X...] [Y...], [X...] [Y...] I... [K...]
Circle by two points and known radius.
- G820 [X...] [Y...], [X...] [Y...], [X...] [Y...] [K...]
Circle by three points

Coordinates of the circle center/point or of the support point of the straight line.

- [X...] [Y...] Cartesian coordinates in G17.
- [Z...] [X...] Cartesian coordinates in G18.
- [Y...] [Z...] Cartesian coordinates in G19.
- [Q...] [R...] Polar coordinates.

Optional. The coordinates are programmed with the name of the axes and the values.

For circles given by two points and a known radius or circles going through three points, the coordinates must always be in absolute Cartesian coordinates. In the rest of the cases, the point can also be indicated in incremental coordinates with G91.

When programming one of the two axes of the plane and leaving the other one out, it will use the coordinate programmed last for the axis that is not programmed. This data may be left out completely if it is identical to those of the last circle programmed in the same profile.

Coordinate axes that do not belong to the reference plane.

- [Z...] [A...] Coordinate axes that do not belong to the reference plane.

In this type of block, the names of all the axes of the machine may be used. In this case, one must indicate the final depth of the axes to be reached at the end of the block.

Circle radius.

- [I...] Circle radius. Optional.

Positive value for counterclockwise and negative for clockwise. If a G91 was programmed earlier, the radius value will be incremental. If the I value is written, the circle is calculated in the current plane; XY plane if in G17, ZX if in G18 and YZ if in G19.

If parameter I is left out, a point will be defined, not a circle. The I value defined in previous blocks will be ignored.

If the point is left out and only G820 I is used, it will use the point of the block defined last as the center of the support circle with an I radius.

Intersection number.

- [K...] Selection of the intersection with an element programmed in the previous block. Optional.

K1 first intersection between the straight line and the circle moving in the direction of the straight line.

K2 second intersection.

If the previous geometric element is a straight line:

K1 chooses the first intersection in the direction of the previous straight line.

K2 chooses the second intersection in the direction of the previous straight line.

If the previous geometric element is a circle:

K1 chooses the intersection to the left of the straight line that joins the center of the previous circle with the center of that circle.

K2 chooses the intersection to the right of the straight line that joins the center of the previous circle with the center of that circle.

If not programmed, by default it assumes K=1.

2.

PROGTL3 LANGUAGE:
G820. Define a circle or a point.

2.4 G821. Define a chamfer.

A chamfer is defined as follows.

- G821 J...
J... Size of the chamfer (distance to the vertex).

2.5 G821. Define a rounding or an arc.

A rounding is defined as follows.

- G821 I...
I... Rounding or arc radius.

Positive rounding or arc value if counterclockwise and negative if clockwise.

2.

PROGTL3 LANGUAGE:
G821. Define a chamfer.

FAGOR 

FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

2.6 G840. Linear or tangential exit and cancellation of tool radius compensation.

This function is programmed as follows:

- G840 [X...] [Y...] [Z...] [A...][K...] [I...]

Linear or tangential exit and cancellation of tool radius compensation.

Function G840.

G840 Exit from profile.

Profile exit point.

[X...] [Y...] [Z...] [A...] Profile exit point. Optional.

Choosing the type of exit from the profile.

[K...] Choosing between two types of exits from profile and canceling tool radius compensations.

K1 automatic exit from the profile with a straight line perpendicular to the last element programmed.

K2 automatic exit from the profile with a semi-circle tangent to the last element programmed.

Leaving K out provides a programmed (not automatic) exit from the profile.

- If K is not programmed, the profile ends in a segment and point, the end of the profile will be perpendicular to the segment at that point. If the profile ends with two segments, the end point of the profile is the intersection point or tangency point between the first and second elements before the G840, tool radius offset.
- If K is not programmed, the last element programmed will not be part of the tool path, but it will only be used to define the intersection point or tangency point with the before-last segment defined. The end of the profile will be on the before-last segment declared.

Radius for tangential exit.

[I...] Radius for tangential exit.

2.

PROGTL3 LANGUAGE.

G840. Linear or tangential exit and cancellation of tool radius compensation.



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

2.7 G841/G842. Begin profile with left-hand or right-hand tool radius compensation.

The beginning of the profile with tool radius compensation may be defined in the following ways.

- G841 [X...] [Y...] [Z...] [A...] ... [K...] [I...]
Begin profile with left-hand tool radius compensation. Function G841 indicates that there is tool radius compensation on the left of the profile (as viewed in the direction of the tool path).
- G842 [X...] [Y...] [Z...] [A...] ... [K...] [I...]
Begin profile with right-hand tool radius compensation. Function G842 indicates that there is tool radius compensation on the right of the profile (as viewed in the direction of the tool path).

Profile entry point.

[X...] [Y...] [Z...] [A...] Profile entry point. Optional.

The point may also be programmed in incremental coordinates with G91.

Choosing the type of entry to the profile.

[K...] Choosing between two types of entries to the profile.

K1 automatic entry to the profile with a straight line perpendicular to the first element programmed.

K2 automatic entry to the profile with a semi-circle tangent to the first element programmed.

Leaving K out provides a programmed (not automatic) entry from the profile.

- If K is not programmed, the profile begins in a segment and a point, the beginning of the profile will be perpendicular to the segment at that point. If the profile begins with two segments, the starting point of the profile is the intersection point or tangency point between the first and second elements after functions G841 and G842, offset to the tool radius.
- If K is not programmed, the first element programmed will not be part of the tool path, but it will only be used to define the intersection point or tangency point with the before-last segment defined. The beginning of the profile will be on the second segment declared.

Radius for tangential entry.

[I...] Radius for tangential entry. Optional.

The radius can also be programmed in incremental coordinates with G91, respect to the radius programmed earlier.

2.

PROGTL3 LANGUAGE:

G841/G842. Begin profile with left-hand or right-hand tool radius compensation.

FAGOR 

FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

2.8 G850. Cancel rotational offset.

Function G850 cancels the secondary zero offset and returns to the main part zero

2.9 G851. Activate rotational offset.

Rotation and offset are set as follows.

- G851 [X...] [Y...] [J...] [K...]

Activate rotational offset.

With G851, it is possible to set an unlimited number of secondary of zero offsets or rotate them with respect to part zero or with respect to the last secondary offset programmed. The whole program can also be rotated with respect to any point.

Rotation and offset.

[X...] [Y...]	Coordinates of the new zero offset with respect to the active zero offset or point around which the program is to be rotated.
[J...]	Angle between the X axis and the new zero offset and the X axis of the active zero offset, or program rotation angle.
[K]	K1: Rotational offset around the part zero (default option). K2: Rotational offset around the last secondary zero offset programmed. K3: Rotation around the programmed point.

2.

PROGTL3 LANGUAGE:
G850. Cancel rotational offset.



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

ISO functions (G801, G802, G803, G808, G809, G831, G832 and G833) may be used to define a profile through the segments and arcs that make it up in a similar way as functions G01, G02, G03, G08, G09 but with more options. Tool paths are programmed using straight or curved segments that start from the end point of the previous segment and finish at the point programmed in that segment. This group of functions offer the user the possibility to combine segments or arcs with straight lines and circles.

·G·	Meaning.
G801	Ray or segment.
G802	Clockwise circle or arc.
G803	Counterclockwise circle or arc.
G808	Arc tangent to previous path.
G809	Arc by two points and beginning at the previous end point.
G831	Ray or segment tangent to previous path.
G832	Circle or clockwise arc tangent to previous path.
G833	Circle or counterclockwise arc tangent to previous path.

The characteristics of these functions are the following.

- The coordinates can be programmed in inches, millimeters or with a scaling factor.
- The data that is left out is recovered from the previous data only for G801, G802, G803. G808 and G809 require defining the points. For G831, G832 and G833 the data that is left will be calculated according to the programmed data.
- In arcs, the radius is defined with I1= and the direction with G802, G832 (clockwise) or G803, G833 (counterclockwise).
- By default, the center of the arcs will be in incremental coordinates referred to the end point.

3.1 G801. Define a ray or segment.

A ray or segment may be defined with G801 as follows.

- G801 [X...] [Y...] J [K...]
Ray that ends at a point and at a given angle.
- G801 J
Ray with starting point at the previous segment and at a given angle.
- G801 [X...] [Y...]
Segment that ends at a point(same as G1) or end point of previous ray.
- G801
End point of the ray previously defined with G811 or G813.

Coordinates of the end point of the ray (modes 1 and 3).

- [X...] [Y...] Cartesian coordinates in G17.
- [Z...] [X...] Cartesian coordinates in G18.
- [Y...] [Z...] Cartesian coordinates in G19.
- [Q...] [R...] Polar coordinates.

The end point of the segment may be programmed in either absolute or incremental (Cartesian or Polar) coordinates. If a G91 has been programmed before the G801, the coordinates of the point will be considered incremental with respect to the last programmed point.

Defining the segment correctly requires indicating the coordinates of the end point and the angle of the ray; if one of the coordinates is left out, it uses the last coordinate programmed earlier.

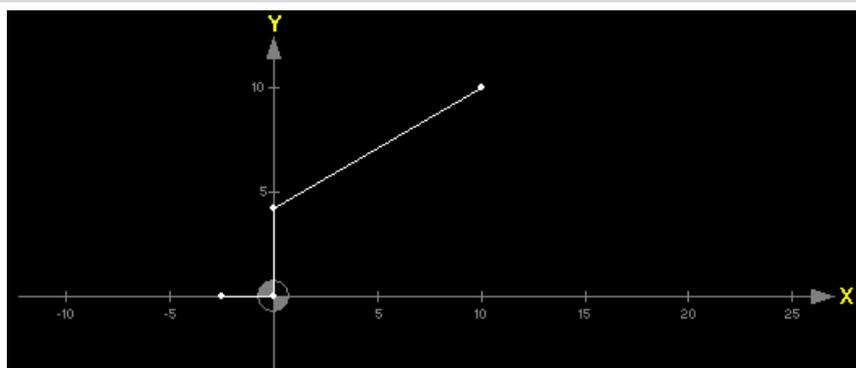
Angle.

- J... Angle of the ray (with G91, incremental with respect to the previous programmed angle).

Intersection number.

- [K...] Intersection number with the previous element.

Example mode 1: G801 [X...] [Y...] J [K...]



```
G0 X-2.5 Y0
G841 K1
G813 X0 J90
G801 X10 Y10 J30
```

3.

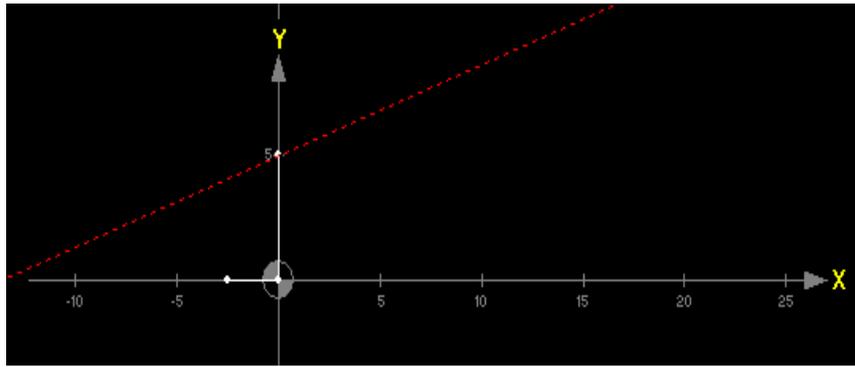
ISO FUNCTIONS.
G801. Define a ray or segment.

FAGOR 
FAGOR AUTOMATION

CNC 8060
CNC 8065

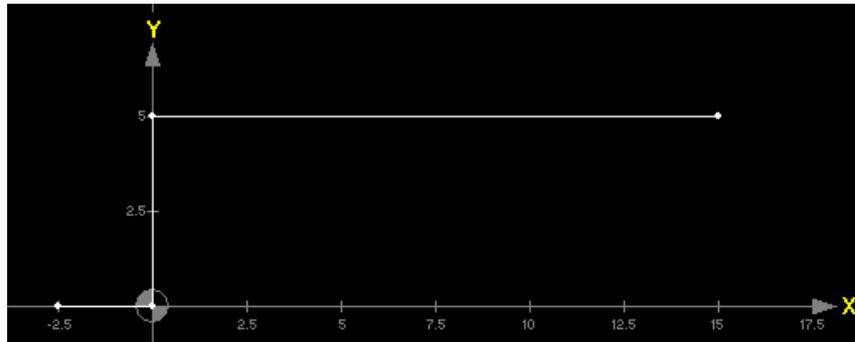
(REF: 1709)

Example mode 2: G801 J



```
G0 X-2.5 Y0  
G841 K1  
G813 X0 J90  
G801 X0 Y5  
G801 J20
```

Example mode 3: G801 [X...] [Y...]



```
G0 X-2.5 Y0  
G841 K1  
G813 X0 J90  
G801 X0 Y5  
G801 X15 Y5
```

3.

ISO FUNCTIONS.
G801. Define a ray or segment.

3.2 G802/G803. Define a circle of an arc.

A circle or an arc may be defined as follows.

- G802/G803 [X...] [Y...] [I...] [J...] [K...]
Clockwise/counterclockwise circle or arc that begins at the end point of the previous element or at the intersection with the previous element and ends at the programmed point and with the given center.
- G802/G803 [X...] [Y...] I1=...
Clockwise/counterclockwise circle or arc that begins at the end point of the previous element and ends at the programmed point and with the given center.
- G802/G803 [X...] [Y...]
Arc that ends at a point. When programming both coordinates, it must be a valid point; if only one is programmed, the other one is already calculated.
- G802/G803 [I...] [J...]
Circle with starting point at the previous end point and with the given center.

Coordinates of the end point of the circle or arc (modes 1, 2 and 3).

[X...] [Y...] Cartesian coordinates in G17.

[Z...] [X...] Cartesian coordinates in G18.

[Y...] [Z...] Cartesian coordinates in G19.

[Q...] [R...] Polar coordinates.

Center of the circle (modes 1 and 4).

[I...] [J...] Coordinates of the center of the circle or of the arc.

The end point of the arc and the center may be programmed in either absolute or incremental (Cartesian or Polar) coordinates. If one of the coordinates is left out, it uses the last coordinate programmed earlier. At least one of the coordinates must be programmed. If a G91 has been programmed before the G802, the coordinates of the point will be considered incremental with respect to the previous programmed point.

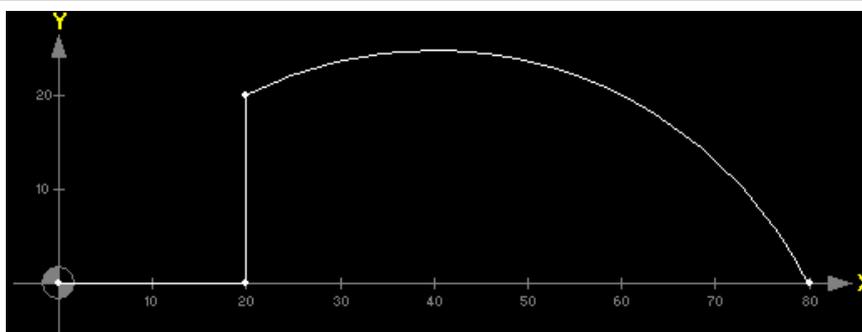
By default, the arc center I J will be in incremental referred to the starting point of the arc and it will be necessary to program a G06 in the block to be considered absolute or a G262.

Radius of the circle (mode 2).

I1=... Radius of the circle (with G91, incremental with respect to the previous programmed radius).

[K...] Intersection number with the previous element.

Example mode 1: G802/G803 [X...] [Y...] [I...] [J...] [K...]



```
G0 X0 Y0
G841 K1
G813 X20 J90
G802 G06 X80 Y0 I40 J-20 K2
```

3.

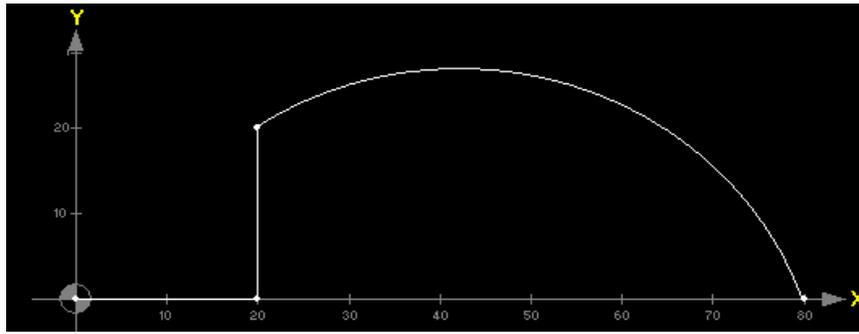
ISO FUNCTIONS.
G802/G803. Define a circle of an arc.

FAGOR 
FAGOR AUTOMATION

CNC 8060
CNC 8065

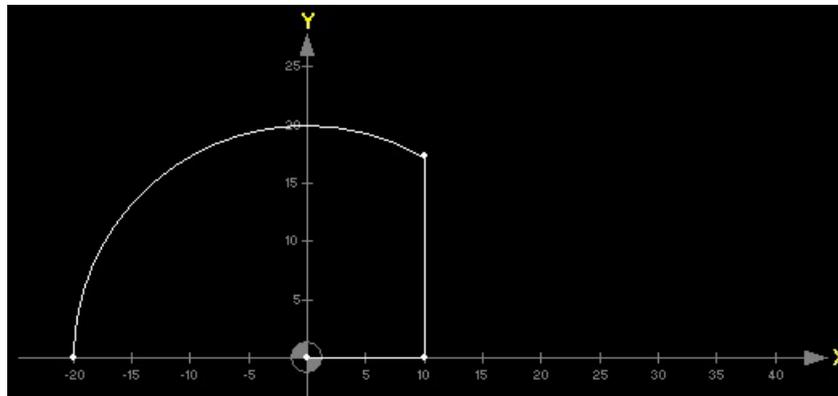
(REF: 1709)

Example mode 2: G802/G803 [X...] [Y...] I1=... [K...]



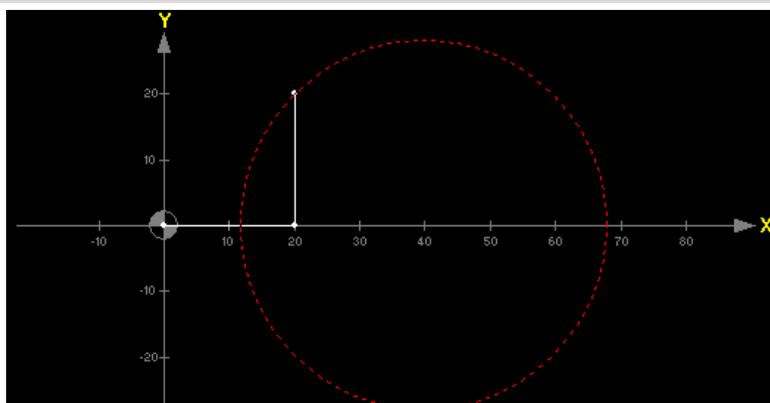
```
G0 X0 Y0  
G841 K1  
G813 X20 J90  
G801 Y20  
G802 X80 Y0 I1=40
```

Example mode 3: G802/G803 [X...] [Y...]



```
G0 X0 Y0  
G841 K1  
G813 X10 J90  
G820 X0 Y0 I20  
G803 X-20 Y0
```

Example mode 4: G802/G803 [I...] [J...]



```
G0 X0 Y0  
G841 K1  
G813 X20 J90  
G801 Y20  
G802 G06 I40 J0
```

3.

ISO FUNCTIONS.
G802/G803. Define a circle of an arc.

FAGOR 

FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

3.3 G808. Arc tangent to previous path.

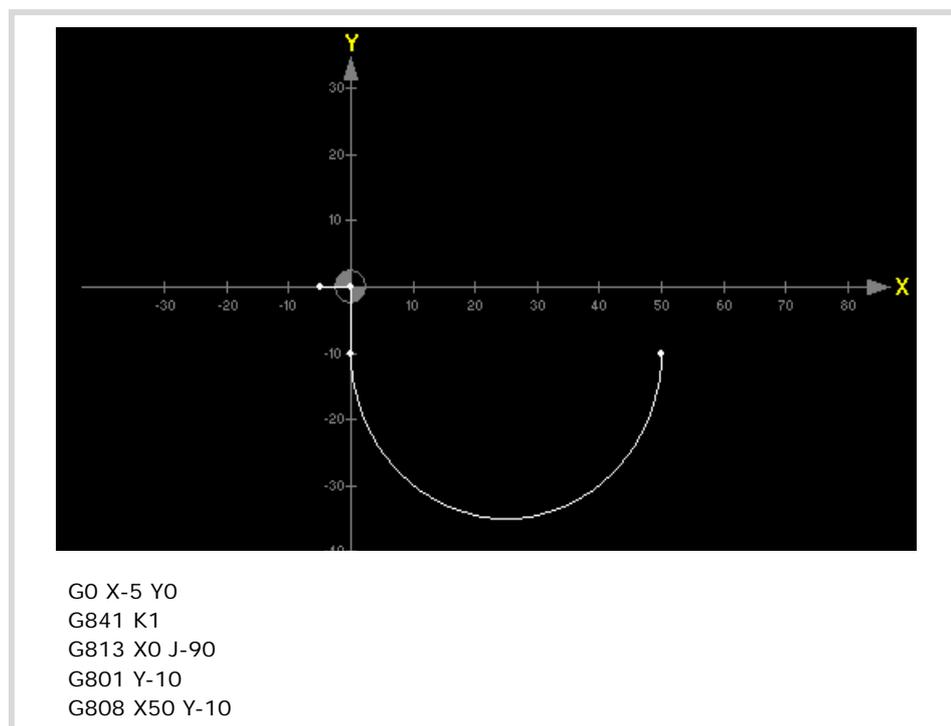
An arc tangent to the previous path is defined with the end point of the arc.

