CNCelite

8058/8060
8065

Examples manual (·T· model).

Ref. 2109

FAGOR

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

FAGOR AUTOMATION

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

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

BLANK PAGE

BASIC CONCEPTS.

1

Purpose of the exercises.

The purpose of the following programming examples is to familiarize with editing, simulating and executing programs. Machining starts with raw piece that after running various operations and cycles on it, it becomes the finished part, programming also the relevant machining conditions as well as the tools to be used.

The feedrate and spindle speed values are only illustrative and depend on the material of the part and the tool being used. When using the examples of this manual to make real parts (on a machine), the feedrate and spindle speed values must be adapted properly.

1.1 Basic CNC operating concepts

Some useful keys.

Some operating modes.

Key.	Function.
AUTO	Automatic mode. Execute a part-program in "single block" or "automatic" mode.
EDIT	EDISIMU mode. Edit and simulate the execution of a part-program displaying a graphic representation of the program being simulated.
TABLES	User tables (zero offsets, fixtures and arithmetic parameters).
TOOLS	Tool and magazine table.

Execution keys.

Key.	Function.
	Cycle start key (START). Execute the selected program in automatic mode, a block in MDI/MDA mode, etc.
	Cycle stop key (STOP). Interrupt the execution of the CNC.
RESET	Reset key. It initializes the system setting the initial conditions as defined by machine parameters.
SINGLE	Single-block execution mode.

Editing a program.

EDIT

Programs are edited in the EDISIMU mode. Once in this mode, the "Open program" softkey may be used to select the program to be edited which may be either a new one or an existing one. When selecting this option, the CNC shows a list of the available programs.

To select a program from the list:

1. Select the folder that contains the program. If it is a new program, it will be saved in this folder.
2. Select the program from the list or write its name in the bottom window. To edit a new program, write the name of the program in the lower window and the CNC will open an empty program or a predefined template depending on how the editor is configured.
3. Press [ENTER] to accept the selection and open the program or [ESC] to cancel it and close the program listing.

Syntax check.

The CNC analyzes each program block while editing. If the CNC detects a syntax error in the block, the error window at the bottom of the screen will display it.

The entire program can also be checked. To do that, press the vertical softkey for syntax check. The errors found will be indicated like with the other method.

Program simulation.

The program simulation procedure is the following:

1 Choose the type of graphic representation, its dimensions and the point of view. This data may also be modified during the simulation of the program.

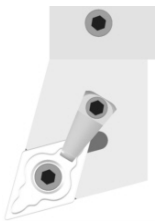
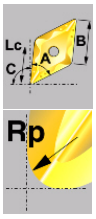
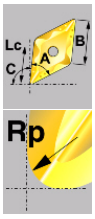
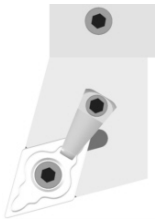
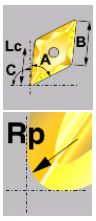
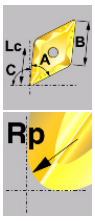
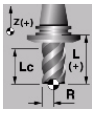
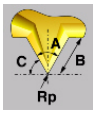
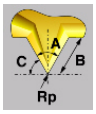
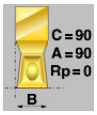
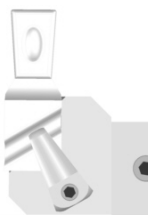
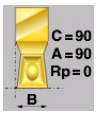
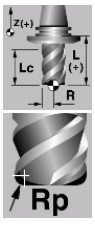
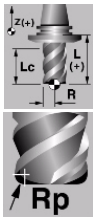
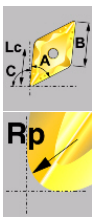
2 Activate the desired simulation options using the softkey menu.

3 Pressing the [START] softkey begins the simulation of the program that is being edited. The simulation may be interrupted with the [STOP] softkey or canceled with the [RESET] softkey.