- G808 X... Y...

Coordinates of the final point of the circle or of the arc.

- [X...] [Y...] Cartesian coordinates in G17.
- [Z...] [X...] Cartesian coordinates in G18.
- [Y...] [Z...] Cartesian coordinates in G19.
- [Q...] [R...] Polar coordinates.

Both coordinates are required. If a G91 has been programmed before the G808, the coordinates of the point will be considered incremental with respect to the previous programmed point.



3.

ISO FUNCTIONS.
G808. Arc tangent to previous path.

3.4 G809. Arc going through three points.

An arc that begins at the end point of the previous element and goes through the programmed intermediate point and end point is defined as follows.

- G809 X... Y... I... J...

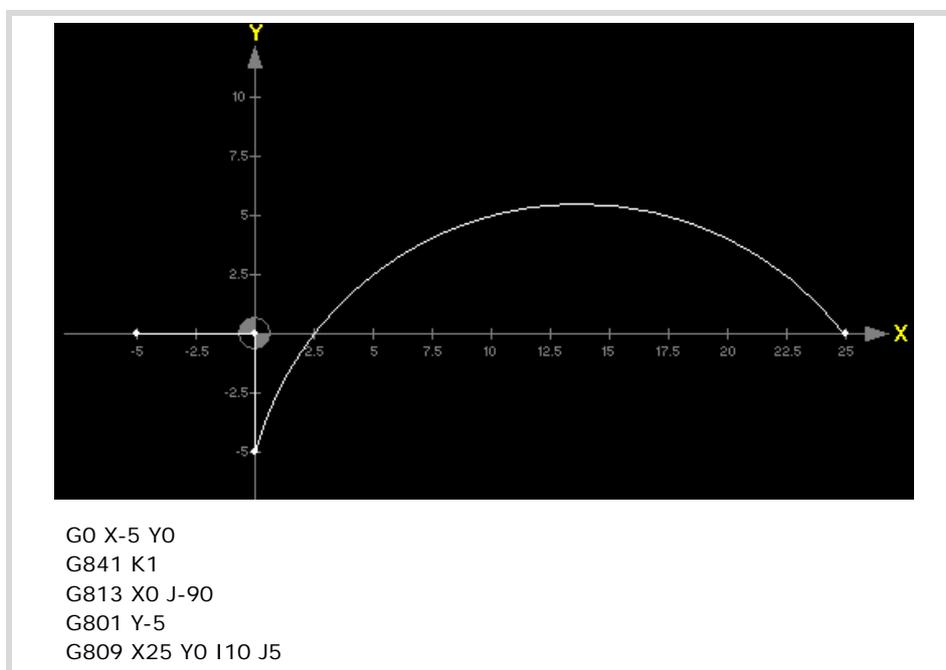
Coordinates of the end point of the arc

- [X...] [Y...] Cartesian coordinates in G17.
- [Z...] [X...] Cartesian coordinates in G18.
- [Y...] [Z...] Cartesian coordinates in G19.
- [Q...] [R...] Polar coordinates.

Intermediate point of the arc (mode 1).

- I... J... Coordinates of an intermediate point of the arc.

Both coordinates of each point must be programmed. If a G91 has been programmed before the G809, the coordinates of the end point and intermediate point will be considered incremental with respect to the previous programmed point.



3.

ISO FUNCTIONS.

G809. Arc going through three points.

FAGOR 

FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

3.5 G831. Define a ray or segment tangent to the previous element.

A ray or segment tangent to the previous element is done as follows.

- G831 [X...] [Y...] [J...] [K...]

Ray tangent to the previous element that ends at the programmed point and has an angle J.

Coordinates of the end point of the ray.

[X...] [Y...] Cartesian coordinates in G17.

[Z...] [X...] Cartesian coordinates in G18.

[Y...] [Z...] Cartesian coordinates in G19.

[Q...] [R...] Polar coordinates.

The end point of the segment may also be programmed in incremental coordinates with G91. In the case of tangent elements such as G831, G832 and G833, the missing data will be calculated considering the tangency point with the previous element and the rest of the data provided by the user.

Angle of the ray.

[J...] Angle of the ray.

The angle value will be incremental if a G91 has been programmed earlier.

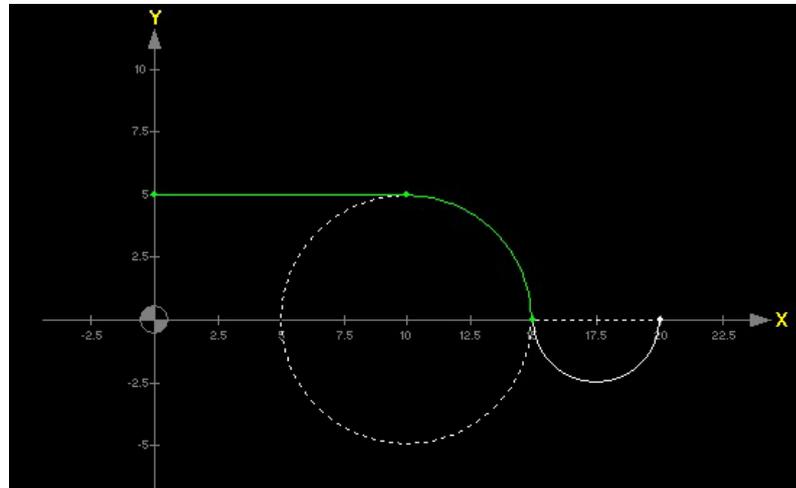
Tangency solution number.

[K...] Chosen solution number. Tangency point with previous element

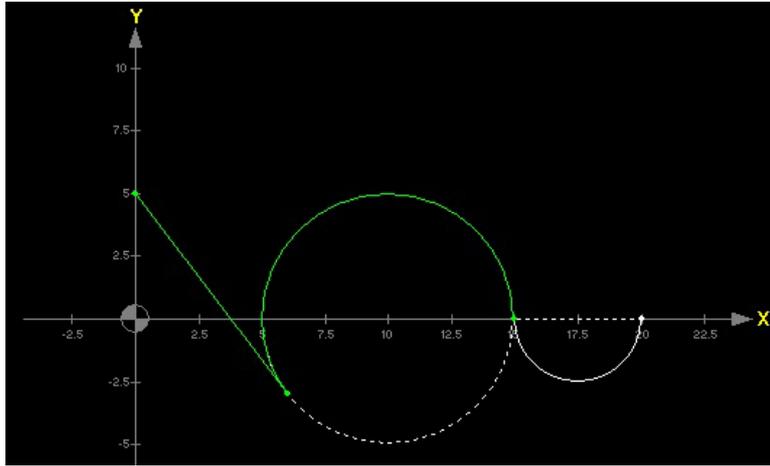
When there are more than one solution, the different solutions will be displayed graphically.

Example G831 [X...] [Y...] [K...]

Known data; end point and tangency with previous element.



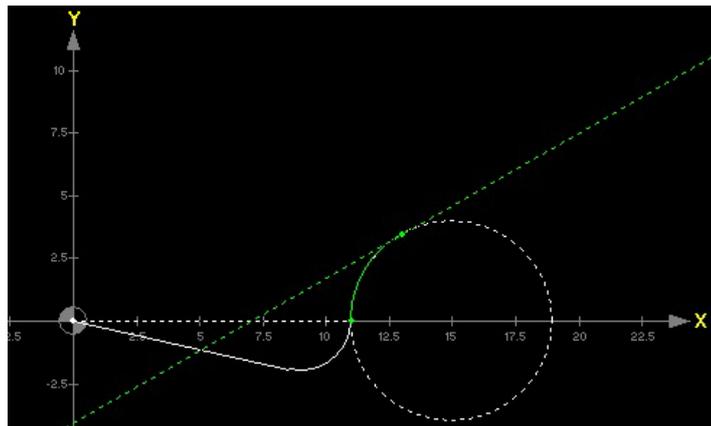
```
G0 X20 Y0
G842 K2
G820 X10 I5
G831 X0 Y5 K1
```



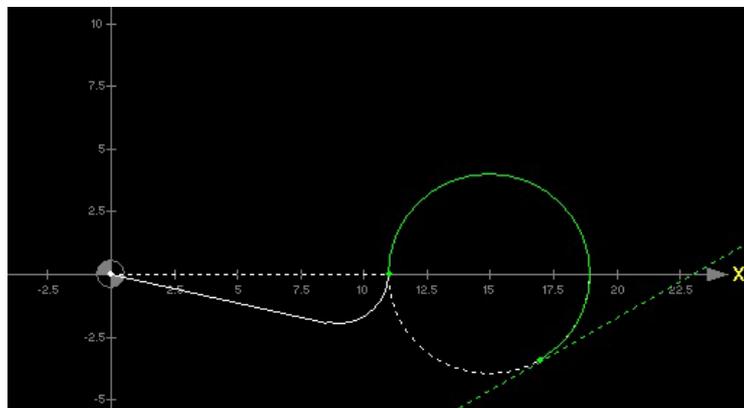
```
G0 X20 Y0
G842 K2
G820 X10 I5
G831 X0 Y5 K2
```

Example G831 [J...] [K...]

Known data; angle of the straight line and tangency point with the previous element.



```
G0 X0 Y0
G841 K2 I2
G820 X15 Y0 I-4
G831 J30 K1
```



```
G0 X0 Y0
G841 K2 I2
G820 X15 Y0 I-4
G831 J30 K2
```

3.

ISO FUNCTIONS.
G831 . Define a ray or segment tangent to the previous element.



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

3.6 G832/G833. Define a circle or arc tangent to the previous element.

A circle or arc tangent to the previous element is defined as follows.

- G832/G833 [X...] [Y...] [I...] [J...] [I1=...] [K...]

Coordinates of the end point of the arc

- [X...] [Y...] Cartesian coordinates in G17.
- [Z...] [X...] Cartesian coordinates in G18.
- [Y...] [Z...] Cartesian coordinates in G19.
- [Q...] [R...] Polar coordinates.

If a G91 has been programmed earlier, the coordinates of the point will be considered incremental with respect to the ones programmed earlier. The missing data will be calculated considering the tangency point with the previous element and the rest of the data provided by the user.

Incremental coordinates of the center of the circle.

- [I...] [J...] Circle center coordinates.

By default, the center coordinates will be incremental; programming them as absolute requires a G06 in the block or a G262.

Tangency solution number.

- [K...] Chosen solution number. Tangency point with previous element.

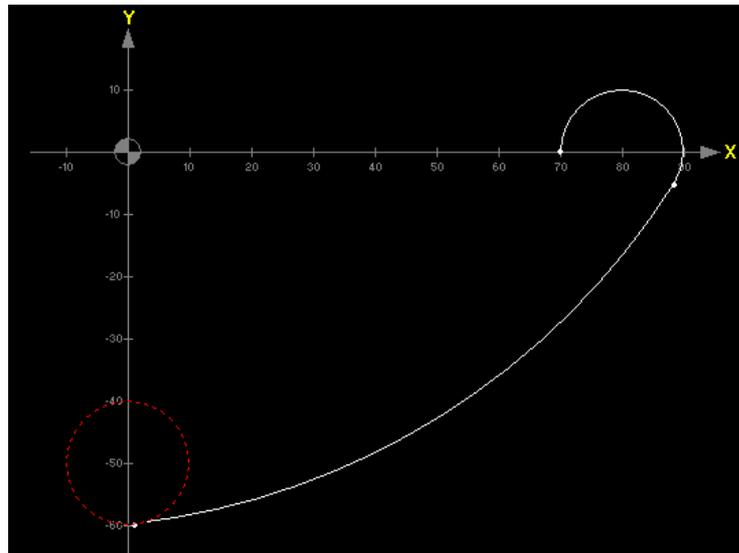
When there are more than one solution, the different solutions will be displayed graphically.

Circle radius.

- [I1=...] Circle radius. Optional.

Positive value for counterclockwise and negative for clockwise. If a G91 was programmed earlier, the radius value will be incremental.

Examples G832 I1= / G832 I J I1=



```
X70 Y0
G842 K2
G820 X80 Y0 I-10
G832 I1=120 K1
G832 G06 I0 J-50 I1=10
```

3.

ISO FUNCTIONS. G832/G833. Define a circle or arc tangent to the previous element.

FAGOR 
FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

INCREMENTAL PROGRAMMING WITH THE "I" COMMAND.

4

The "I" command may be added to the programmed coordinate and it may be used to make it incremental. This command is only valid on the axis and the block where it is programmed. This way, it is possible to program absolute and incremental movements in the same block without having to use the G90/G91 functions.

The "I" command may be added to the following data:

- Programmed point (abscissa and ordinate). [X] [Y]
- Programmed support point (abscissa and ordinate). [X] [Y]
- Angle. [J]
- Radius. [I]
- Offset point and rotation angle. [X] [Y] [J]

Example:

X27 Y-13 Z-4I

X and Y axis movement in absolute coordinates.

Incremental Z axis movement with respect to the previous position.

G809 X100I Y100, X150 Y50

The X coordinate of the first point of the circle is programmed in incremental coordinates (X100I) and the second point in absolute coordinates (X150).

G820 X100 Y-50 I-50I

The radius of the circle is programmed in incremental coordinates (I-50I).

Considerations about the coordinate history.

The CNC always memorizes the previous coordinate in absolute coordinates, therefore, it is irrelevant whether the coordinate has been programmed in absolute or incremental coordinates.

If a point is programmed in Cartesian coordinates and the previous one was in Polar coordinates, the CNC converts the previous point into Cartesian coordinates and increments the position value (coordinate). If a point is programmed in Polar coordinates and the previous one was in Cartesian coordinates, the CNC converts the previous point into Polar coordinates and increments the position value (coordinate).

4.1 G801. Ray or segment.

[X] [Y] programmed point.

Each programmed coordinate may be incremental with respect to the last programmed coordinate. The CNC always memorizes the previous coordinate in absolute coordinates, therefore, it is irrelevant whether the coordinate has been programmed in absolute or incremental coordinates.

If a point is programmed in Cartesian coordinates and the previous one was in Polar coordinates, the CNC converts the previous point into Cartesian coordinates and increments the position value (coordinate). If a point is programmed in Polar coordinates and the previous one was in Cartesian coordinates, the CNC converts the previous point into Polar coordinates and increments the position value (coordinate).

[J] Angle.

The programmed angle may be incremental with respect to the last programmed angle.



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

4.2 G802/G803. Clockwise/counterclockwise circle or arc.

[X] [Y] programmed point.

Each programmed coordinate may be incremental with respect to the last programmed coordinate. The CNC always memorizes the previous coordinate in absolute coordinates, therefore, it is irrelevant whether the coordinate has been programmed in absolute or incremental coordinates.

If a point is programmed in Cartesian coordinates and the previous one was in Polar coordinates, the CNC converts the previous point into Cartesian coordinates and increments the position value (coordinate). If a point is programmed in Polar coordinates and the previous one was in Cartesian coordinates, the CNC converts the previous point into Polar coordinates and increments the position value (coordinate).

[I J] Center.

By default, the arc center will be in incremental coordinates. When programming G06 or G262 to have it in absolute coordinates, it is possible to use the option of programming each coordinate of the center in incremental coordinates.

[I1=] Radius.

The radius can only be programmed in absolute coordinates.

4.3 G808. Arc tangent to previous path.

[X] [Y] programmed point.

Each programmed coordinate may be incremental with respect to the last programmed coordinate. The CNC always memorizes the previous coordinate in absolute coordinates, therefore, it is irrelevant whether the coordinate has been programmed in absolute or incremental coordinates.

If a point is programmed in Cartesian coordinates and the previous one was in Polar coordinates, the CNC converts the previous point into Cartesian coordinates and increments the position value (coordinate). If a point is programmed in Polar coordinates and the previous one was in Cartesian coordinates, the CNC converts the previous point into Polar coordinates and increments the position value (coordinate).

4.4 G809. Arc by two points and beginning at the previous end point.

[X] [Y] programmed point.

Each programmed coordinate may be incremental with respect to the last programmed coordinate. The CNC always memorizes the previous coordinate in absolute coordinates, therefore, it is irrelevant whether the coordinate has been programmed in absolute or incremental coordinates.

If a point is programmed in Cartesian coordinates and the previous one was in Polar coordinates, the CNC converts the previous point into Cartesian coordinates and increments the position value (coordinate). If a point is programmed in Polar coordinates and the previous one was in Cartesian coordinates, the CNC converts the previous point into Polar coordinates and increments the position value (coordinate).

[I][J] Intermediate point of the programmed arc.

Each programmed coordinate may be incremental with respect to the last programmed coordinate.

4.

INCREMENTAL PROGRAMMING WITH THE "I" COMMAND.
G802/G803. Clockwise/counterclockwise circle or arc.

4.5 G810. First point or circle to define a straight line.

[X] [Y] Programmed support point.

Each programmed coordinate may be incremental with respect to the last programmed support point. The CNC always memorizes the previous support point in absolute coordinates, therefore, it is irrelevant whether it has been programmed in absolute or incremental coordinates.

If a point is programmed in Cartesian coordinates and the previous one was in Polar coordinates, the CNC converts the previous point into Cartesian coordinates and increments the position value (coordinate). If a point is programmed in Polar coordinates and the previous one was in Cartesian coordinates, the CNC converts the previous point into Polar coordinates and increments the position value (coordinate).

[I] Radius.

The radius may be incremental with respect to the last programmed radius.

4.6 G811. Second point or circle to define a straight line.

[X] [Y] Programmed support point.

Each programmed coordinate may be incremental with respect to the last programmed support point. The CNC always memorizes the previous support point in absolute coordinates, therefore, it is irrelevant whether it has been programmed in absolute or incremental coordinates.

If a point is programmed in Cartesian coordinates and the previous one was in Polar coordinates, the CNC converts the previous point into Cartesian coordinates and increments the position value (coordinate). If a point is programmed in Polar coordinates and the previous one was in Cartesian coordinates, the CNC converts the previous point into Polar coordinates and increments the position value (coordinate).

[I] Radius.

The radius may be incremental with respect to the last programmed radius.

4.7 G813. Straight line that goes through a point or is tangent to a circle and forms a known angle with the abscissa axis.

[X] [Y] Programmed support point.

Each programmed coordinate may be incremental with respect to the last programmed support point. The CNC always memorizes the previous support point in absolute coordinates, therefore, it is irrelevant whether it has been programmed in absolute or incremental coordinates.

If a point is programmed in Cartesian coordinates and the previous one was in Polar coordinates, the CNC converts the previous point into Cartesian coordinates and increments the position value (coordinate). If a point is programmed in Polar coordinates and the previous one was in Cartesian coordinates, the CNC converts the previous point into Polar coordinates and increments the position value (coordinate).

[I] Radius.

The radius may be incremental with respect to the last programmed radius.

[J] Angle.

The programmed angle may be incremental with respect to the last programmed angle.

4.

INCREMENTAL PROGRAMMING WITH THE "I" COMMAND.

G810. First point or circle to define a straight line.

FAGOR 

FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

4.8 G820. Define a circle or a point.

[X][Y] Programmed support point (and also for the second and third point of the circle).

Each programmed coordinate may be incremental with respect to the last programmed support point. The CNC always memorizes the previous support point in absolute coordinates, therefore, it is irrelevant whether it has been programmed in absolute or incremental coordinates.

[I] Radius.

The radius may be incremental with respect to the last programmed radius.

4.9 G831. Define a rounding or an arc. Define a chamfer.

[X] [Y] programmed point.

Each programmed coordinate may be incremental with respect to the last programmed coordinate. The CNC always memorizes the previous coordinate in absolute coordinates, therefore, it is irrelevant whether the coordinate has been programmed in absolute or incremental coordinates.

If a point is programmed in Cartesian coordinates and the previous one was in Polar coordinates, the CNC converts the previous point into Cartesian coordinates and increments the position value (coordinate). If a point is programmed in Polar coordinates and the previous one was in Cartesian coordinates, the CNC converts the previous point into Polar coordinates and increments the position value (coordinate).

[J] Angle.

The programmed angle may be incremental with respect to the last programmed angle.

4.10 G832/G833. Clockwise/counterclockwise circle or arc tangent to the previous path.

[X] [Y] programmed point.

Each programmed coordinate may be incremental with respect to the last programmed coordinate. The CNC always memorizes the previous coordinate in absolute coordinates, therefore, it is irrelevant whether the coordinate has been programmed in absolute or incremental coordinates.

If a point is programmed in Cartesian coordinates and the previous one was in Polar coordinates, the CNC converts the previous point into Cartesian coordinates and increments the position value (coordinate). If a point is programmed in Polar coordinates and the previous one was in Cartesian coordinates, the CNC converts the previous point into Polar coordinates and increments the position value (coordinate).

[I J] Center.

By default, the arc center will be in incremental coordinates. When programming G06 or G262 to have it in absolute coordinates, it is possible to use the option of programming each coordinate of the center in incremental coordinates.

[I1=] Radius.

The radius can only be programmed in absolute coordinates.

4.

INCREMENTAL PROGRAMMING WITH THE "I" COMMAND.
G820. Define a circle or a point.

4.11 G851. Rototranslation.

[X] [Y] rotation origin point

Each coordinate of the origin point may be incremental with respect to the last programmed origin point.