The simulation of the program starts at the first block of the program and ends after executing one of the end-of-program functions "M02" or "M30". As an option, it is possible to define the first and last blocks of the simulation.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

1.2 Tools used in the examples.

Tool	Geometry	Data	Tool	Geometry	Data
T2 		D: 1 F: 3 A: 60º B: 7 mm. C: 100º Lc: 6 mm. Rp: 0.4 mm.	**T3** 		D: 1 F: 2 A: 60º B: 7 mm. C: 60º Lc: 6 mm. Rp: 0.2 mm.
T4 		D: 1 F: 3 A: 30º B: 7 mm. C: 100º Lc: 6 mm. Rp: 0.4 mm.	**T8** 		D: 1 F: 5 A: 60º B: 7,5 mm. C: 100º Lc: 6 mm. Rp: 0.4 mm.
T9 		D: 1 L: 100 mm. R: 10 mm. Lc: 10 mm. Rp: 0 mm.	**T10** 		D: 1 F: 5 A: 50º B: 5 mm. C: 65º Lc: 5 mm. Rp: 0.1 mm.
T11 		D: 1 F: 2 A: 50º B: 5 mm. C: 65º Lc: 5 mm. Rp: 0.1 mm.	**T12** 		D: 1 F: 3 A: 90º B: 4 mm. C: 90º Lc: 4 mm. Rp: 0 mm.
T13 		D: 1 F: 6 A: 90º B: 4 mm. C: 90º Lc: 4 mm. Rp: 0 mm.	**T15** 		D: 1 L: 40 mm. R: 5 mm. Lc: 10 mm. Rp: 0 mm.
T16 		D: 1 L: 40 mm. R: 5 mm. Lc: 5 mm. Rp: 0 mm.	**T17** 		D: 1 F: 2 A: 40º B: 7 mm. C: 70º Lc: 6 mm. Rp: 0.2 mm.

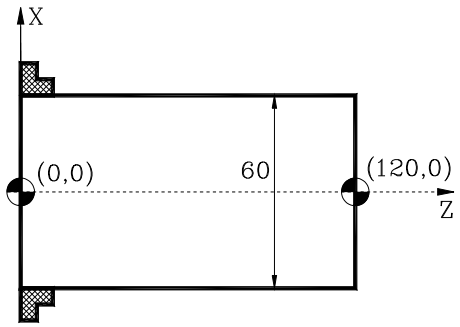
1.3 Set part zero.

With this CNC, it is possible to program movements in the machine reference system or apply offsets in order to use reference systems referred to the fixtures or the part without having to change the coordinates of the different points of the part in the program.

The part zero must be located so as to make programming easier. If no part zero is set, the coordinates will be referred to the machine reference system.

Coordinate preset (G92).

When presetting a coordinate, the CNC interprets that the axis coordinates programmed after the G92 set the current position of the axes. The rest of the axes that have not been defined with G92 are not affected by the preset.




```
G90 G00 X32 Z120          ; Tool approach
G01 X0                    ; Facing and positioning in (120,0)
G92 X0 Y0                 ;Presetting (120,0) as part zero
```

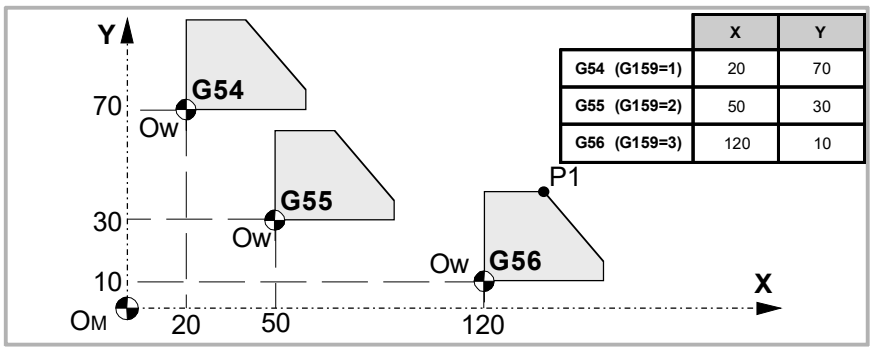
Zero offsets (G54-G59/G159).

Using zero offsets, it is possible to place the part zero in different positions of the machine. When applying a zero offset, the CNC assumes as the new part zero the point defined by the selected zero offset.

In order to apply a zero offset, it must have been previously defined. To do that, the CNC has a table where the operator may define up to 99 different zero offsets. The table data may be defined:

- Manually from the CNC's front panel (as described in the Operating Manual).
- By program, assigning the corresponding value (of the "n" offset and of the "Xn" axis) to the "V.A.ORGT[n].Xn" variable.

Once the zero offsets have been defined in the table, they may be activated via program by programming function G59 followed by the offset number to be activated. The first six zero offsets of the table can also be applied using functions G54 through G59; G54 for the first one (same as G159=1), G55 for the second one (same as G159=2) and so on.



	X	Y
G54 (G159=1)	20	70
G55 (G159=2)	50	30
G56 (G159=3)	120	10

```
N100 V.A.ORGT[1].X=20   V.A.ORGT[1].Y=70
N110 V.A.ORGT[2].X=50   V.A.ORGT[2].Y=30
N100 V.A.ORGT[3].X=120   V.A.ORGT[3].Y=10
...
N100 G54
     (It applies the first zero offset)
N200 G159=2
     (It applies the second zero offset)
N300 G56 X20 Y30
     (It applies the third zero offset)
     (The axes move to point X20 Y30 (point P1) referred to the third origin)
```

Cancellation of the part zero (G53).

The part zero stays active until it is canceled with a preset, a zero offset or with a "G53".

1.4 Programming of the machining conditions.

Feedrate programming units (G94/G95).

G94 Feedrate in millimeters/minute (inches/minute).

The feedrate is independent from the spindle speed.

G95 Feedrate in millimeters/revolution (inches/revolution).

The feedrate changes with the spindle speed (usual operation on lathes).

By default the type of feedrate is set in machine parameter IFEED.

Speed programming units (G96/G97).

G96 Constant surface speed.

The G96 function only affects the master spindle of the channel. At constant surface speed, the CNC changes the spindle speed as the perpendicular axis moves in order to maintain the cutting speed constant between the tool and the part, thus optimizing the machining conditions. When working at constant surface speed, it is recommended to limit by program the maximum turning speed that the spindle can reach.

G97 Constant turning speed.

Turning speed limit

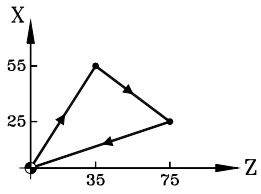
Function G192 limits the spindle turning speed in both work modes; G96 and G97. This function is especially useful when working at constant cutting speed while machining large parts or when doing spindle maintenance work. If function G192 is not programmed, the turning speed is limited by machine parameter G00FEED of the gear.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

1.5 Programming coordinates.

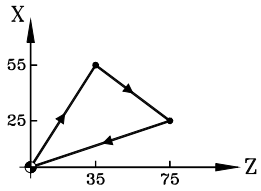
Absolute (G90) or incremental (G91) coordinates.

G90 Programming in absolute coordinates. The coordinates of the point are referred to the current origin of the coordinate system, usually the part zero.




```
N10 G00 G71 G90 Z0 X0
N20 G01 Z35 X55 F450
N30 Z75 X25
N40 X0 Z0
N50 M30
```

G91 Programming in incremental coordinates. The coordinates of the point are referred to the current tool position.




```
N10 G00 G71 G90 Z0 X0
N20 G01 G91 Z35 X55 F450
N30 Z40 X-30
N40 Z-75 X-25
N50 M30
```

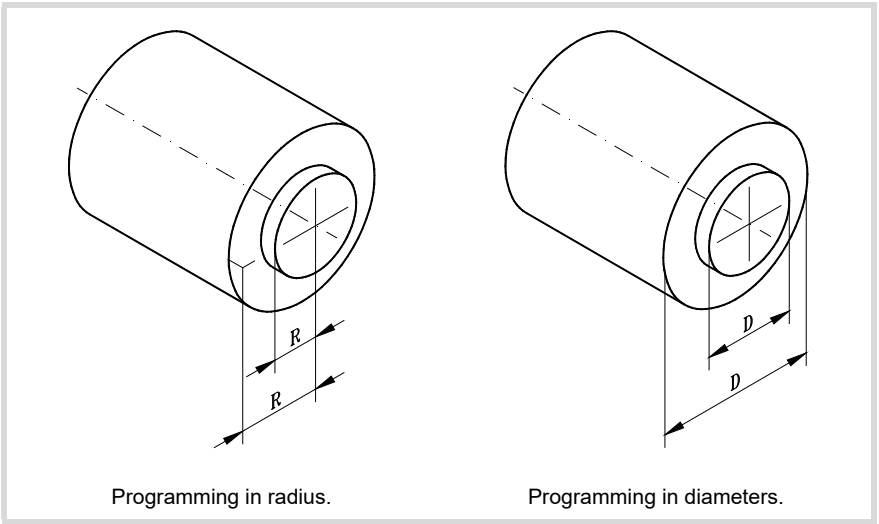
By default the type of feedrate is set in machine parameter ISYSTEM.