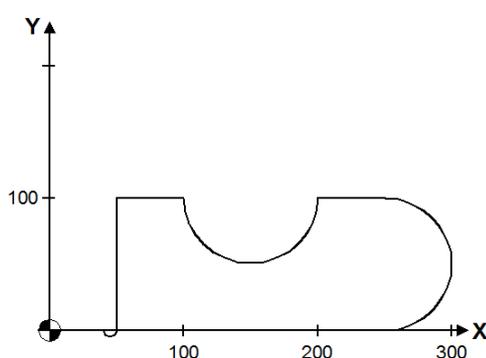
[J] Rotation angle.

The rotation angle may be incremental with respect to the last programmed angle.

4.12 Examples.

Point programmed in incremental coordinates.

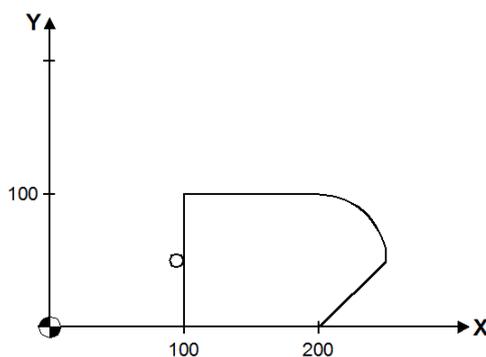
Point programmed in G801, G802, G803, G808, G809, G832, G833. The programmed point is incremental with respect to the last programmed end point.



```
X40 Y0
G841 K2
G813 X50 J90
G801 Y100I
G801 X50I
G809 X100I Y100, X150 Y50
G801 X50I
G808 X250 Y-100I
G801 X50 Y0
G840 X40 Y0 K1
```

Support point programmed in incremental coordinates.

Point programmed in G810, G811, G813, G820, G851. The programmed point is incremental with respect to the last programmed support point.



```
X90 Y50
G841 K2
G813 X100 J90
G813 X100I Y100 J0
G820 Y-50I I-50
G810 X250 Y50
G811 X-50I Y-50I
G813 J180
G813 X100 J90
G840 X90 Y50 K2
```

4.

INCREMENTAL PROGRAMMING WITH THE "I" COMMAND.
G851. Rototranslation.

FAGOR 

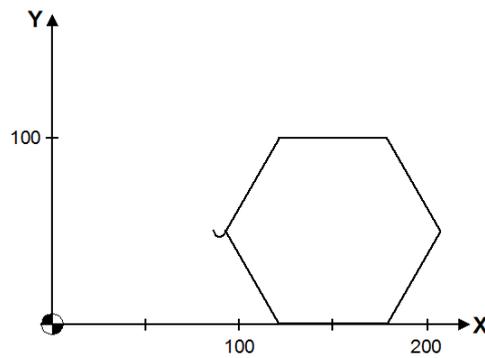
FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

Angle programmed in incremental coordinates.

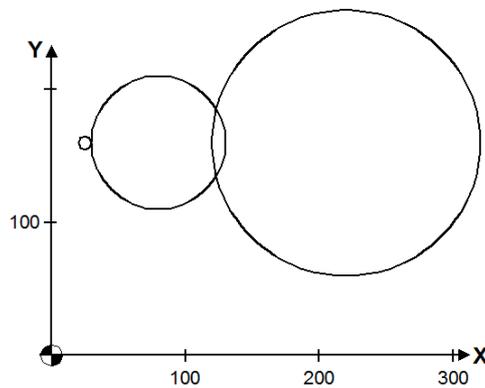
The programmed angle is incremental with respect to the last programmed angle.



```
X90 Y50
G841 K2
N1:
G813 X150 Y50 I-50 J60I
N2:
#RPT[N1,N2,6]
G840 X90 Y50 K2
```

Radius programmed in incremental coordinates.

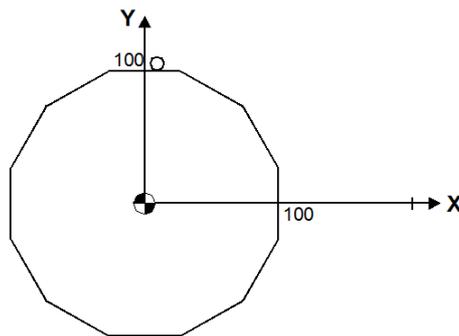
The programmed radius is incremental with respect to the last programmed radius.



```
X20 Y150
G841 K2
G820 X60I I-50
G820 X140I I-50I
G820 X-140I I50I
G840 X20 K2
```

Offset point and rotation angle.

The programmed offset point is incremental with respect to the last programmed offset point.
The rotation angle may be incremental with respect to the last programmed rotation angle.



```
X10 Y110
G841 K2
G813 Y100 J0
N1:
G851 J-30I
G813 Y100 J0
N2:
#RPT[N1,N2,11]
G850
G840 X10 Y110 K2
```

4.

INCREMENTAL PROGRAMMING WITH THE "I" COMMAND.
Examples.



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

There are 5 types of geometrical entities; point, circle, origin and distance. The definition of these geometrical entities may be saved and recalled later on in a profile or they may be used to define other geometrical entities.

Although the CNC saves the entities after the end of the program, it is recommended to define them in each program that will be used.

Defining geometrical entities.

The CNC lets save up to 30 geometrical entities. The geometrical entities are identified with the letter E followed by a number (E1 to E30) and may be defined directly (programming all the information required to define the entity) or indirectly (invoking other geometrical entities saved earlier). To define a geometrical entity, use the "," (comma) character to separate one entity (origin, point, circle, straight line) from another.

- Direct definition. E1 = G820 X50 Y12 I-12
 E2 = G851 X30 Y30 J45, G820 X50 Y33
- Indirect definition. E6 = G820 X10 Y10 I30, E13
 E12 = E5, E8

The geometrical entities may be defined anywhere in the program, before or after the definition of a profile, but at least one block before they are used.

Deleting geometrical entities.

The geometrical entities are deleted when turning the CNC off; they are not deleted after executing an M02 or an M30 or after a reset.

5.1 Saving the various types of geometrical entities.

5.1.1 Saving a point.

A point may be defined with coordinates or as an intersection between elements. The geometrical entities may be programmed in the block or may be called upon if they were defined earlier. The point may be defined as follows.

- Point in Cartesian or Polar coordinates.
- Point as intersection of two straight lines.
- Point as intersection of two circles.
- Point as intersection of a straight line and a circle.
- Point as center of a saved circle.

The straight line and the circle used to define the point may be defined either directly (using the proposed definitions) or indirectly (using geometrical entities saved earlier).

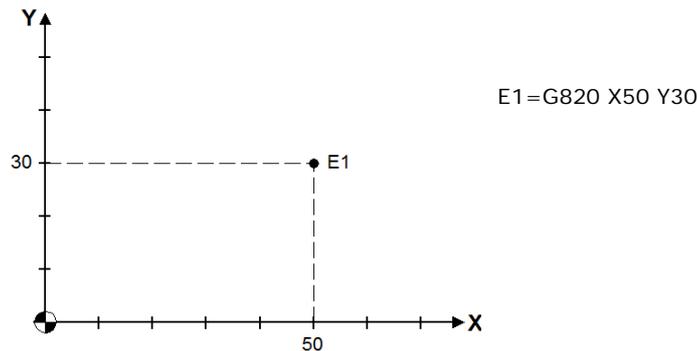
```
Ep = G813 X.. Y.. J.., G820 X.. Y.. I.. K2
Ep = G810 X.. Y.., G811 X.. Y.., G820 X.. Y.. I..
Ep = G810 X.. Y.., G811 Ep, Ec
Ep = Er, G820 X.. Y.. I..
Ep = Er, G820 Ep I..
Ep = Ec, Er K2
```

Point in Cartesian or Polar coordinates.

A point may be defined either in Cartesian or Polar coordinates. A point in Polar coordinates will be referred to the part zero.

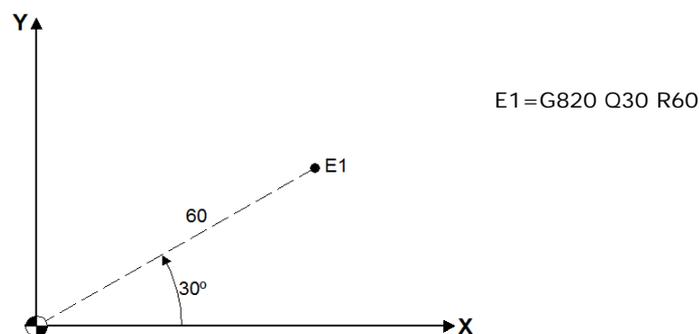
- The format is the following:

Ep = G820 X.. Y.. (Cartesian coordinates)



- The format is the following:

Ep = G820 Q.. R.. (Polar coordinates)



(REF: 1709)

5.

GEOMETRICAL ENTITIES.
Saving the various types of geometrical entities.

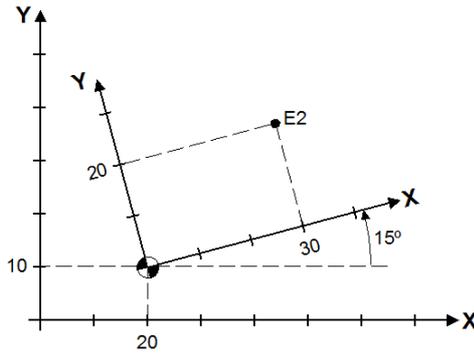
FAGOR 
FAGOR AUTOMATION

CNC 8060
CNC 8065

A point in Cartesian coordinates may be referred to an origin other than the part zero, in which case, the origin will be programmed directly or indirectly before defining the point.

The format is the following:

Ep = G851 X.. Y.. J.., G820 X.. Y..
 Ep = Ep, G820 X.. Y..



E2=G851 X20 Y10 J15, G820 X30 Y20

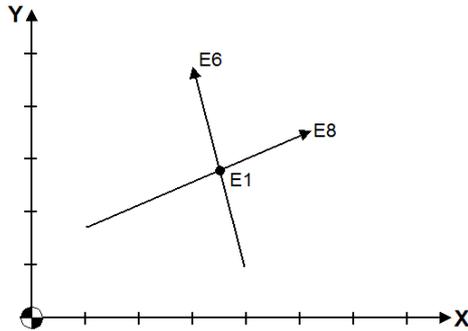
E1=G851 X20 Y10 J15
 E2=E1, G820 X30 Y20

Point as intersection of two straight lines.

The format is the following, the optional part is shown between brackets:

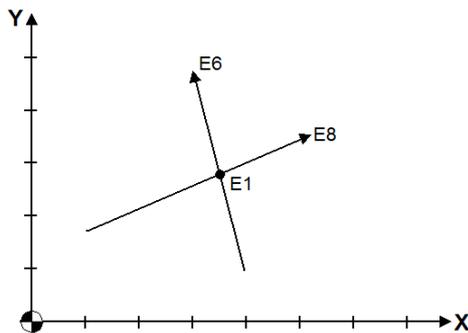
Ep = [origin,] straight1, straight2

Each straight line may be defined with G813 or with G810-G811.



E1=G813 X.. Y.. J.., G813 X.. Y.. J..

E6=G813 X.. Y.. J..
 E8=G813 X.. Y.. J..
 E1=E6, E8



E10=G810 X.. Y.. I.., G811 X.. Y.. I..
 G810 X.. Y.. I.., G811 X.. Y.. I..

E6=G810 X.. Y.. I.., G811 X.. Y.. I..
 E8=G810 X.. Y.. I.., G811 X.. Y.. I..
 E1=E6, E8

5.

GEOMETRICAL ENTITIES.
 Saving the various types of geometrical entities.



FAGOR AUTOMATION

CNC 8060
 CNC 8065

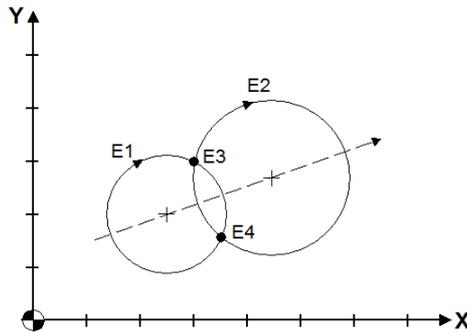
(REF: 1709)

Point as intersection of two circles.

The format is the following, the optional part is shown between brackets:

$$E_p = [\text{origin},] \text{circle1}, \text{circle2} [\text{K2}]$$

Parameter K2 selects the second intersection, the one to the right of the line joining the center of the first circle with the center of the second circle. If parameter K2 is not programmed, it takes the first intersection, the one to the left of the line joining the center of the first circle with the center of the second circle.



$$E3 = G820 X.. Y.. I.., G820 X.. Y.. I..$$

$$E4 = G820 X.. Y.. I.., G820 X.. Y.. I.. K2$$

$$E1 = G820 X.. Y.. I..$$

$$E2 = G820 X.. Y.. I..$$

$$E3 = E1, E2$$

$$E4 = E1, E2 K2$$

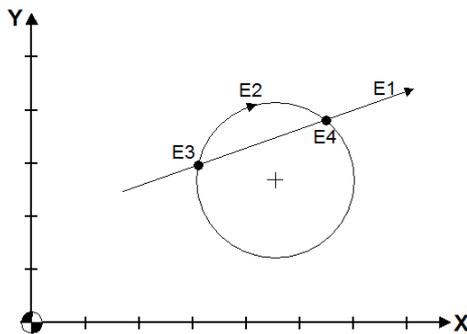
Point as intersection of a straight line and a circle.

The format is the following, the optional part is shown between brackets:

$$E_p = [\text{origin},] \text{straight}, \text{circle} [\text{K2}]$$

$$E_p = [\text{origin},] \text{circle}, \text{straight} [\text{K2}]$$

Parameter K2 selects the second intersection according to the direction of the straight line. If K2 is not programmed, it takes the first intersection.



$$E3 = G813 X.. Y.. J.., G820 X.. Y.. I..$$

$$E4 = G813 X.. Y.. J.., G820 X.. Y.. I.. K2$$

$$E1 = G813 X.. Y.. J..$$

$$E2 = G820 X.. Y.. I..$$

$$E3 = E1, E2$$

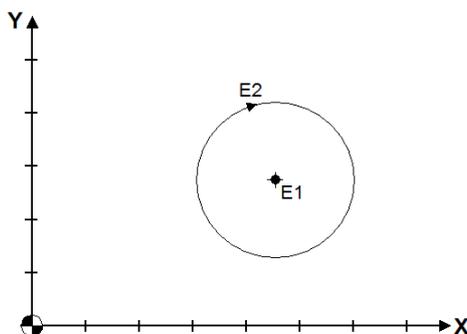
$$E4 = E1, E2 K2$$

Point as center of a saved circle.

The format is the following:

$$E_p = E_c I0$$

This option saves point E_p , the coordinates of the center of circle E_c .



$$E1 = E2 I0$$

5.

GEOMETRICAL ENTITIES.

Saving the various types of geometrical entities.



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

5.1.2 Saving a straight line.

A straight line may be defined directly or indirectly, recalling the geometrical elements saved earlier. A straight line may be defined as follows:

- Straight line that goes through a point and forms an angle with the X axis.
- Straight line that is tangent to a circle and that forms an angle with the X axis.
- Straight line going through two points.
- Straight line tangent to two circles.
- Straight line that goes through one point and is tangent to a circle.
- Straight line that is tangent to a circle and goes through a point.
- Straight line defined with two straight lines and a chamfer.
- Straight line parallel to a predefined straight line.
- Straight line traveled in the opposite direction.

The points and circles used to define the point may be defined either directly (using the proposed definitions) or indirectly (using geometrical entities saved earlier).

```
Er = G851 X.. Y.. J.., G813 X.. Y.. J..
Er = G813 Ec J..
Er = G810 X.. Y.., G811 X.. Y..
Er = G810 X.. Y.., G811 Ep
Er = G810 Ep, G811 Ec
Er = G810 X.. Y.. I.., G811 Ec
Er = G810 Ec, G811 Ec
```

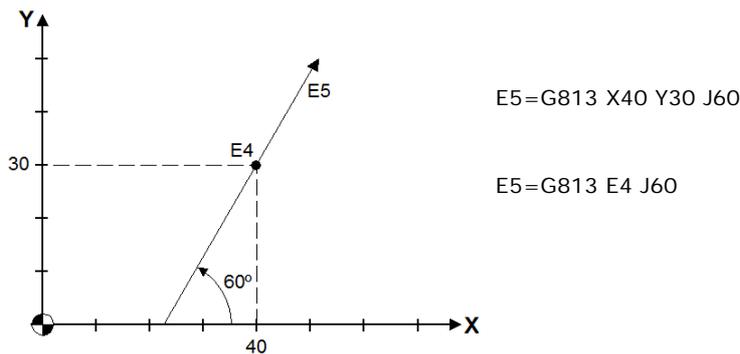
When defining straight lines tangent to a circle, when calling a predefined circle, it is possible to change the orientation with the "-" (minus) sign to ensure compatibility in the directions of the straight line and the circle and the tangency point of 0 degrees.

```
Example:
G810 X.. Y.., -Ec
```

Straight line that goes through a point and forms an angle with the X axis.

The straight line is defined by a point it goes through and the angle it forms with the positive X axis. The format is the following, the optional part is shown between brackets:

```
Er = [origin] G813 point J..
```



5.

GEOMETRICAL ENTITIES.

Saving the various types of geometrical entities.



FAGOR AUTOMATION

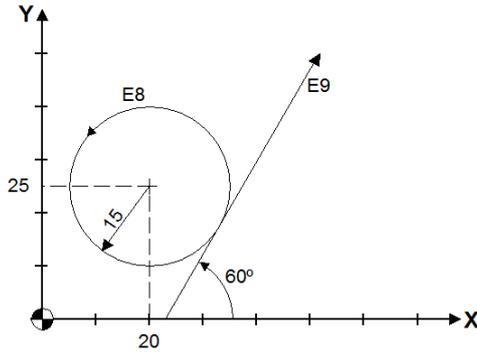
CNC 8060
CNC 8065

(REF: 1709)

Straight line that is tangent to a circle and that forms an angle with the X axis.

The straight line is defined by a circle it is tangent to and the angle it forms with the positive X axis. The format is the following, the optional part is shown between brackets:

Er = [origin] G813 circle J..



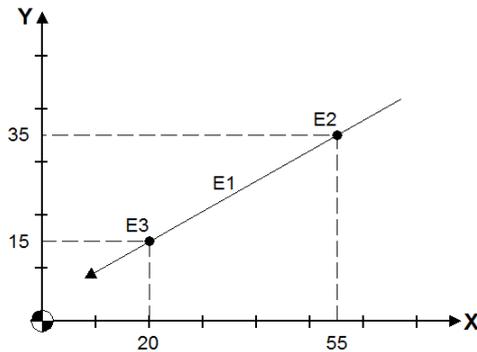
E9=G813 X20 Y25 I15 J60

E9=G813 E8 J60

Straight line going through two points.

The straight line is defined by two points that it goes through. The format is the following, the optional part is shown between brackets:

Er = [origin,] G810 point1, G811 point2



E1=G810 X55 Y35, G811 X20 Y15

E1=G810 E2, G811 E3

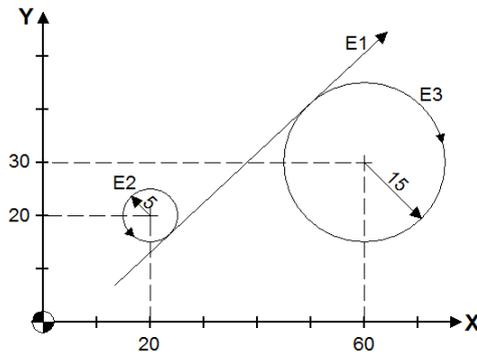
Straight line tangent to two circles.

The straight line is defined by the two circles it is tangent to. The format is the following, the optional part is shown between brackets:

Er = [origin,] G810 circle1, G811 circle2

E5 = G810 X12 Y15 I6, G811 X50 Y28 I-13

E5 = G810 E7, G811 E8



E1=G810 X20 Y20 I5, G811 X60 Y30 I-15

E1=G810 E2, G811 E3

5.

GEOMETRICAL ENTITIES.
Saving the various types of geometrical entities.



FAGOR AUTOMATION

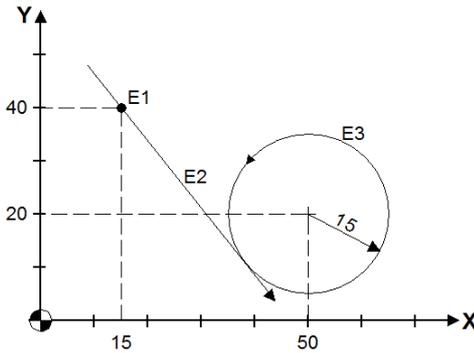
CNC 8060
CNC 8065

(REF: 1709)

Straight line that goes through one point and is tangent to a circle.

The straight line is defined by a point it goes through and a circle it is tangent to. The format is the following, the optional part is shown between brackets:

Er = [origin,] G810 point, G811 circle



E2=G810 X15 Y40, G811 X50 Y20 I15

E2=G810 E1, G811 E3

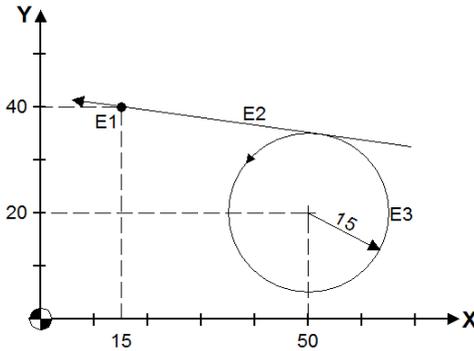
5.

GEOMETRICAL ENTITIES.
Saving the various types of geometrical entities.

Straight line that is tangent to a circle and goes through a point.