Programming in radius (G152) or in diameters (G151).

Programming in diameters is only available on the axes allowed by the machine manufacturer (DIAMPROG=YES).

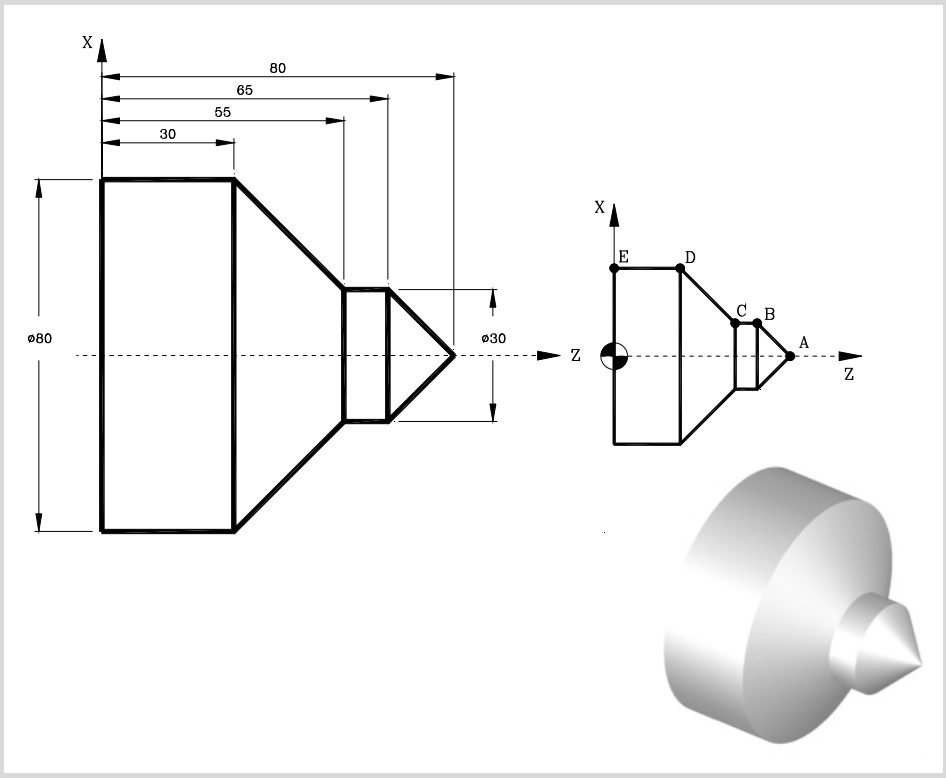
G151 Programming in diameters.

G152 Programming in radius.



Programming in radius. Programming in diameters.

1.5.1 Example. Absolute and incremental coordinates.



Programming in radius.

Absolute coordinates (G90).	Incremental coordinates (G91).
G90 G95 G96 F0.15 S180 T2 D1 M4 M41 G0 X50 Z100 G1 X0 Z80 ; Point A G1 X15 Z65 ; A-B section Z55 ; B-C section X40 Z30 ; C-D section Z0 ; D-E section G0 X50 Z100 M30	G90 G95 G96 F0.15 S180 T2 D1 M4 M41 G0 X50 Z100 G1 X0 Z80 ; Point A G1 G91 X15 Z-15 ; A-B section Z-10 ; B-C section X25 Z-25 ; C-D section Z-30 ; D-E section G0 G90 X50 Z100 M30

Programming in diameters.

Absolute coordinates (G90).	Incremental coordinates (G91).
G90 G95 G96 F0.15 S180 T2 D1 M4 M41 G0 X100 Z100 G1 X0 Z80 ; Point A G1 X30 Z65 ; A-B section Z55 ; B-C section X80 Z30 ; C-D section Z0 ; D-E section G0 X100 Z100 M30	G90 G95 G96 F0.15 S180 T2 D1 M4 M41 G0 X100 Z100 G1 X0 Z80 ; Point A G1 G91 X30 Z-15 ; A-B section Z-10 ; B-C section X50 Z-25 ; C-D section Z-30 ; D-E section G0 G90 X100 Z100 M30

1.6 Tool path programming

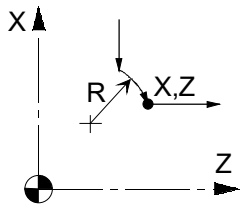
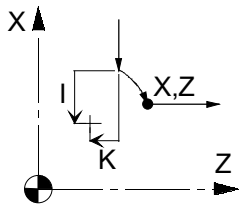
G00 **Rapid traverse.**

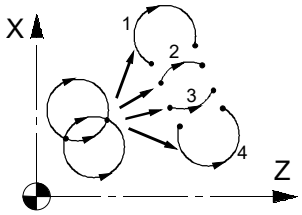
G01 **Linear interpolation.**

G02 **Clockwise circular interpolation.**

G03 **Counterclockwise circular interpolation.**

Functions G02/G03 offer two ways of programming in Cartesian coordinates.

Setting the end point and the radius.	Setting the end point and the center.
 G02/G03 X Z R	 G02/G03 X Z I K

Sign of the radius.	
	Arc 1: G02 X... Z... R-... Arc 2: G02 X... Z... R+... Arc 3: G03 X... Z... R+... Arc 4: G03 X... Z... R-...

G36 **Corner rounding, radius blend.**

The programming format is "G36 I–" where "I" is the radius. Parameter I is valid for all four functions G36, G37, G38 and G39 and stays active until a new value is programmed.

G37 **Tangential entry.**

The programming format is "G37 I–" where "I" is the radius.

G38 **Tangential exit.**

The programming format is "G38 I–" where "I" is the radius.

G39 **Corner chamfering.**

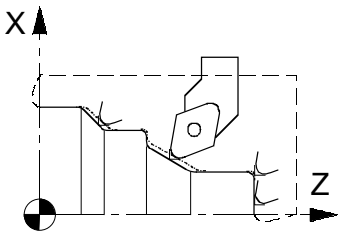
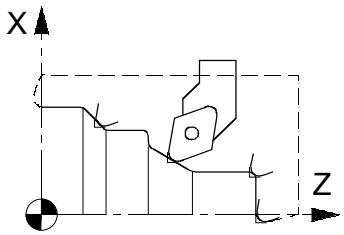
The programming format is "G39 I–" where "I" is the size of the chamfer.

G40 **Cancellation of tool radius compensation.**

G41 **Left-hand tool radius compensation.**

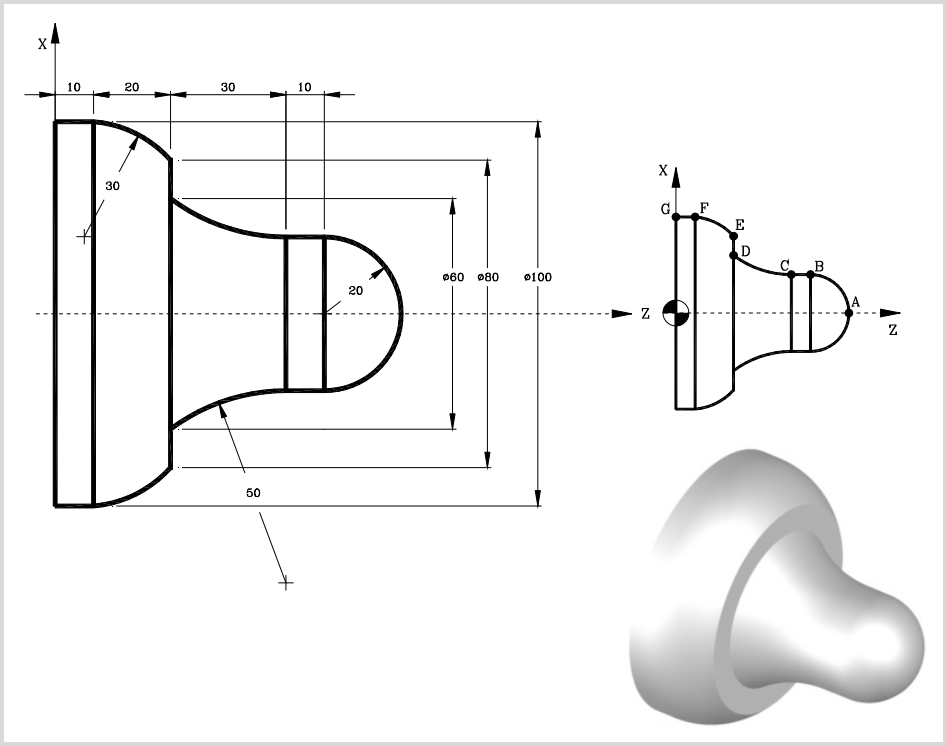
G42 **Right-hand tool radius compensation.**

The tool will position to the left or to the right of the programmed path, according to the machining direction.