The straight line is defined by a circle it is tangent to and a point that it goes through. The format is the following, the optional part is shown between brackets:

Er = [origin,] G810 circle, G811 point



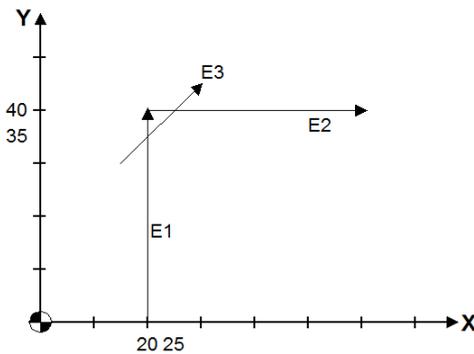
E2=G810 X50 Y20 I15, G811 X15 Y40

E9=G810 E3, G811 E1

Straight line defined with two straight lines and a chamfer.

The straight line is defined by a chamfer between two straight lines, where each straight line may be defined by G813 or by G810-G811. The format is the following, the optional part is shown between brackets:

Er = [origin,] G813 straight, G821 J, G813 straight



E3=G813 X20 Y0 J90, G821 J5,
G813 Y40 J0

E3=G810 X20 Y0, G811 X20 Y20,
G821 J5, G813 Y40 J0

E3=G813 X20 Y0 J90, G821 J5,
G810 X0 Y40, G811 X20 Y40

E3=G810 X20 Y0, G811 X20 Y20,
G821 J5, G810 X0 Y40, G811 X20 Y40

E3=G813 E1, G821 J5, G813 E2



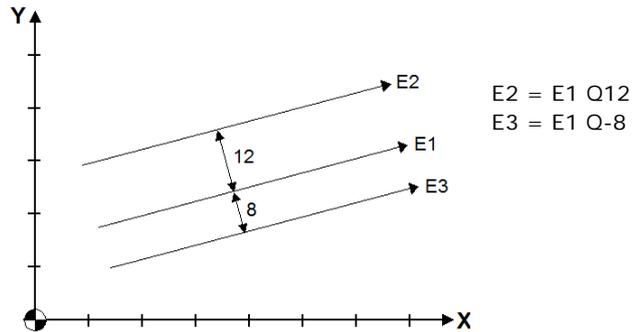
CNC 8060
CNC 8065

Straight line parallel to a predefined straight line.

A straight line may be defined as parallel to another predefined straight line. The format is the following, the optional part is shown between brackets:

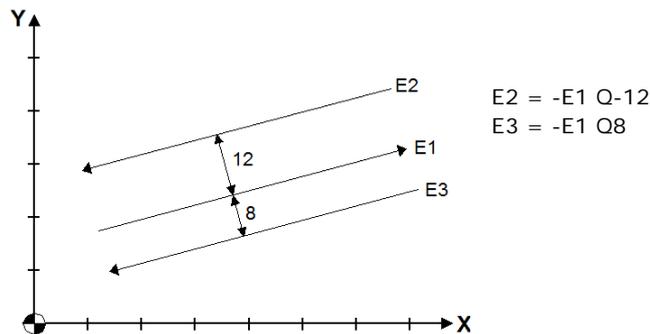
$$Er = \text{straight } Q..$$

Where Q is the distance between the two straight lines, positive if the straight line to be defined is to the left and negative if it is to the right, as viewed in the direction of the predefined straight line.



The orientation of the straight line may be changed with the "-" (minus) sign in the definition. In this case, to define the Q distance, the straight line is shown with the new orientation. The format is the following:

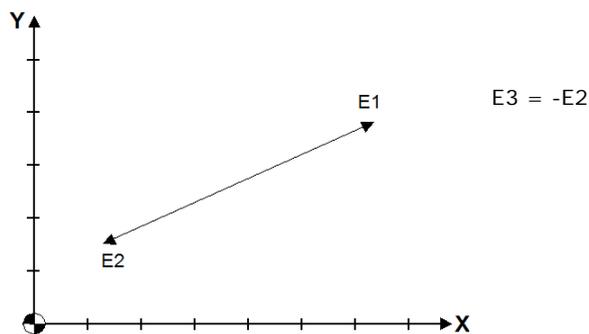
$$Er = - \text{straight } Q..$$



Straight line traveled in the opposite direction.

The "-" (minus) sign is used to change the direction of a straight line. This way, the two straight lines coincide but the new straight line travels in the opposite direction. The format is the following:

$$Er = -Er$$



5.

GEOMETRICAL ENTITIES.
 Saving the various types of geometrical entities.



CNC 8060
CNC 8065

(REF: 1709)

5.1.3 Saving circles.

A circle may be defined either directly defining the center and the radius or indirectly. A circle may be defined as follows:

- Circle of known center and radius.
- Circle of known radius and tangent to two straight lines.
- Circle of known radius and tangent to a straight line and to a circle.
- Circle of known radius and tangent to a circle and to a straight line.
- Circle of known radius and tangent to two circles.
- Circle of known radius going through a point and tangent to a straight line and vice versa.
- Circle of known radius going through a point and tangent to a circle and vice versa.
- Circle of known radius and goes through two points.
- Circle going through three points.
- Circle with center in a point and tangent to a straight line.
- Circle with center in a point and tangent to a circle.
- Concentric circles.
- Circle traveled in the opposite direction.
- Circle tangent to three geometrical entities.
- Circle tangent to two geometrical entities in a single point.

The points, straight lines and circles used to define the point may be defined either directly (using the proposed definitions) or indirectly (using geometrical entities saved earlier).

```
Ec = G813 X.. Y.. J.., G821 I.., G813 X.. Y.. J..
Ec = G813 Ec J.., G821 I.., G813 X.. Y.. J..
Ec = Er, G821 I.., G813 X.. Y.. I.. J..
Ec = G820 X.. Y.. I.., G821 I.., G820 X.. Y.. I..
Ec = Ec, G821 I.., G820 Ep I..
```

The elements saved earlier may be used with the opposite diirection if they have the "-" (minus) sign.

```
Ec = G820 X.. Y.. I.., G821 I.., - Ec
```

Circle of known center and radius.

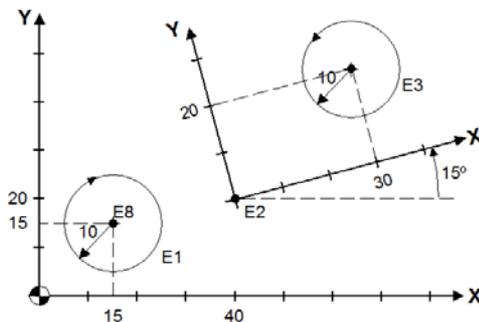
The format is the following:

```
Ec = G820 X.. Y.. I..
```

X.. Y.. Center coordinates.

I.. Circle radius, positive when traveled counterclockwise and negative when traveled clockwise.

A circle may be referred to an origin other than the part zero, in which case, the origin will be programmed directly or indirectly before defining the circle. The center of the circle may be referred to the point saved earlier.



```
E1=G820 X15 Y15 I-10
E2=G851 X40 Y20 J15, G820 X30 Y20 I10
E1=G820 E8 I-10
E2=E2, G820 X30 Y20 I10
```

5.

GEOMETRICAL ENTITIES.
Saving the various types of geometrical entities.



FAGOR AUTOMATION

CNC 8060
CNC 8065

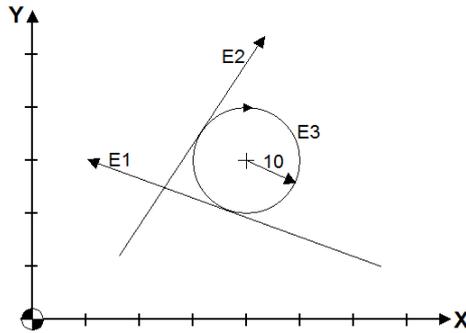
(REF: 1709)

Circle of known radius and tangent to two straight lines.

The format is the following, the optional part is shown between brackets:

$$E_c = [\text{origin},] \text{straight1}, G821 \text{ I}.., \text{straight2}$$

Each straight line may be defined with G813 or with G810-G811.



$$E3 = G813 \text{ X}.. \text{Y}.. \text{J}.., G821 \text{ I-10}, G813 \text{ X}.. \text{Y}.. \text{J}..$$

$$E3 = G810 \text{ X}.. \text{Y}.. \text{I}.., G811 \text{ X}.. \text{Y}.. \text{I}.., G821 \text{ I-10}, G813 \text{ X}.. \text{Y}.. \text{J}..$$

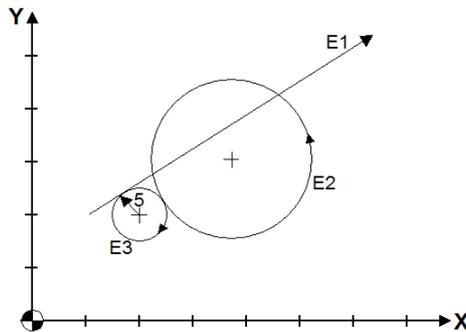
$$E3 = G810 \text{ X}.. \text{Y}.. \text{I}.., G811 \text{ X}.. \text{Y}.. \text{I}.., G821 \text{ I-10}, G810 \text{ X}.. \text{Y}.. \text{I}.., G811 \text{ X}.. \text{Y}.. \text{I}..$$

$$E3 = E1, G821 \text{ I-10}, E2$$

Circle of known radius and tangent to a straight line and to a circle.

The format is the following, the optional part is shown between brackets:

$$E_c = [\text{origin},] \text{straight}, G821 \text{ I}.., \text{circle}$$



$$E3 = G813 \text{ X}.. \text{Y}.. \text{J}.., G821 \text{ I-5}, G820 \text{ X}.. \text{Y}.. \text{I}..$$

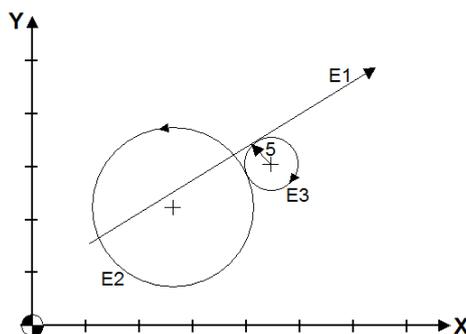
$$E3 = E1, G821 \text{ I-5}, E2$$

Circle of known radius and tangent to a circle and to a straight line.

The format is the following, the optional part is shown between brackets:

$$E_c = [\text{origin},] \text{straight}, G821 \text{ I}.., \text{straight}$$

The straight line may be defined with G813 or with G810-G811.



$$E3 = G820 \text{ X}.. \text{Y}.. \text{I}.., G821 \text{ I-5}, G813 \text{ X}.. \text{Y}.. \text{J}..$$

$$E3 = G820 \text{ X}.. \text{Y}.. \text{I}.., G821 \text{ I-5}, G810 \text{ X}.. \text{Y}.. \text{I}.., G811 \text{ X}.. \text{Y}.. \text{I}..$$

$$E3 = E2, G821 \text{ I-5}, E1$$

5.

GEOMETRICAL ENTITIES.
Saving the various types of geometrical entities.



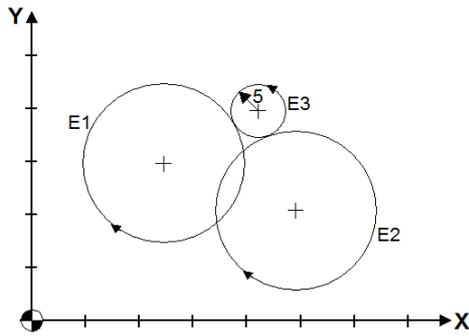
CNC 8060
CNC 8065

(REF: 1709)

Circle of known radius and tangent to two circles.

The format is the following, the optional part is shown between brackets:

Ec = [origin,] circle1, G821 I.., circle2



E3=G820 X.. Y.. I.., G821 I5,
G820 X.. Y.. I..

E3=E1, G821 I5, E2

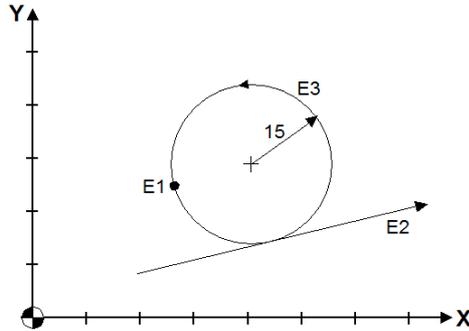
Circle of known radius going through a point and tangent to a straight line and vice versa.

The format is the following, the optional part is shown between brackets:

Ec = [origin,] point, G821 I.., straight

Ec = [origin,] straight, G821 I.., point

The straight line may be defined with G813 or with G810-G811.



E3=G820 X.. Y.., G821 I15,
G813 X.. Y.. J..

E3=G820 X.. Y.., G821 I15,
G810 X.. Y.. I.., G810 X.. Y.. I..

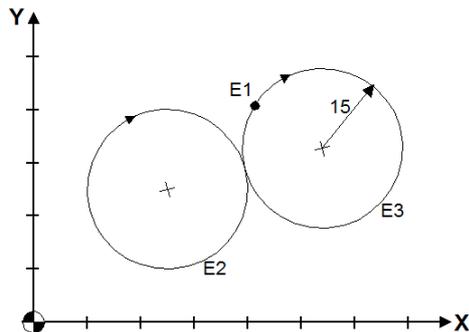
E3=E1, G821 I15, E2

Circle of known radius going through a point and tangent to a circle and vice versa.

The format is the following, the optional part is shown between brackets:

Ec = [origin,] point, G821 I.., circle

Ec = [origin,] circle, G821 I.., point



E3=G820 X.. Y.., G821 I-15,
G820 X.. Y.. I..

E3=E1, G821 I-15, E2

5.

GEOMETRICAL ENTITIES.
Saving the various types of geometrical entities.



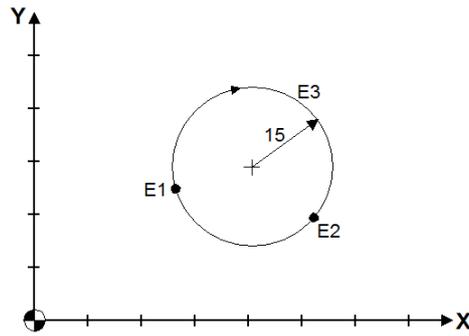
**CNC 8060
CNC 8065**

(REF: 1709)

Circle of known radius and goes through two points.

The format is the following, the optional part is shown between brackets:

Ec = [origin,] point, G821 I.., point



E3=G820 X.. Y.., G821 I-15, G820 X.. Y..

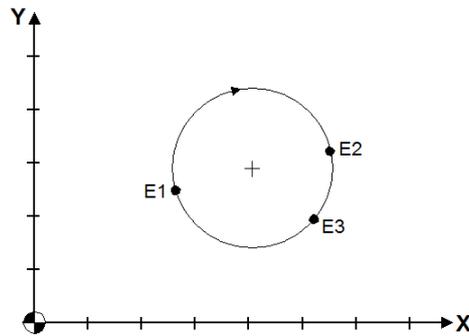
E3=E1, G821 I-15, E2

Circle going through three points.

The format is the following, the optional part is shown between brackets:

Ec = [origin,] G810 point, G820 point, G811 point

Ec = [origin,] G820 point, G820 point, G820 point



E3=G810 X.. Y.., G820 X.. Y.., G811 X.. Y..

E3=G810 E1, G820 E2, G811 E3

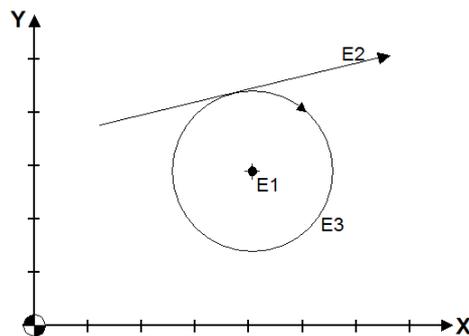
E3=G820 X.. Y.., G820 X.. Y.., G820 X.. Y..

E3=G820 E1, G820 E2, G820 E3

Circle with center in a point and tangent to a straight line.

The format is the following, the optional part is shown between brackets:

Ec = [origin,] G821 point, straight line



E3=G821 X.. Y.., G813 X.. Y.. J..

E3=G821 E1, E2

5.

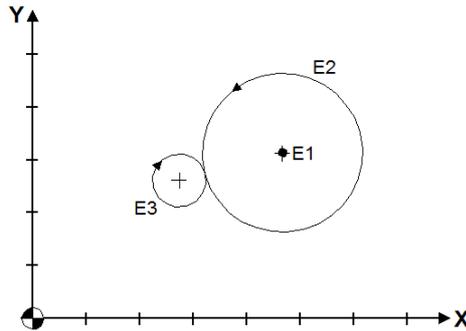
GEOMETRICAL ENTITIES.
Saving the various types of geometrical entities.

Circle with center in a point and tangent to a circle.

The format is the following, the optional part is shown between brackets:

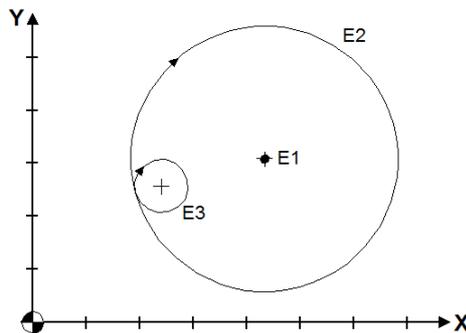
$E_c = [origin,] G821 \text{ point, circle } [K2]$

Parameter K2 selects the solution with the largest radius. If K2 is not programmed, it takes the solution with the smallest radius.



$E2 = G821 X.. Y., G820 X.. Y.. I..$

$E2 = G821 E1, E3$



$E2 = G821 X.. Y., G820 X.. Y.. I.. K2$

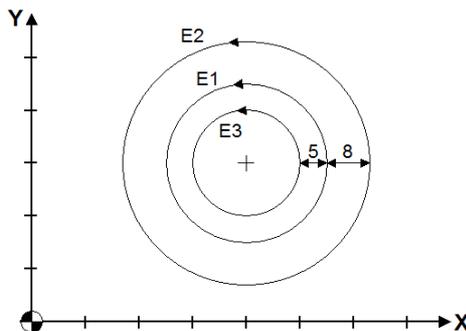
$E2 = G821 E1, E3 K2$

Concentric circles (1).

The format is the following:

$E_c = \text{circle } Q..$

Where Q is the distance between the two circles, positive if the circle to be defined is to the left and negative if it is to the right, as viewed in the direction of the predefined circle.



$E3 = E1 Q5$

$E3 = E1 Q-8$

5.

GEOMETRICAL ENTITIES.
Saving the various types of geometrical entities.



FAGOR AUTOMATION

CNC 8060
CNC 8065

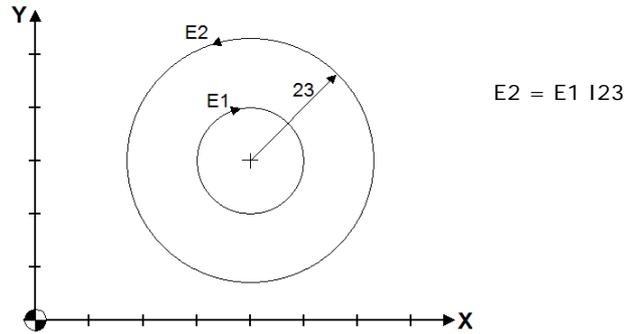
(REF: 1709)

Concentric circles (2).

The format is the following:

$$E_c = \text{circle } I..$$

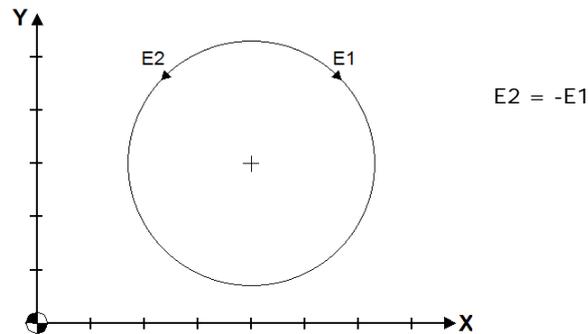
It defines a circle of radius I with the same center. The predefined circle can only be an element saved earlier.



Circle traveled in the opposite direction.

The "-" (minus) sign is used to change the direction of a circle. This way, the two circles coincide but the new circle travels in the opposite direction. The format is the following:

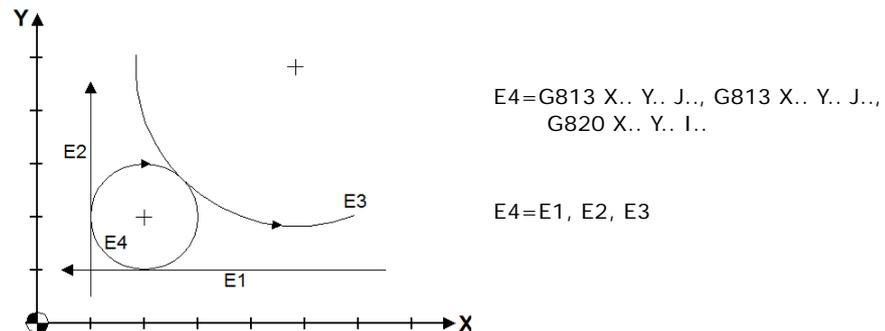
$$E_c = -E_c$$



Circle tangent to three geometrical entities.

The three support geometrical entities may be points, straight lines or circles. The format is the following, the optional part is shown between brackets:

$$E_c = [\text{origin},] \text{entity1}, \text{entity2}, \text{entity3}$$



5.

GEOMETRICAL ENTITIES.
 Saving the various types of geometrical entities.



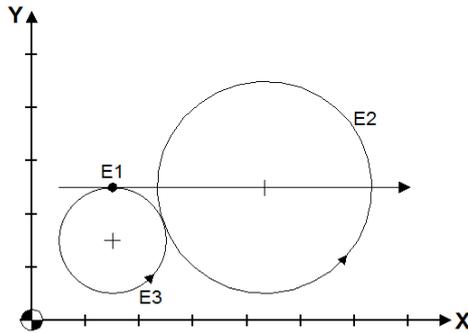
CNC 8060
CNC 8065

(REF: 1709)

Circle tangent to two geometrical entities in a single point.

The point must belong to one of the two entities (circle or straight line) and must be programmed as a circle of 0 radius (G820 X.. Y..).

Ec = straight, G820 point, circle



E1 = G813 X0 Y25 J0

E2 = G820 X45 Y25 I20

E3 = E1, G820 X16 Y20, E2

5.

GEOMETRICAL ENTITIES.
Saving the various types of geometrical entities.



FAGOR AUTOMATION

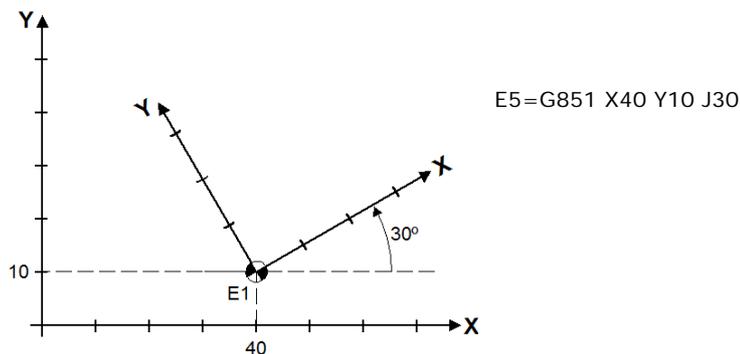
CNC 8060
CNC 8065

(REF: 1709)

5.1.4 Saving origin changes.

An origin may be defined either directly or indirectly using a point defined earlier.

- Direct definition. $E_o = G851 X.. Y.. J..$
- Indirect definition. $E_o = G851 E_p J..$



The rotational translation function G851 may be programmed directly inside a profile. The geometrical entities after its definition will be referred to the new origin. This programming method allows having different parts of a profile referred to different origin points. The rotational translation is canceled with function G850.

The profile can only be translated with G852.

The geometrical entities defined with G851 are saved resolved with respect to that G851 (always K1 type). For example, a straight line defined with G851 is resolved and saved with a point and angle. If later on in the profile, this entity is used with a previous G851, this element will be affected by the G851 programmed earlier or by the G851 used for its definition, but this time it will behave like a K2 type G851.

5.

GEOMETRICAL ENTITIES.
Saving the various types of geometrical entities.



FAGOR AUTOMATION

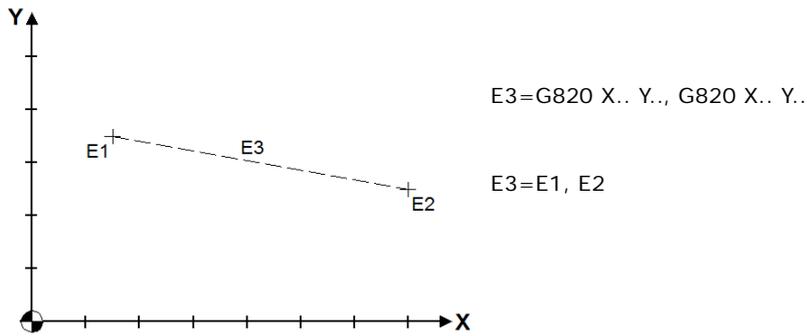
CNC 8060
CNC 8065

(REF: 1709)

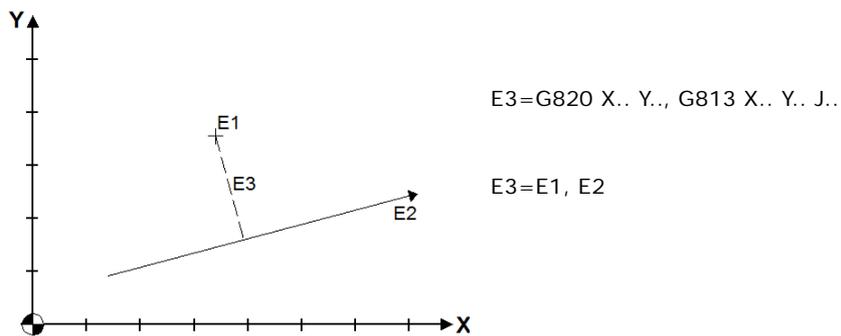
5.1.5 Saving a distance.

A distance is defined indirectly.

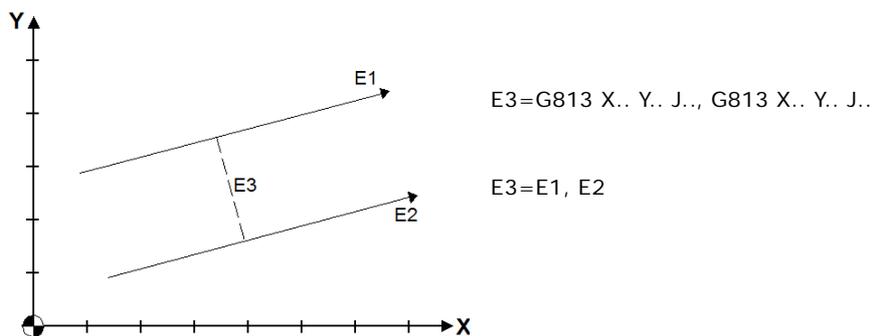
- Distance between two points.



- Distance between a point and a straight line.



- Distance between two straight lines.



5.

GEOMETRICAL ENTITIES.

Saving the various types of geometrical entities.



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

5.2 Using the saved entities inside a profile.

The saved entities (points, straight lines and circles) may be used inside a profile just writing the E character followed by the number that identifies it.

```
Example:
G813 X0 Y-60 J180
G821 I-12
E7
G813 X-25 Y0 J90
E8
```

The saved entities may also be used to build other elements of the profile.

```
Example:
E7
G813 E4 J-45
G810 X-23 Y35 I-45
G811 E16
G820 E12 I-33 K2
```

The "point" entity can also be called from a canned cycle as a point where a G851 will be applied.

Using a geometrical entity in the profile requires programming the known data of the next element, without leaving any values out.

For a rapid linear interpolation, add a G0 to the block that contains the element.

5.

GEOMETRICAL ENTITIES.
Using the saved entities inside a profile.

5.3 Variables related to the geometrical entities.

(V.)G.E1[0]

..

(V.)G.E30[0]

Variable that can be read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Type of geometrical entity.

Syntax.

·n· Number of the geometrical entity (between 1 and 30).

V.G.E1[0] Type of E1 entity.

Values of the variable.

Value.	Meaning.
0	Undefined entity.
1	Point.
2	Circle.
3	Straight line.
4	Origin.
5	Distance.

(V.)G.E1[1]

..

(V.)G.E30[1]

Variable that can be read from the program.

The variable returns the execution value; reading it interrupts block preparation.

First value of the geometrical entity.

- If the entity is an origin, a point or a circle, the variable returns the value of the abscissa.
- If the entity is a straight line, the variable returns the abscissa value.
- If the entity is a distance, the variable returns the distance.
- If the entity is not defined, the variable returns an error.

Syntax.

·n· Number of the geometrical entity (between 1 and 30).

V.G.E1[1] First value of the E1 entity.

(V.)G.E1[2]

..

(V.)G.E30[2]

Variable that can be read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Second value of the geometrical entity.

- If the entity is an origin, a point or a circle, the variable returns the value of the ordinate.
- If the entity is a straight line, the variable returns the ordinate value
- If the entity is a distance, the variable returns an error.
- If the entity is not defined, the variable returns an error.

Syntax.

·n· Number of the geometrical entity (between 1 and 30).

V.G.E1[2] Second value of the E1 entity.

5.

GEOMETRICAL ENTITIES.

Variables related to the geometrical entities.

FAGOR 

FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

(V.)G.E1[3]

..

(V.)G.E30[3]*Variable that can be read from the program.**The variable returns the execution value; reading it interrupts block preparation.*

Third value of the geometrical entity.

- If the entity is a circle, the variable returns the value of the radius.
- If the entity is a straight line, the variable returns the value of the angle.
- If the entity is an origin, a point or a distance, the variable returns an error.
- If the entity is not defined, the variable returns an error.

Syntax.

·n· Number of the geometrical entity (between 1 and 30).

V.G.E1[3]

Third value of the E1 entity.

5.**GEOMETRICAL ENTITIES.**

Variables related to the geometrical entities.



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

5.4 Summary of the possible ways to define the geometrical entities.

The following summary shows the possible cases to define geometrical entities by programming the elements directly, but they can also be defined using other entities defined earlier.

Defining a "point" entity.

Ep = G810 X.. Y.. I.., G811 X.. Y.. I.., G810 X.. Y.. I.., G811 X.. Y.. I..
 Ep = G810 X.. Y.. I.., G811 X.. Y.. I.., G813 X.. Y.. J..
 Ep = G813 X.. Y.. J.., G810 X.. Y.. I.., G811 X.. Y.. I..
 Ep = G813 X.. Y.. J.., G813 X.. Y.. J..
 Ep = G813 X.. Y.. J.., G820 X.. Y.. I.. [K2]
 Ep = G820 X.. Y..
 Ep = G820 X.. Y.. I.., G813 X.. Y.. J.. [K2]
 Ep = G820 X.. Y.. I.., G820 X.. Y.. I.. [K2]
 Ep = Ec IO

Defining a "straight line" entity.

Er = G810 X.. Y.. I.., G811 X.. Y.. I..
 Er = G810 X.. Y.. I.., G811 X.. Y.. I.., G821 J.., G813 X.. Y.. J..
 Er = G813 X.. Y.. J..
 Er = G813 Ec J..
 Er = G813 X.. Y.. J.., G821 J.., G810 X.. Y.. I.., G811 X.. Y.. I..
 Er = G813 X.. Y.. J.., G821 J.., G813 X.. Y.. J..
 Er = Er Q
 Er = -Er

Defining a "circle" entity.

G810 X.. Y.. I.., G811 X.. Y.. I.., G821 I.., G810 X.. Y.. I.., G811 X.. Y.. I..
 G810 X.. Y.. I.., G811 X.. Y.. I.., G821 I.., G813 X.. Y.. J..
 G810 X.. Y.. I.., G811 X.. Y.. I.., G821 I.., G820 X.. Y..
 G810 X.. Y.. I.., G811 X.. Y.. I.., G821 I.., G820 X.. Y.. I..
 G810 X.. Y.., G820 X.. Y.., G811 X.. Y..
 G813 X.. Y.. J.., G821 I.., G810 X.. Y.. I.., G811 X.. Y.. I..
 G813 X.. Y.. J.., G821 I.., G813 X.. Y.. J..
 G813 X.. Y.. J.., G821 I.., G820 X.. Y.. I..
 G820 X.. Y.., G810 X.. Y.. I.., G811 X.. Y.. I..
 G820 X.. Y.., G813 X.. Y.. J..
 G820 X.. Y.., G820 X.. Y.., G820 X.. Y..
 G820 X.. Y.., G821 I.., G810 X.. Y.. I.., G811 X.. Y.. I..
 G820 X.. Y.., G821 I.., G813 X.. Y.. J..
 G820 X.. Y.., G821 I.., G820 X.. Y..
 G820 X.. Y.., G821 I.., G820 X.. Y.. I..
 G820 X.. Y.. I..
 G820 X.. Y.. I.., G813 X.. Y.. J.., G810 X.. Y.. I.., G811 X.. Y.. I..
 G820 X.. Y.. I.., G821 I.., G810 X.. Y.. I.., G811 X.. Y.. I..
 G820 X.. Y.. I.., G821 I.., G820 X.. Y..
 G820 X.. Y.. I.., G821 I.., G820 X.. Y.. I..
 G821 X.. Y.., G810 X.. Y.. I.., G811 X.. Y.. I..
 G821 X.. Y.., G813 X.. Y.. J..
 G821 X.. Y.., G820 X.. Y.. I.. [K2]
 Ec Q..
 Ec I..
 E1, E2, E3 (any element, point, straight line, circle)
 E1, G820 X.. Y.., E3
 G820 X.. Y.., G820 X.. Y..

5.

GEOMETRICAL ENTITIES.

Summary of the possible ways to define the geometrical entities.


FAGOR

FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

Defining a "distance" entity.

G813 X.. Y.. J.., G813 X.. Y.. J.. (distance between two parallel straight lines)
G820 X.. Y.., G813 X.. Y.. J..
G820 X.. Y.., G820 X.. Y..

Defining an "origin" entity.

G851 X.. Y.. J..

5.

GEOMETRICAL ENTITIES.

Summary of the possible ways to define the geometrical entities.



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

PROGRAMMING CYCLES (MACROS).

6

6.1 G8726/ G8727/G8728. Ruled surfaces.

Functions G8726/ G8727/G8728 allow 3D machining generating a ruled surfaced between any two profiles located in parallel planes at a particular distance. The functions for programming the cycle are the following.

- G8726 Define the general parameters of the cycle and the starting point of the first profile.
- G8727 Second profile.
- G8728 Activate the cycle.

The profiles are programmed in the XY plane with a depth in Z. The machining operation may be done in any plane using the plane rotation functions (#CS, G73, etc).

Programming.

The structure of the cycle is the following:

- Program function G8726 together with the cycle calling parameters and the starting point of the surface profile.
- Program the second profile. Program function G8727 together with the calling parameters and then the profile.
- Program function G8728 to end the macro program and activate the execution of the cycle.

General syntax.

G8726 [X.. Y..] Z.. I.. [J..] [H..] [D1=..] [D2=..] [D6=] (Beginning of the cycle)

G.. X.. Y.. (Surface profile)

G.. X.. Y.. (Surface profile)

G.. X.. Y.. (Surface profile)

G8727 [X.. Y..] Z.. [K1]

G.. X.. Y.. (Second profile)

G.. X.. Y.. (Second profile)

G.. X.. Y.. (Second profile)

G8728 (Activate execution of the cycle)

Programming format. G8726.

The format is the following, the optional part is shown between brackets:

G8726 [X.. Y..] Z.. I.. [J..] [H..] [D1=..] [D2=..] [D6=]

X.. Y.. Starting point of the first profile.

Z.. Z Position of the first profile.

I.. Distance between passes, in the first profile.

J.. The excess material (if not programmed, J=0 mm).

H.. Link management.

- H1: Links not managed (by default).
- H2: Links managed.



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

- D1=.. Absolute Z coordinate for the rapid tool withdrawal for unidirectional passes (if the passes are not programmed, they will be considered bidirectional).
- D2=.. Incremental withdrawal and repositioning Z coordinate for unidirectional passes (if not programmed, D2=2 mm).
- D6=.. Tool length with zero in the center or on the tip.
- D6=0: Tool with zero on the tip (default value).
 - D6=1: Tool with zero in the center; tool center if it is spherical, center of the tool tip radius if the tool is toric and it does nothing if the tool is flat.

6.

PROGRAMMING CYCLES (MACROS).
G8726/ G8727/G8728. Ruled surfaces.

Programming format. G8737.

The format is the following, the optional part is shown between brackets:

G8727 [X.. Y..] Z.. [K1]

X.. Y.. Profile starting point

Z.. Z Position of the profile.

K1 The CNC uses the profile where K1 has been programmed (K1 can only be programmed in one profile) to calculate the distance between the ruled surfaces. The CNC uses this profile to define the direction of the tool compensation on the ruled surfaces (parameter K1 in G8726) through G41/G42. If no profile defined with G8727 has parameter K1, the CNC uses the first profile (i.e. the profile defined after G8726).

Programming format. G8738.

The G8738 function ends the programming of the cycle and activates its execution. The format is the following.

G8738

Some general points to consider.

Tool radius compensation and tool type.

The machining operation can be carried out with spherical or toric tools (by defining them in the tool table or through the CNC variables V.G.TOR, etc.) only when machining the surface in the G17 plane (tool in Z), G18 (tool in Y), G19 (tool in X).

In any case, for example when the machining is rotated using coordinate transformation (#CS, G73, etc) only spherical end mills may be used with zero in the center of the tool D6=1).

Tool radius compensation and compensation strategy.

Tool radius compensation in ruled surfaces is obtained by shifting the straight paths that make up the surface one by one; if these paths do not have the same orientation (which is very common) even if these paths end in the same Z coordinate, their compensated paths will not end in the same Z coordinate..

Point profiles.

The first and/or last profile may be a single ISO point (with no entry or exit); in this case, as type of tool compensation only the compensation on straight paths of the ruled surface is allowed (G8726 K1) and the tool compensation on the profiles (G8726 K2) cannot be used.

Movement "perpendicular" to the surface.

In single-direction passes, the CNC always generates the fast withdrawal movements (D1=...) and in-process repositioning (D2=...) along the axis perpendicular to the work plane; Z axis for plane G17, Y axis for G18, X axis for G19.



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

Surface rotation and offset.

The rotation and offset values may be programmed before function G8726, with functions G158, G73 and #CS.

6.

PROGRAMMING CYCLES (MACROS):
G8726/ G8727/G8728. Ruled surfaces.

FAGOR 

FAGOR AUTOMATION

CNC 8060
CNC 8065

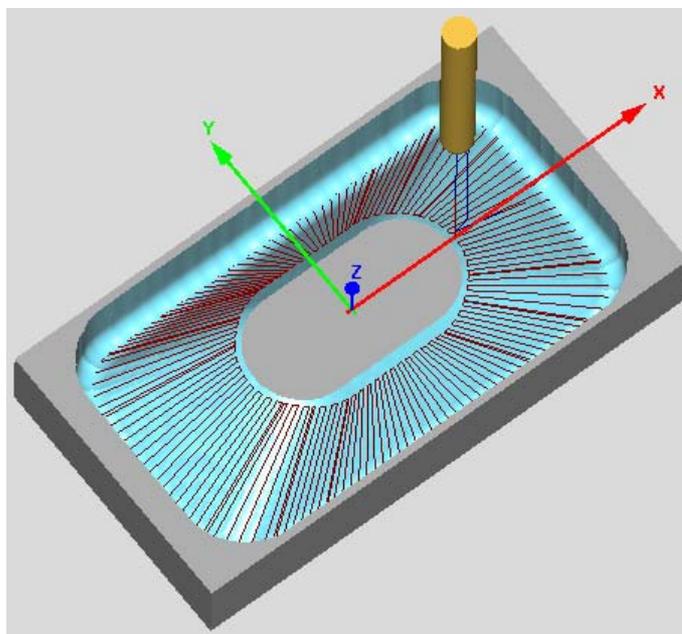
(REF: 1709)

6.1.1 Programming example.

```

G17
G54
T11 M6
F1000 S2000 M3
G0 Z100
X60 Y0
Z10
V.G.TOR=7.5 V.G.TOTIPR=7.5
G8726 X60 Y0 Z0 I2 (declaration of macro and parameters)
  G842 K1 X60 Y0
  G820 X25 Y0 I25
  G813 J180
  G820 X-25 Y0 I25
  G813 J0
  G820 X25 Y0 I25
  G840 X60 Y0 K1
G8727 X120 Y0 Z-30 (second profile of the surface)
  G842 K1 X120 Y0
  G813 X100 J90
  G821 I15
  G813 Y50 J180
  G821
  G813 X-100 J-90
  G821
  G813 Y-50 J0
  G821
  G813 X100 J90
  G840 X120 Y0K1
G8728 (activate the cycle)
G0 Z100 (tool withdrawal)
M30

```



6.

PROGRAMMING CYCLES (MACROS).
G8726/ G8727/G8728. Ruled surfaces.

6.2 G8734/G8735. Spiral profile milling.

Functions G8734/G8735 may be used for spiral milling of a plane profile. The profile may be programmed in ISO or ProGTL3 language or with the profile editor. The functions for programming spiral machining are the following.

- G8735 Define the cycle and its parameters.
 G8734 Activate the cycle.

Programming.

Program function G8735 and the cycle calling parameters. Then, program the blocks that define the profile. Program function G8734 to end the profile program and activate the execution of the cycle. We recommend a Z withdrawal after function G8734.

The profile must be closed; i.e. its starting and end points must be the same.

Programming format. G8735.

The format is the following, the optional part is shown between brackets:

G8735 I.. [J..] [K..] [H..] [Z..] [D0=..] [D1=..]

I.. Depth increment.

- Depth increment of the profile per turn (in H1 mode).
- Depth increment of the profile per mm (in H2 mode).
- Number of repetitions (in H3 mode).

J.. Direction of the increment.

- J1: Increment in Z- (by default).
- J2: Increment in Z+.

K.. Repeat the profile at the final depth (yes/no).

- K1: Repeat the profile at constant Z (by default).
- K2: Do not repeat the profile at the final depth.

H.. Type of increment.

- H1: Defined profile depth increment per turn (by default).
- H2: Defined profile depth increment per mm.
- H3: Increment calculated depending on Z and I.

Z.. Final coordinate (machining depth).

D0=.. Exit from the profile at each turn or only at the last turn.

- D0=0: Exit from the profile at each turn (by default).
- D0=1: Exit from the profile only at the last turn.

D1=.. Feedrate multiplying factor for the last exit from the profile (from 0.01 to 1; by default D1=1)

Programming format. G8734.

The G8734 function must be programmed alone in the block. The G8734 function ends the programming of the cycle and activates its execution.

Execution.

The cycle begins at the Z coordinate programmed before function G8735. The entry to the profile is done at constant Z, the spiral movement of the Z axis begins over the final point of the entry; i.e. over the starting point of the profile.

When the cycle reaches the final Z coordinate of the work, it ends or repeats the profile at constant Z according to what has been programmed (parameter K).

6.

PROGRAMMING CYCLES (MACROS):

G8734/G8735. Spiral profile milling.

FAGOR 

FAGOR AUTOMATION

CNC 8060
CNC 8065

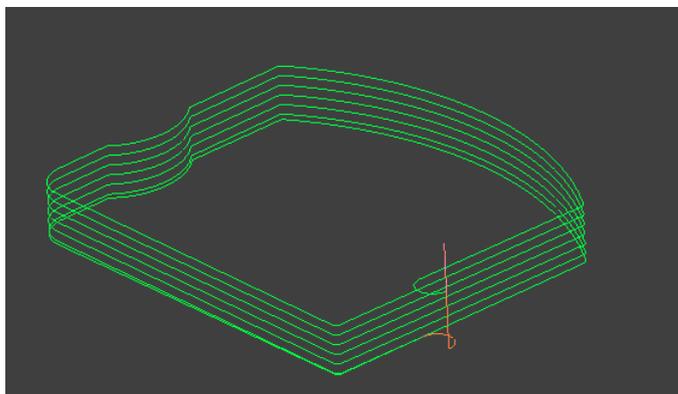
(REF: 1709)

6.2.1 Programming example.

```

T1 M6
F1000 S2000
G0 Z20
G0 X120 Y0
G0 Z-5 (machining begins at Z-5)
G8735 Z-35 I5 (declaration of macro and parameters)
  G842 K2 X120 Y0
  G813 X100 J90
  G820 X0 Y-50 I180 K2
  G813 X-100 Y50 J-90
  G820 X-130 Y20 I-40 K1
  G813 X-100 Y0 J-90 K2
  G821 I15
  G813 Y-50 J0
  G813 X100 J90
  G840 X120 Y0 K2
G8734 (activate the cycle)
G1 Z20 (tool withdrawal)
M30

```



6.

PROGRAMMING CYCLES (MACROS).
G8734/G8735. Spiral profile milling.

6.3 G8736/G8737/G8738. Milling of a surface defined by a plane profile and one or more section profiles.

Functions G8736/G8737/G8738 may be used to machine a surface generated by a plane profile that moves along one or more section profiles (up to 10). The functions for programming the cycle are the following.

- G8736 Define the cycle and its parameters.
- G8737 Beginning of the section profiles.
- G8738 Activate the cycle.

The plane profile may be either closed or open. The profiles may be programmed in ISO and ProGTL3 languages or with the profile editor; they can also be defined directly in the program or in a local or global subroutine.

Programming.

The structure of the cycle is the following:

- Program function G8736 together with the cycle calling parameters.
- Program the plane profile.
- Program the section profiles. Program function G8737 together with the cycle calling parameters and then the profile. Repeat this step for each section profile.
- Program function G8738 to end the profile program and activate the execution of the cycle. We recommend a Z withdrawal after function G8738.

General syntax.

G8736 I.. [X.. Y..]/[R.. Q..] [J..] [K..] [H..] [D0=..] .. [D9=..] (Beginning of the cycle)

G.. X.. Y.. (Beginning of plane profile)

G.. X.. Y.. H1 (support block for section profile 1)

G.. X.. Y..

G.. X.. Y.. Hn (support block for section profile n)

G.. X.. Y.. (End of plane profile)

G8737 [H1] [K1] (Beginning of section profile 1)

G.. X.. Y.. (blocks of section profile)

G.. X.. Y..

G8737 [Hn] (Beginning of section profile n)

G.. X.. Y.. (blocks of section profile)

G.. X.. Y..

G8738 [J..] [X.. Y..] (Activate execution of the cycle)

Programming format. G8736.

The format is the following, the optional part is shown between brackets:

G8736 I.. [X.. Y..]/[R.. Q..] [J..] [K..] [H..] [D0=..] .. [D9=..]

I.. Distance between passes, measured on the section profile that contains parameter K1 (in G8737).

X.. Y.. Starting point of the plane profile in the Cartesian (XY) or Polar (RQ) system.

R.. Q..

J.. Excess stock.

K.. Type of machining of the plane profile junctions depending on the section profile.

- K1: Variable roundings and constant chamfers (by default).
- K2: Constant roundings and constant chamfers.
- K3: Constant roundings and variable chamfers.
- K4: Variable roundings and variable chamfers.

6.

PROGRAMMING CYCLES (MACROS).

G8736/G8737/G8738. Milling of a surface defined by a plane profile and one or more section profiles.

FAGOR 

FAGOR AUTOMATION

CNC 8060
CNC 8065

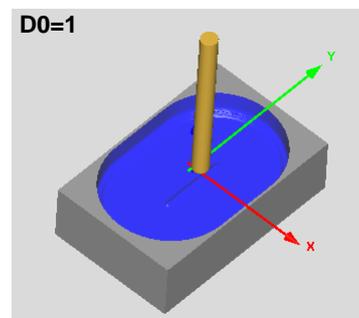
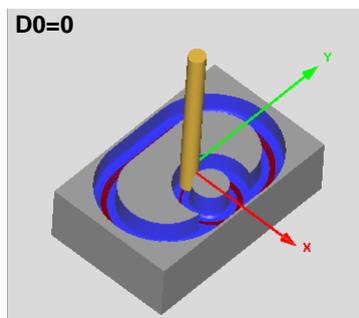
(REF: 1709)

6.

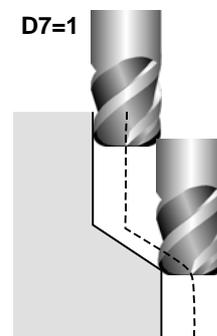
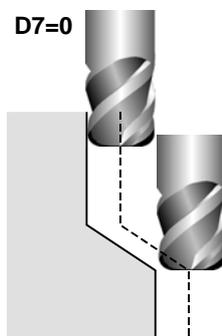
PROGRAMMING CYCLES (MACROS).

G8736/G8737/G8738. Milling of a surface defined by a plane profile and one or more section profiles.

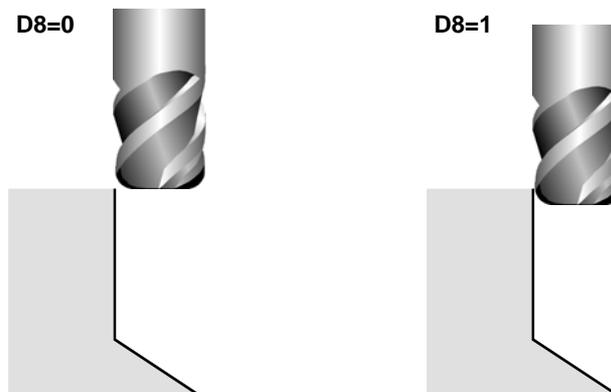
- H.. Type of passes, when the plane profile is open.
- H1: Unidirectional passes (by default).
 - H2: Bi-directional passes.
- D0=.. Type of machining, when the plane profile is closed.
- D0=0: Plane milling (by default).
 - D0=1: Surface emptying (pocket).
 - D0=2: Spiral milling.
 - D0=3: Machine only the bottom of the pocket.



- D1=.. Z coordinate for the rapid tool withdrawal at the end of each pass, when they are unidirectional.
- D2=.. Safety distance for tool withdrawal and repositioning at work feedrate at the beginning and end of each pass when they are unidirectional (if not programmed, D2=2).
- D3=.. Chordal error for spiral milling, when it is used to divide an arc in many segments (by default, 0.01mm).
- D4=.. Minimum chord for spiral milling; i.e. minimum distance between points (by default 0.1mm).
- D5=.. Maximum chord for spiral milling; i.e. maximum distance between points (by default 2 mm). This parameter is used to divide the lines into many segments.
- D6=.. Not used.
- D7=.. Execute automatic corner rounding to join blocks of the section profile; similar to the transition type (G136/G137) in plane profiles. This parameter is applied to all the section profiles.
- D7=0: Without corner rounding (default value).
 - D7=1: With corner rounding.

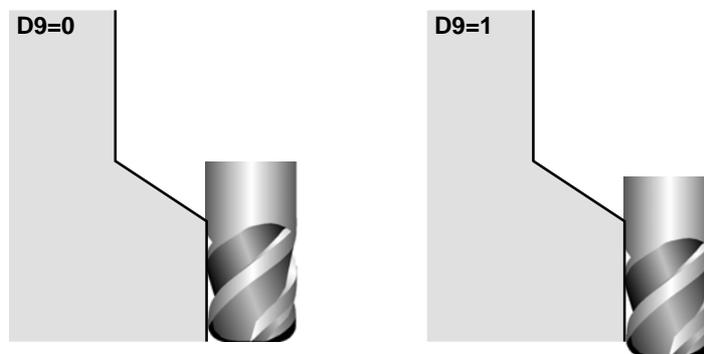


- D8=.. Treatment of the starting coordinate of the section profile (only for toric tool). If there are more than one section profiles, this parameter is not valid and it assumes its default value (D8=0).
- D8=0: The starting coordinate of the section profile is machined with the side of the tool (by default).
 - D8=1: The starting coordinate of the section profile is machined with the tip of the tool.



D9=.. Treatment of the final coordinate of the section profile (only for toric tool). If there are more than one section profiles, this parameter is not valid and it assumes its default value (D9=0).

- D9=0: The final coordinate of the section profile is machined with the side of the tool (by default).
- D9=1: The final coordinate of the section profile is machined with the tip of the tool.



Programming format. G8737.

The format is the following:

G8737 [H..] [K1]

H.. Number of the section profile (from H1 to H10) (it not programmed, H=1).

K1 The cycle uses the section profile that contains parameter K1 to calculate the distance between the various passes depending on the I distance programmed in G8736. If K1 is not programmed, the cycle calculates based on the first section profile.

Programming format. G8738.

The G8738 function ends the programming of the cycle and activates its execution. The format is the following, the optional part is shown between brackets:

G8738 [J..]

J.. Gap between machining passes, given in tool radius (by default J=1). Between 0 (not included) and 2 (included).

This parameter is only taken into account for G8736 D0=1.

Considerations for the profiles.

The profiles may be programmed in ISO and ProGTL3 languages or with the profile editor; they can also be defined directly in the program or in a local or global subroutine.

6.

PROGRAMMING CYCLES (MACROS).

G8736/G8737/G8738. Milling of a surface defined by a plane profile and one or more section profiles.

Plane profile.

- The plane profile is programmed after the G8736 block.
- The plane profile may be either closed or open.
- In the geometrical entities or plane profile points, the section profile to be applied (straight line, circle or rounding if it is variable) may be defined with parameter H (from H1 to H10). The section profile is applied based on the beginning of the entity and remains active until calling a new section profile.
- If there are several section profiles, the last one remains active (or H1, if none has been programmed) until a different one is activated.

Section profile.

- Each section profile must begin with function G8737.
- The section profile must be open.
- If there are several section profiles, all of them must have the same depth.
- The section profile is defined in the XY plane and its coordinates will be treated as "relative", where the 0,0 point of the section is supported on the plane profile, regardless of its own vertical coordinate (Y).

Considerations for the tool.

The cycle may be executed with spherical, toric or flat tools. The type of tools and its dimensions may be read in the tool table or may be defined in the program using variables.

V.G.DSUBTYPE Subtype of offset; 1 (flat), 2 (spherical), 3 (toric).

V.G.TOR Tool radius.

V.G.TOTIPR Tool tip radius (toric tool).

Example:

T1 M6 (Load tool 1)

V.G.DSUBTYPE=3 (Toric tool)

V.G.TOR=5 (Radius 5 mm)

V.G.TOTIPR=1 (tip radius 1 mm)



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

6.3.1 Rules for defining section profiles.

The section profile, defined in the XY plane, must follow the following rules.

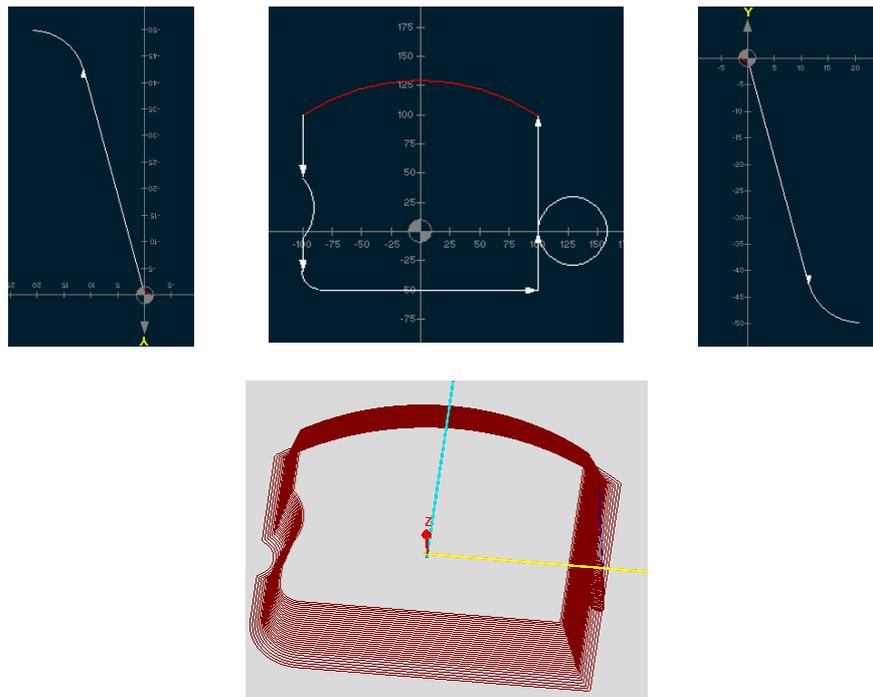
- The section profiles must be programmed in the same plane as that of the plane profile.
- For the machining operation, the positive Y axis must coincide with the positive perpendicular (Z) axis.
- The section profile must be open, with the starting and final points defined.
- Functions G841/G842/G840 must be programmed without parameter K of automatic entry and exit.
- If the macro contains several profile sections, the starting and final points of each profile (shifted the tool radius) must have the same Y coordinate. One way to make sure of this is to define the section profile in ProGTL3 language and program the same horizontal support straight line at the beginning and at the end of the profile.
- Functions G841/G842 of the plane profile are used to define whether the section profile is inside or outside the plane profile (pocket or boss).

G842 The section profiles develops as if its positive Y axis coincided with the direction of the plane profile.

G841 The section profiles develops as if its positive Y axis were in the opposite direction to the direction of the plane profile.

Profile programmed with G842.

The section profile develops as would be seen by aligning its positive Y axis in the same direction as that of the plane profile (in this case, outside the plane profile).



6.

PROGRAMMING CYCLES (MACROS):
G8736/G8737/G8738. Milling of a surface defined by a plane profile and one or more section profiles.

G8736/G8737/G8738. Milling of a surface defined by a plane profile and one or more section profiles.



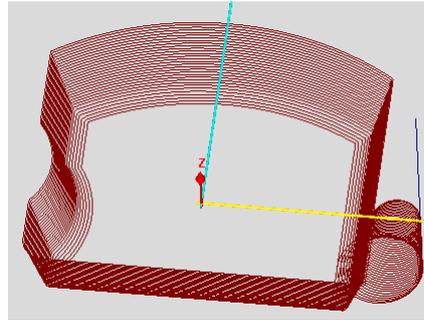
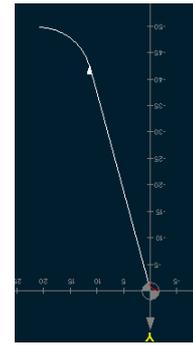
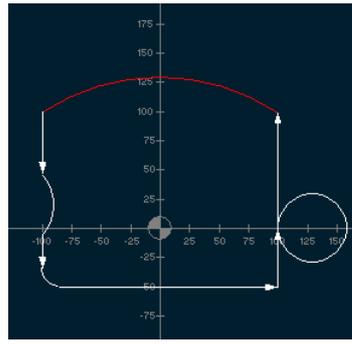
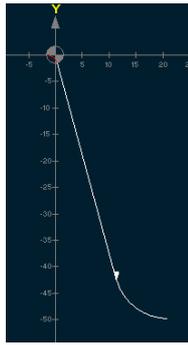
FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

Profile programmed with G841.

The section profile develops as would be seen by aligning its positive Y axis in the opposite direction to that of the plane profile (in this case, inside the plane profile).

**6.****PROGRAMMING CYCLES (MACROS).**

G8736/G8737/G8738. Milling of a surface defined by a plane profile and one or more section profiles.

FAGOR 
FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

6.3.2 Programming example (1)

Example that shows the different possibilities of defining the profiles (plane or section), as ISO and ProGTL3 languages, profile editor, etc.

Example:

%L 1 (Section profile - Definition in subroutine)

```
G841
G813 X0 Y0 J0
G821 I-10
G813 X0 Y0 J-85
G821 I10
G813 X0 Y-50 J0
G840
M29
```

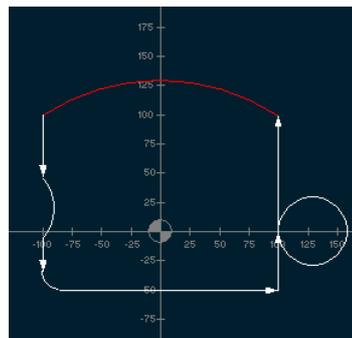
%PROG

```
G17
T1 M6
F1000 S1200
G0 Z100
X160 Y0
Z10
```

G8736 X160 Y0 I2 (declaration of macro and parameters)

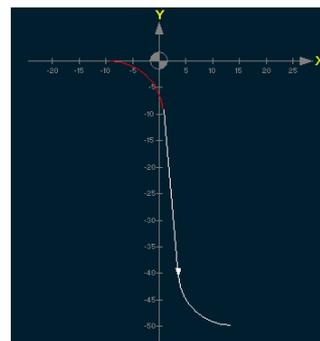
(Plane profile - Definition in ProGTL3)

```
G842 K2 X160 Y0
G813 X100 J90 H1
G820 X0 Y-50 I180 K2
G813 X-100 Y50 J-90 H2
G820 X-130 Y20 I-40 K1 H3
G813 X-100 Y0 J-90 K2
G821 I15 H4
G813 Y-50 J0
G813 X100 J90
G840 X160 Y0 K2
```



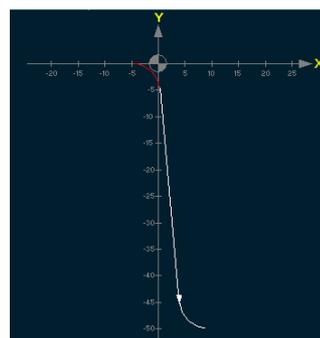
G8737 H1 (Section profile 1 - Edited in ProGTL3)

```
G841
G813X0Y0J0
G821I-10
G813X0Y0J-85
G821I10
G813X0Y-50J0
G840
```



G8737 H2 (Section profile 2 - Edited on the profile editor in ProGTL3)

```
(# PROFILE BEGIN) (0)
G20 X=1 Y=2
G841
G813X0Y0J0
G821I-5
G813X0Y0J-85
G821I5
G813X0Y-50J0
G840
(# PROFILE END)
```



6.

PROGRAMMING CYCLES (MACROS):
G8736/G8737/G8738. Milling of a surface defined by a plane profile and one or more section profiles.

G8736/G8737/G8738. Milling of a surface defined by a plane profile and one or more section profiles.



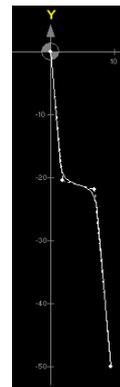
FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

G8737 H3 (Section profile 3 - Edited on the profile editor in ISO)

```
(# PROFILE BEGIN) (0)
G20 X=1 Y=2
G00 G90 X0 Y0 G41 G138 (-progtl3-G841)
(-progtl3-G813X0Y0J0)
G01 G90 X1.7918 Y-20.4801 (-progtl3-G813X0Y0J-85)
G36 I3 (-progtl3-G821I3)
G01 G90 X6.9118 Y-21.852 (-progtl3-G813X0Y-20J-15)
G36 I3 (-progtl3-G821I-3)
G01 G90 X9.3744 Y-50 (-progtl3-G813X5Y0J-85)
G40 (-progtl3-G813X0Y-50J0)
(-progtl3-G840)
(# PROFILE END)
```

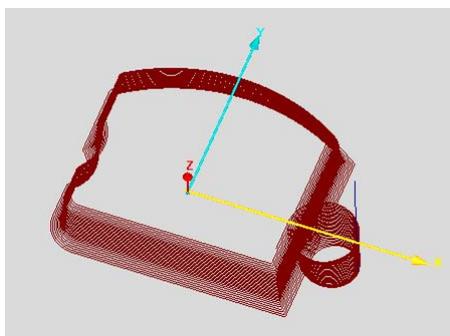


G8737 H4 (Section profile 4 - Edited in subroutine)

LL 1 (same section profile H1)

G8738 (activate the cycle)

```
G0 Z100
M30
```



6.

PROGRAMMING CYCLES (MACROS).

G8736/G8737/G8738. Milling of a surface defined by a plane profile and one or more section profiles.

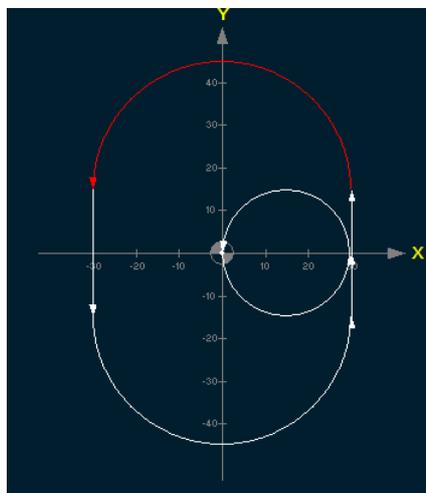
6.3.3 Programming example (2) G8736 using D7/D8/D9.

Example that shows the different possibilities of machining a profile depending on parameters D7, D8 y D9.

Example:

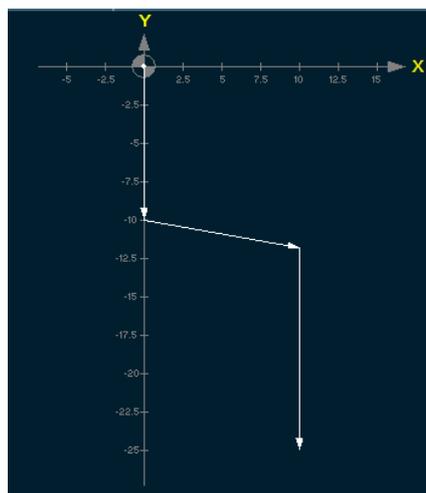
G8736 I0.5 X0 Y0 D7=0 D8=0 D9=0 (Plane profile)

```
G841 K2 X0Y0
G813 X30 J90
G820 X0 Y15 I30
G813 J270
G820 X0 Y-15 I30
G813 J90
G840 X0 Y0 K2
```

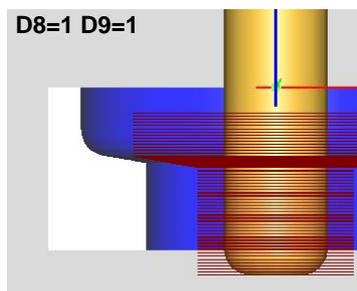
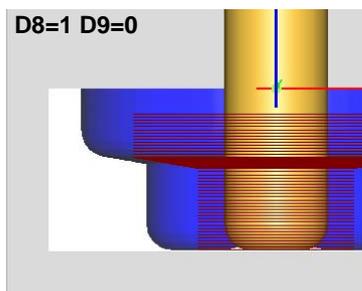
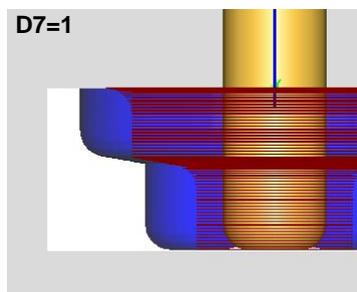
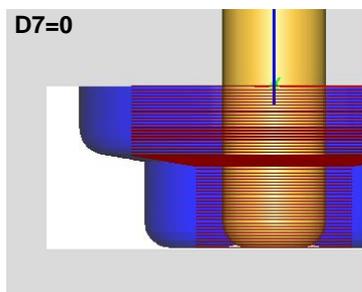


G8737 (Section profile)

```
G841
G820X0Y0
G813 X0J270
G813 Y-10 J-10
G813 X10 J270
G820Y-25
G840
```



G8738 (activate the cycle)
M30



6.

PROGRAMMING CYCLES (MACROS).

G8736/G8737/G8738. Milling of a surface defined by a plane profile and one or more section profiles.



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

6.4 G8754/G8753. Invert the direction of a profile.

The functions G8754 and G8753 allow for the machining direction of a profile to be inverted. The CNC stores the entire profile (where the maximum number of entities is 600) and begins machining from the end point to the starting point.

G8754 Invert the machining direction of the profile. Start of profile.

G8753 Invert the machining direction of the profile. End of the profile.

These functions may be used in the case where it is necessary for roughing the profile in one direction and finishing it in the opposite direction.

Programming.

Both instructions must be programmed alone in the block. The profile must be defined between both functions.

For example, the profile may be defined via a local subroutine at the beginning of the program. For roughing, the program calls the subroutine and executes the profile in the programmed direction. For finishing, the program calls the subroutine between functions G8754 and G8753 and executes the profile in the opposite direction to that programmed.

```
%program
..
..
; Roughing.
#CALL profile.nc
..
..
; Finishing.
G8754
#CALL profile.nc
G8753
..
..
M30
```

6.

PROGRAMMING CYCLES (MACROS).
G8754/G8753. Invert the direction of a profile.

6.5 G8777/G8778. Circular polygon pocket defined by a profile.

Functions G8777/G8778 allow for the machining of circular polygonal pockets defined by a profile. The functions for programming the cycle are the following.

- G8777 Define the cycle and its parameters.
- G8778 Activate the cycle.
- G8701 Profile pocket. Start of the profile and internal island profiles.
- G8702 Profile pocket. Profile section of the pocket and islands (optional).

Programming.

Programming format. G8777.

The format is the following:

G8777 Z.. J.. I.. [H..] [D2=..] D5=..

- Z.. Final coordinate of the pocket (machining depth).
- J.. Initial pocket coordinate.
- I.. Depth of pass.
- H.. Safety coordinate for rapid repositioning outside the part (if not programmed, H=J+2).
- D2=.. Machining allowance for finishing (default, D2=0).
- D5=.. Reduction coefficient for the Z feedrate.

Programming format. G8778.

The format is the following:

G8778 [X..] [Y..] [J..] D0=..

- X.. Y.. Starting point for the tool positioning (if not programmed, the tool is moved to the starting point of the first pass).
- J.. Milling pass (default, J=1.6)
- D0=.. Type of machining
 - D0=0: Roughing only for the pocket.
 - D0=1: Finishing only for the pocket.

Programming format. G8701.

The format is the following:

G8701 X.. Y.. [D6=..]

- X.. Y.. Initial and final point.
- D6=.. Slope of walls (default is vertical wall, D6 = 0)

Programming format. G8702.

The format is the following:

G8702 X.. Y..

- X.. Y.. Initial and final point.

Note: Optional vertical profile, alternative to parameter D6 of G8701.

6.

PROGRAMMING CYCLES (MACROS):
G8777/G8778. Circular polygon pocket defined by a profile.

FAGOR 

FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

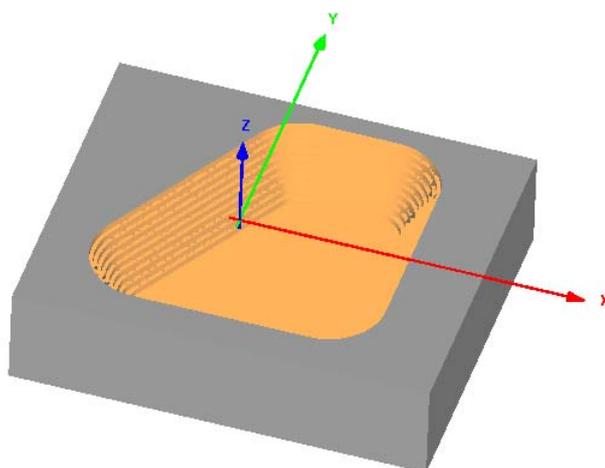
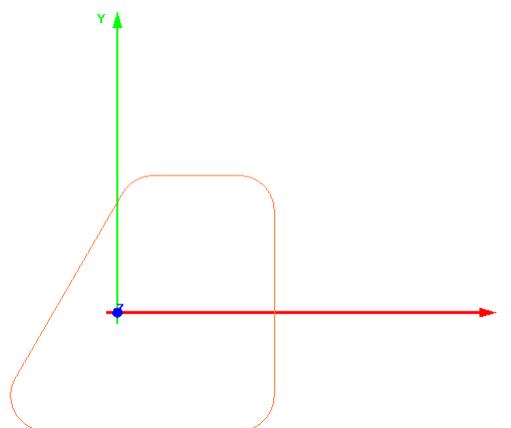
6.5.1 Programming example.

```

G17 G90 T2 D1 M6
F1000 S2000 M3
G0 X0 Y-30
G8777 Z-20 I2 J0 D5=0.9

G8701 X0 Y-30 D6=30
G842 K2
G813 Y-40 J180
G821 I-18
G813 X-50 Y-40 J60
G821
G813 Y45 J0
G821
G813 X50 J-90
G821
G813 Y-40 J180
G840 X0 Y-30 K2
G8778 D0=0
G00 Z100
M30

```



6.

PROGRAMMING CYCLES (MACROS).

G8777/G8778. Circular polygon pocket defined by a profile.

FAGOR 
FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

7.1 Generic user subroutines (G8000-G8999).



If this version is updated, the CNC behaves as follows.

- *In the first two installation options, the CNC only updates the subroutines if they do not already exist.*
- *In the third installation option, "rename previous version and install it completely", the CNC always updates the subroutines.*

The CNC lets the user define up to 1000 subroutines, common to all channels and associated with functions G8000 through G8999, so when the CNC executes one of these functions, it will execute its associated subroutine. These subroutines may be executed either in a non-modal or modal way and it also allows resetting the local parameters of the subroutine.

These subroutines are loaded into RAM memory when they are executed for the first time. If there is no room in RAM, the CNC will issue a warning and it will execute the subroutine from the disk. At the end of the program (M30), if no other channel is executing subroutines, the CNC deletes them from its RAM memory. This way, if a user subroutine is edited or modified, the CNC assumes the changes the next time it executes it.

Some of these subroutines have a function pre-assigned by Fagor, being ProGTL3 language macros. If one of these G functions is programmed, the CNC executes the corresponding macro and not the subroutine.

Programming format.

These functions may be programmed anywhere in the program and they don't have to go alone in the block and allow resetting the local parameters of the subroutine.

Programming format. Executing the subroutine in a non-modal way.

The programming format is the following; the list of arguments appears inside the curly brackets. To execute the subroutine in a non-modal way, call it using the G code (G8000,G8001, etc).

```
G8000
G8000 {P0..Pn}
P0..Pn      Optional. Local parameters of the subroutine.
```

```
G8000
G8083 P1=12.3 P2=6
G8088 A12.3 B45.3 P10=6
```

Programming format. Executing the subroutine in a modal way.

The programming format is the following; the list of arguments appears inside the curly brackets. To execute the subroutine in a modal way, call it using the MG code (MG8000, MG8001, etc).

```
MG8000
MG8000 {P0..Pn}
P0..Pn      Optional. Local parameters of the subroutine.
```

```
G8000
G8088 A12.3 B45.3 P10=6
```

How to set local parameters.

These functions allow initializing local parameters in the subroutine. The parameter values may be defined after the calling function and may be defined in two ways; using the parameter numbers (P0, P1, etc.) or their letters A-Z (except "Ñ" and "Ç") like "A" for P0 and "Z" for P25. Both ways of setting local parameters are equivalent and may be combined in the same block.

Additional data in the block.

Besides initializing the parameters, any other type of additional information may be added to these functions, even movements. This information must be programmed before the subroutine calling function; otherwise, the data will be considered as for initializing the parameters. The associated subroutine is executed once the execution of the rest of the information programmed in the block has ended.

```
G01 X50 F450 G8000 P0=15 P1=20
```

It first executes the programmed movement to point X50 and then the subroutine associated with G8000 initializing parameters P0 and P1.

```
G8000 P0=15 P1=20 G01 X50 F450
```

All the data is interpreted as parameter setting, where P6(G)=1, P23(X)=50 and P5(F)=450.

Cancelling a modal subroutine.

A subroutine stops being modal and in the following cases.

- When programming G80 or #MDOFF.
- After executing an M02 or an M30 and after a RESET.

Define the subroutines.

The subroutines associated with these functions will be global and will have the same name as the function, without extension. The subroutines must be defined in the folder ..\Users\Sub. If the CNC executes a function and there is no subroutine, the CNC will issue an error message.

```
G8000      The subroutine G8000 will be associated.
G8001      The subroutine G8001 will be associated.
...
G8099      The subroutine G8099 will be associated.
```

Execute the subroutines.

The CNC executes the subroutine after executing the function with which they are associated. To execute the subroutine as a single block, program the instructions #ESBLK and #RETDSBLK. After executing the instruction #ESBLK, the CNC executes the blocks programmed next as a single block until it reaches the end of the subroutine (#RETDSBLK).

If the file that contains the subroutine has the "hidden" attribute, the CNC does not display the contents of the subroutine while it is executed. The attributes of the files may be modified from the utilities mode (see the operating manual).

Local parameter nesting levels.

If these functions initialize local parameters, this instruction generates a new nesting level for the local parameters. Remember that up to 7 parameter nesting levels are possible within 20 subroutine nesting levels.

7.

USER SUBROUTINES.
Generic user subroutines (G8000-G8999).



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

Properties of the function and Influence of the reset, turning the CNC off and of the M30 function.

Functions G8000-G8999 are not modal. Functions MG8000-MG8999 are modal.

7.

USER SUBROUTINES.

Generic user subroutines (G8000-G8999).



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

7.2 Assistance for subroutines.

7.2.1 Subroutine help files.

Help files may be associated with each OEM subroutine (G180, G380, etc), user subroutine (G500, G800, etc) and each global subroutine called upon using #MCALL or #PCALL and they will be displayed while editing.

The help window is displayed while editing, after a blank space or tab following a G function or following the name of the subroutine. The help window is only informative, it cannot be accessed with the cursor nor browse through it. The help window closes with [ESC], deleting the key word or going on to another line of the program.

The help window of the subroutines is only available when the editor uses the CNC language; when the editor is enabled for the 8055 CNC, these helps are not available. The help window of the subroutines is available even when the contextual helps of the editor are disabled.

When the help file is displayed, its text may be inserted into the part-program using the [INS] key.

Edit the help files.

Each subroutine may have two help files; a text file (txt) and an image file (bmp). There's no need to define both files; either one may be defined alone.

When the help window is only informative, it cannot be accessed with the cursor nor browse through it with the page-up-down keys. This is why it is recommended to use short help files; for example, that only contain the description of the parameters of the subroutine. Also, since the text of the help file can be inserted into the program ([INS] key), the following is recommended.

- That the help file contains the line calling the subroutine. Since the user must have written part of the call to display the help window, the editor deletes the call before inserting the help text.
- That all the lines of the help file follow the format of a CNC comment, except the line containing the call to the subroutine.

```
Help file example for a subroutine.
G180 P0= P1= P2= P3= P4= P5=
#COMMENT BEGIN
----- G180 -----
P1 = Movement in X
P2 = Movement in Y
P3 = Movement in Z
P4 = Feedrate F
P5 = Speed S
-----
#COMMENT END
```

Name and location of files.

Name of the help files.

The name of the files must follow the following rule:

Subroutine.	Name of the help files.
G8000-G8999	The name of the files must be the function it is associated with. For example <i>G8000.txt</i> and <i>G8000.bmp</i> .

7.

USER SUBROUTINES. Assistance for subroutines.

Where to save the help files.

The machine manufacturer can save the help files in the folders `..\Mtb\Sub\Help` and `..\Mtb\Sub\Help\{language}`. Since the modifications to the MTB directory in the "User" work mode disappear when turning the unit off, the user must save his help files in the folder `..\Users\Sub\Help` and `..\Users\Sub\Help\{language}`.

The CNC looks for the files in the following order and displays the first that it finds, this is the reason why it is recommended that the user does not define subroutines and/or help files with the same name as those of the OEM. If there are no help files, the CNC will not show any help and it will not display an error.

```
..\Users\Sub\Help\{idioma}  
..\Users\Sub\Help  
..\Mtb\Sub\Help\{idioma}  
..\Mtb\Sub\Help\
```



For versions V1.60 (8060) y V5.60 (8065), the CNC no longer searches for help files in the following folders.

```
..\Users\Session\Help\{idioma}  
..\Mtb\Sub\Help\{idioma}.  
..\Users\Help\{idioma}.
```



For versions prior to V1.60 (8060) and V5.60 (8065), the CNC first searched for the help files in the manufacturer folders and then in the user folders. For these and later versions, it does so in the opposite order.

7.

USER SUBROUTINES.

Assistance for subroutines.



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

7.2.2 List of available subroutines.

The editor allows having in a text file (txt) a list that is displayed while editing the part-program, every time a #PCALL or an #MCALL instruction is edited.

The editor shows the list of subroutines while editing, after a blank space or tab following a G180-G189 or following a #PCALL or #MCALL instruction. This list works the same way as the lists of variables, it is possible to use the arrow keys to scroll through the various elements. Pressing [ENTER], the editor inserts the selected line in the current position of the cursor. The list of subroutines disappears with [ESC], deleting the key word or going on to another line of the program.

This help is always active, regardless of the editing help softkey, "Prog. help".

List of subroutines.

The list of subroutines must be in a text (txt) file. The file must be edited so each line is the name of a possible subroutine to be called.

Example of a file with a list of subroutines.

```
C:\CNC8070\USERS\SUB\FAGOR.NC
SUBROUTINE.NC
EXAMPLE.NC
POSITIONING.NC
```

Name and location of files.

The name of the file should be *pcall.txt*.

Where to save the list of subroutines.

The machine manufacturer must save the *pcall.txt* file in the folder *..\Mtb\Sub\Help*. Since the modifications to the MTB directory in the "User" work mode disappear when turning the unit off, the user must save his *pcall.txt* file in the folder *..\Users\Sub\Help*.

The CNC looks for the help files in both files; if the files are not there, the CNC will not show any help. If there is a *pcall.txt* file in both directories, the list will show the names of the subroutines contained in both of them.



For versions V1.60 (8060) y V5.60 (8065), the CNC no longer searches for help files in the following folders.

```
..\Users\Session\Help\{idioma}
..\Mtb\Sub\Help\{idioma}.
..\Users\Help\{idioma}.
```



FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

Some types of auxiliary functions may be inserted into a ProGTL3 profile. Auxiliary functions may be programmed alone in the block or may be added to profile defining blocks. These auxiliary functions will be executed when executing the movement programmed in the same blocks.

The possible auxiliary functions are the following:

- F... Work feedrate.
- S... Spindle turning speed.
- M... Call to an M function of the PLC.
- H... Call to an H function of the PLC.

F..., S...

These functions may be written in the ProGTL3 block as well as in a single block. If the instruction is written in the ProGTL3 block, it is applied at the beginning of the movement indicated in the block.

M..., H...

These functions may be written in the ProGTL3 block as well as in a single block. If the command is written in the ProGTL3 block, it will be executed by the PLC and it will be applied before or after the movement depending on the type of M and on how the CNC has been configured by the machine manufacturer (OEM).

G91

To program in incremental coordinates. The effect of G91 on ProGTL3 functions is described, case by case in each G of the ProGTL3.

8.

AUXILIARY FUNCTIONS.

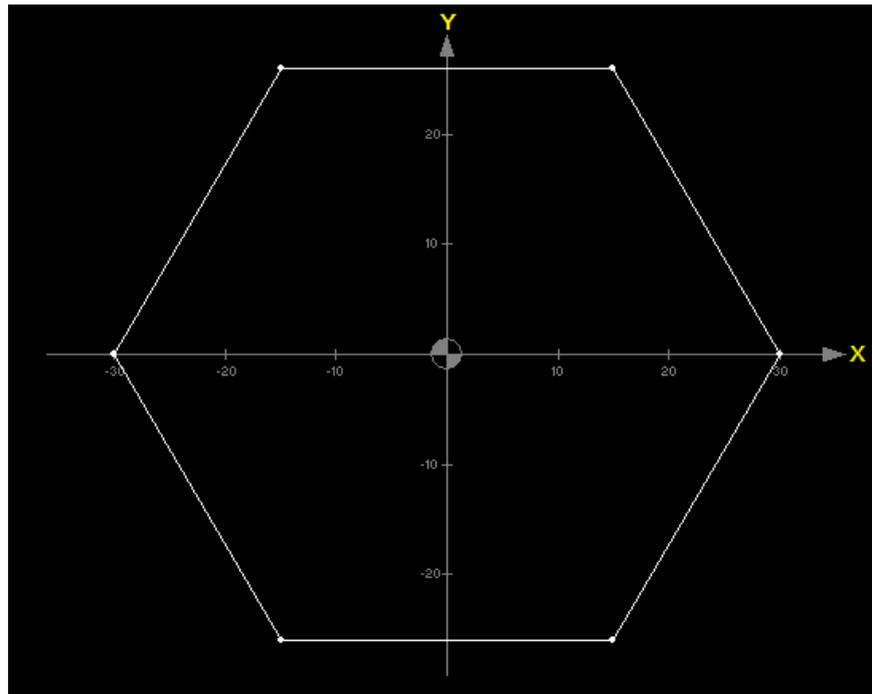


FAGOR AUTOMATION

CNC 8060
CNC 8065

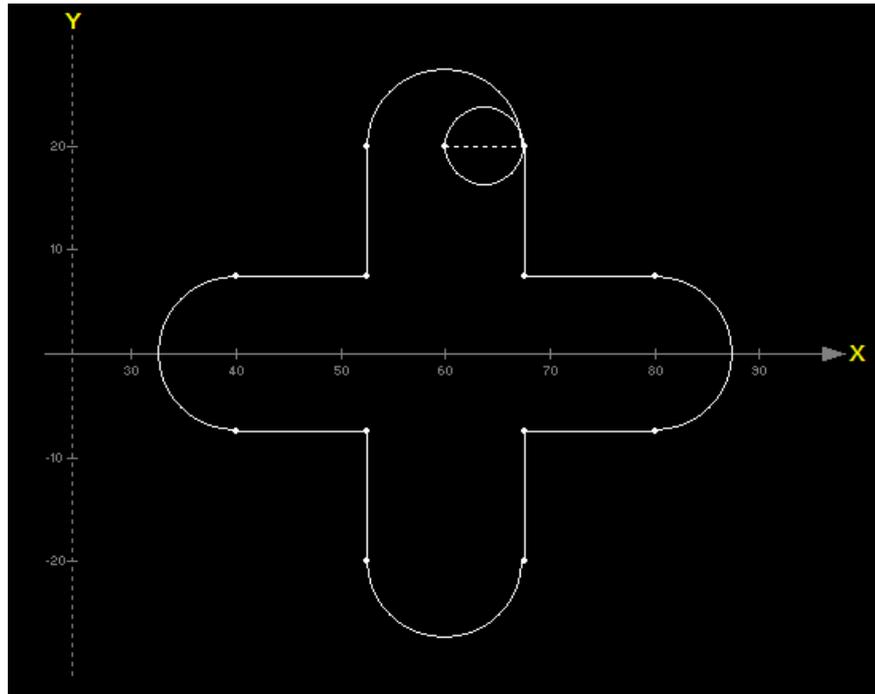
(REF: 1709)

Example 1.



```
G0 X30 Y0  
G841 K1  
G810 X30 Y0 ;1  
G811 Q60 R30 ;2  
G811 Q120 R30 ;3  
G811 X-30 Y0 ;4  
G811 Q-120 R30 ;5  
G811 Q-60 R30 ;6  
G811 X30 Y0 ;7  
G840 Q0 R30 K1
```

Example 2.

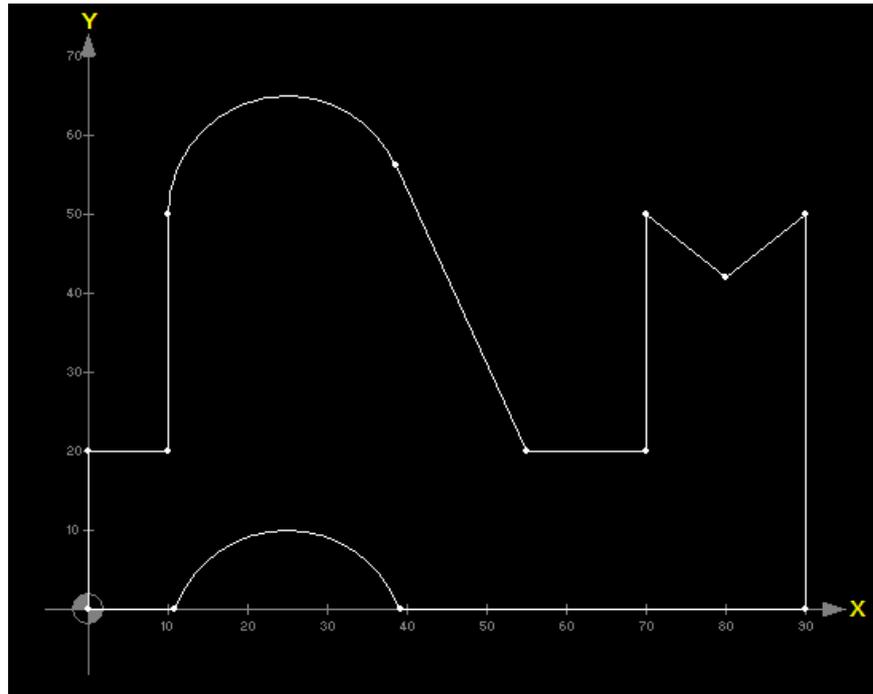


```

G851 X60 Y0
GO X0 Y20
G841 K2 I3.75
G820 I7.5
G813 X-7.5 J-90
G813 Y7.5 J180
G820 X-20 Y0 I7.5
G813 Y-7.5 J0
G813 X-7.5J-90
G820 X0 Y-20 I7.5
G813 X7.5 J90
G813 Y-7.5J0
G820 X20 Y0 I7.5
G813 Y7.5 J 180
G813 X7.5 J90
G840 X0 Y20 K2 I3.75

```

Example 3.



```
G0 X0 Y0
G841 K1
G813 Y0 J0
G820 X25 Y-5 I-15
G813 Y0 J0 K2
G813 X90 J90
G810 X90 Y50
G811 X80 Y42
G811 X70 Y50
G813 X70 J-90
G813 Y20 J180
G810 X55 Y20
G811 X25 Y50 I15
G820 X25 Y50 I15
G813 X10 J-90
G813 Y20J180
G813 X0J-90
G840 X0 Y0
```

9.

PROGRAMMING EXAMPLES.

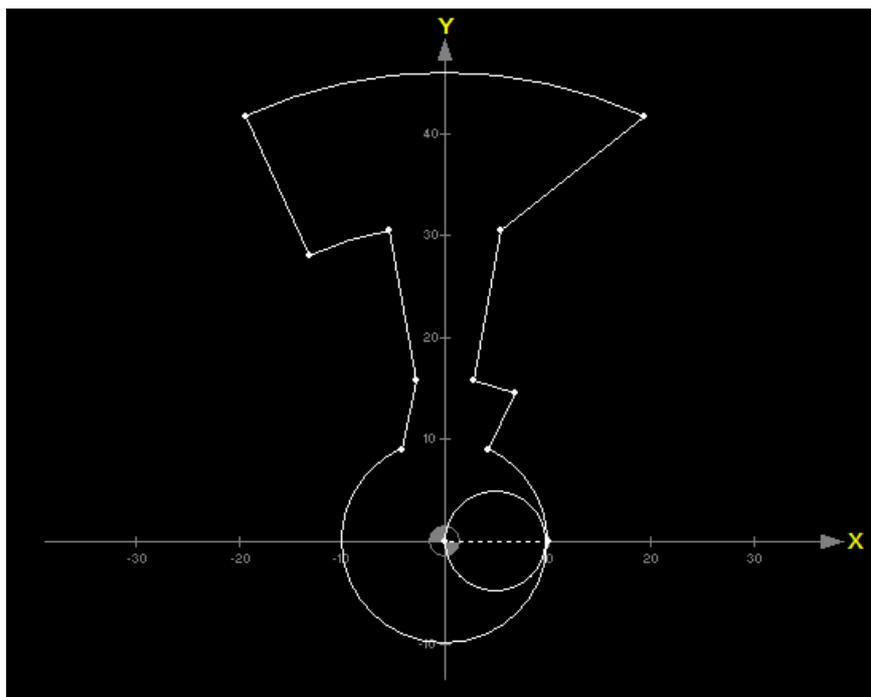
FAGOR 

FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

Example 5.



```
G0 X0 Y0
G841 K2
G820 I10
G813 X0 Y0 J65 K2
G810 Q65 R16
G811 Q80 R016
G813 Q80 R31 J80
G810 Q80 R31
G811 Q65 R46
G820 X0 Y0 I46 K2
G813 Q115 R46 J115 K2
G820 X0 Y0 I-31
G813 Q100 R31 J100 K2
G810 Q100 R16
G811 Q115 R10
G820 X0 Y0 I10
G840 X0 Y0 K2
```

9.

PROGRAMMING EXAMPLES.

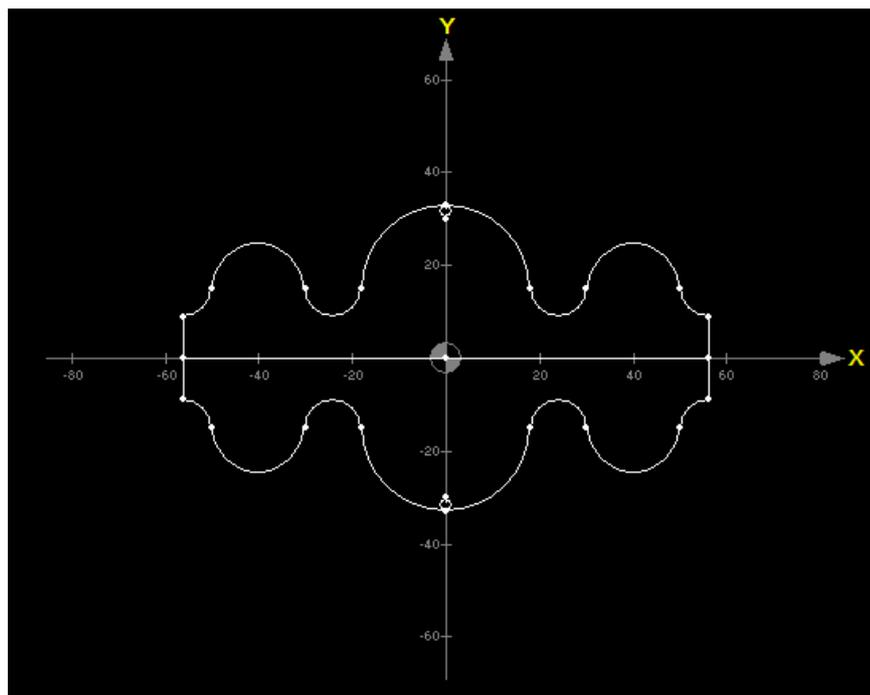
FAGOR 

FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

Example 6.

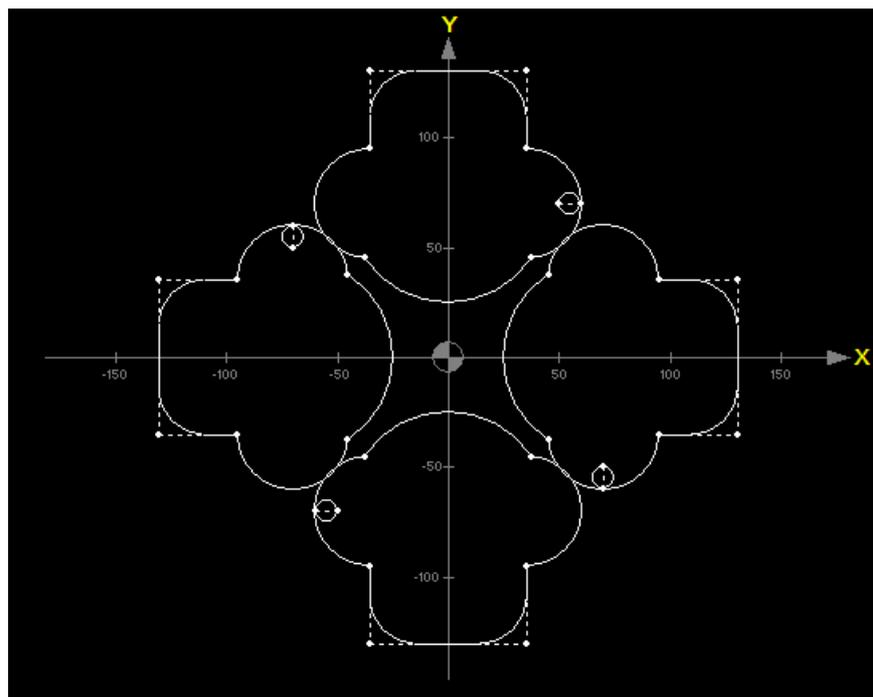


```

G0 X0 Y0
(-----)
G851 X0 Y15
G0 X0 Y15
G841 K2
G820 X0 Y0 I18
G820 X-24 Y0 I-6
G820 X-40 Y0 I10
G820 X-56 Y0 I-6
G813 X-56 J-90 K2
G813 Y-15 J0
G813 X56 J90
G820 X56 Y0 I-6
G820 X40 Y0 I10
G820 X24 Y0 I-6
G820 X0 Y0 I18
G840 X0 Y15 K2
(-----)
G851 X0 Y-15 J180
G0 X0 Y15
G841 K2
G820 X0 Y0 I18
G820 X-24 Y0 I-6
G820 X-40 Y0 I10
G820 X-56 Y0 I-6
G813 X-56 J-90 K2
G813 Y-15 J0
G813 X56 J90
G820 X56 Y0 I-6
G820 X40 Y0 I10
G820 X24 Y0 I-6
G820 X0 Y0 I18
G840 X0 Y15 K2

```

Example 7.



```

G851 X0 Y70
G0 X50 Y0
G841 K2
G820 X35 Y0 I25
G813 X35 Y25 J90 K2
G821 I20
G813 X0 Y60 J180
G821 I20
G813 X-35 Y25 J-90
G820 X-35 Y0 I25
G820 X0Y0 I45 K2
G820 X35 Y0 I25 K2
G840 X50 Y0 K2
(-----)
G851 X-70 Y0 J90
G0 X50 Y0
G841 K2
G820 X35 Y0 I25
G813 X35 Y25 J90 K2
G821 I20
G813 X0 Y60 J180
G821 I20
G813 X-35 Y25 J-90
G820 X-35 Y0 I25
G820 X0Y0 I45 K2
G820 X35 Y0 I25 K2
G840 X50 Y0 K2
(-----)
G851 X0 Y-70 J180
G0 X50 Y0
G841 K2
G820 X35 Y0 I25
G813 X35 Y25 J90 K2
G821 I20
G813 X0 Y60 J180
G821 I20
G813 X-35 Y25 J-90
G820 X-35 Y0 I25
G820 X0Y0 I45 K2
G820 X35 Y0 I25 K2
G840 X50 Y0 K2
(-----)
G851 X70 Y0 J-90
G0 X50 Y0
G841 K2
G820 X35 Y0 I25
G813 X35 Y25 J90 K2
G821 I20
G813 X0 Y60 J180
G821 I20
G813 X-35 Y25 J-90
G820 X-35 Y0 I25
G820 X0Y0 I45 K2
G820 X35 Y0 I25 K2
G840 X50 Y0 K2
    
```

9.

PROGRAMMING EXAMPLES.

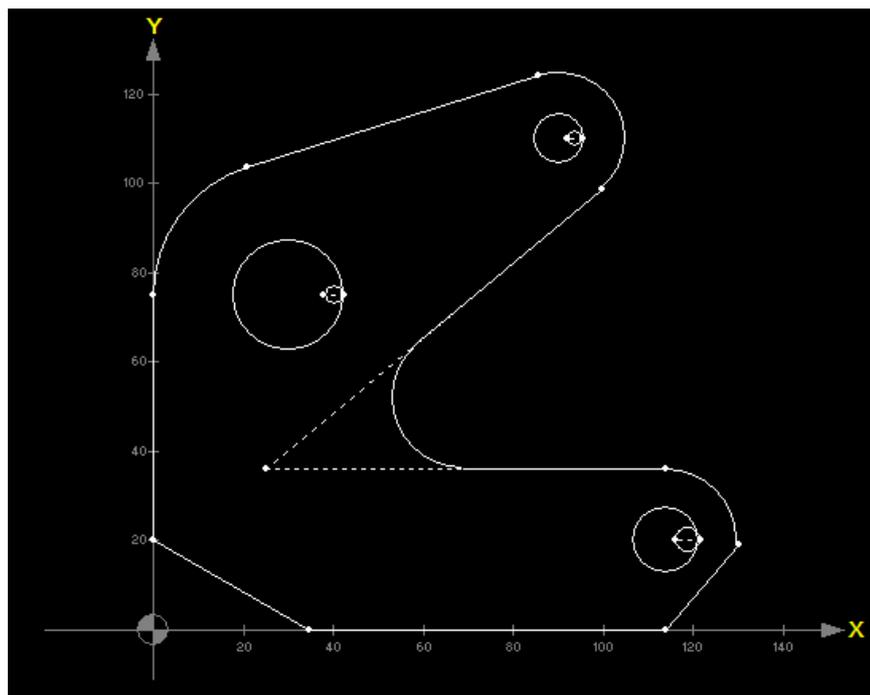
FAGOR 

FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

Example 8.



```

;
G0 X0 Y20
G841
G813 X0 J90
G820 X30 Y75 I-30
G811 X90 Y110 I-15
G820 X90 Y110 I-15
G813 X90 Y110 I-15 J220
G821 I16
G813 Y36 J0
G820 X114 Y20 I-16
G813 X114Y0 J230
G813 Y0 J180
G813 X0 Y20 J150
G840 X0 Y20
;
G0 X38 Y75
G841 K2
G820 X30 Y75 I12.5
G840 X38 Y75 K2
;
G0 X92 Y110
G841 K2
G820 X90 Y110 I5.5
G840 X92 Y110 K2
;
G0 X116 Y20
G841 K2
G820 X114 Y20 I7.5
G840 X116 Y20 K2

```

9.

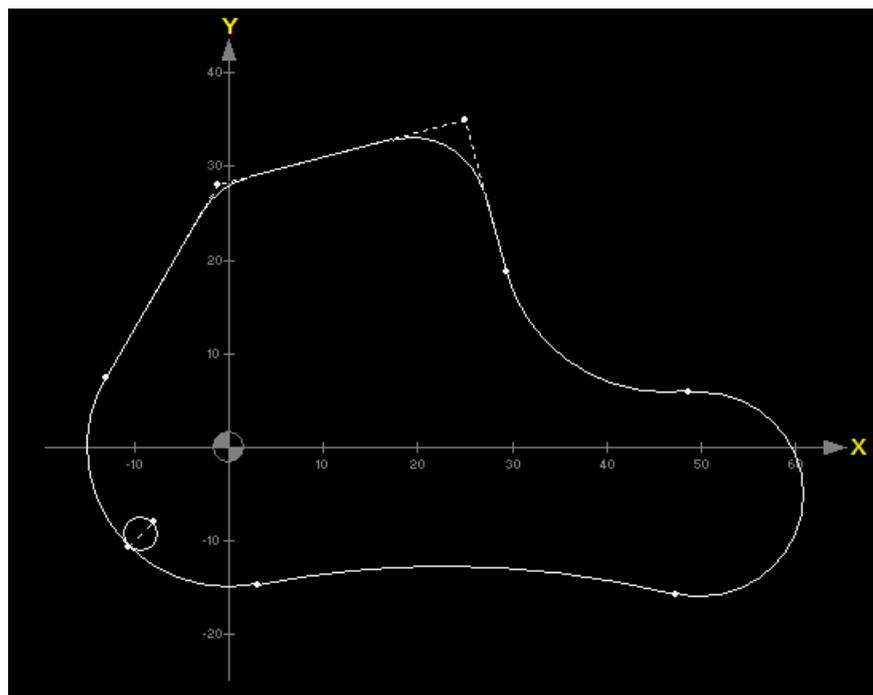
PROGRAMMING EXAMPLES.

FAGOR 
FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

Example 9.



```
G0 X-8 Y-8  
G841 K2  
G820 X0 Y0 I15  
G821 I-100  
G820 X50 Y-5 I11  
G821 I-17.5  
G813 X25 Y35 J-75  
G821 I8  
G813 X25 Y35 J195  
G821 I8  
G813 X0 Y0 I15 J240  
G820 X0 Y0 I15  
G840 X-8 Y-8 K2
```

9.

PROGRAMMING EXAMPLES.

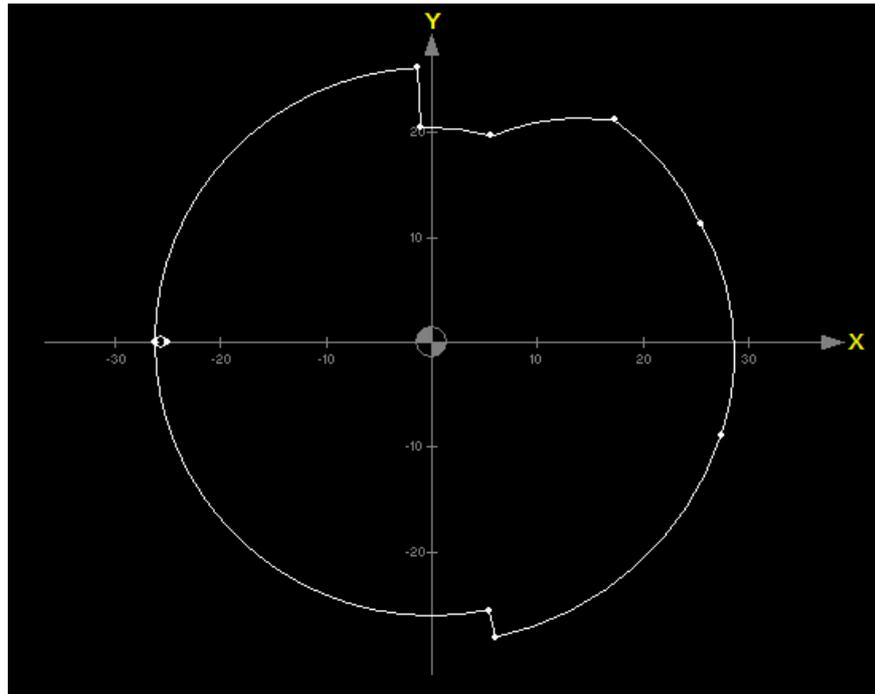
FAGOR 

FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)

Example 10.



```

G0 X-25 Y0
G841 K2
G820 X0 Y0 I26.25
G813 X0 Y0 J-78 K2
G820 X0 Y0 I28.85
G820 X3.1385 Y-1.0198 I25.55
G820 X3.7974 Y1.6907 I23.75 K2
G820 X14.0917 Y0 I21.44
G820 X0 Y0 I20.5 K2
G813 X0 Y0 J93 K2
G820 X0 Y0 I26.25
G840 X-25 Y0 K2

```

9.

PROGRAMMING EXAMPLES.

FAGOR 
FAGOR AUTOMATION

CNC 8060
CNC 8065

(REF: 1709)



FAGOR AUTOMATION

Fagor Automation S. Coop.

Bº San Andrés, 19 - Apdo. 144
E-20500 Arrasate-Mondragón, Spain

Tel: +34 943 719 200

+34 943 039 800

Fax: +34 943 791 712

E-mail: info@fagorautomation.es

www.fagorautomation.com