Without compensation.	With compensation.
	

1.6.1 Example. Arc programming "G02/G03".

Programming in radius.



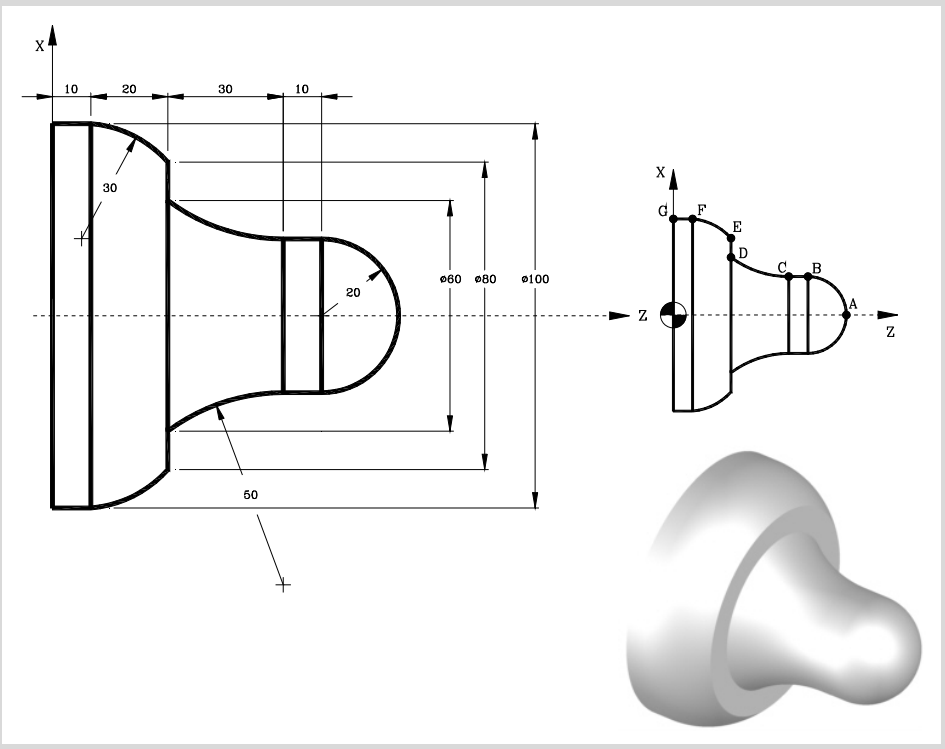
Programming with the arc center.

Absolute coordinates (G90).	Incremental coordinates (G91).
G90 G95 G96 F0.15 S180 T2 D1 M4 G0 X60 Z120 G1 X0 Z90 ; Point A G3 X20 Z70 I0 K-20 ; A-B section G1 Z60 ; B-C section G2 X30 Z30 I50 K0 ; C-D section G1 X40 ; D-E section G3 X50 Z10 I-19.9 K-22.45 ; E-F section G1 Z0 ; F-G section G0 X60 Z120 M30	G90 G95 G96 F0.15 S180 T2 D1 M4 G0 X60 Z120 G1 X0 Z90; Point A G91 G3 X20 Z-20 I0 K-20 ; A-B section G1 Z-10 ; B-C section G2 X10 Z-30 I50 K0 ; C-D section G1 X10 ; D-E section G3 X10 Z-20 I-19.9 K-22.45 ; E-F section G1 Z-10 ; F-G section G0 G90 X60 Z120 M30

Programming with the arc center.

Absolute coordinates (G90).	Incremental coordinates (G91).
G90 G95 G96 F0.15 S180 T2 D1 M4 G0 X60 Z120 G1 X0 Z90 ; Point A G3 X20 Z70 R20 ; A-B section G1 Z60 ; B-C section G2 X30 Z30 R50 ; C-D section G1 X40 ; D-E section G3 X50 Z10 R30 ; E-F section G1 Z0 ; F-G section G0 X60 Z120 M30	G90 G95 G96 F0.15 S180 T2 D1 M4 G0 X60 Z120 G1 X0 Z90 ; Point A G91 G3 X20 Z-20 R20 ; A-B section G1 Z-10 ; B-C section G2 X10 Z-30 R50 ; C-D section G1 X10 ; D-E section G3 X10 Z-20 R30 ; E-F section G1 Z-10 ; F-G section G0 G90 X60 Z120 M30

Programming in diameters.



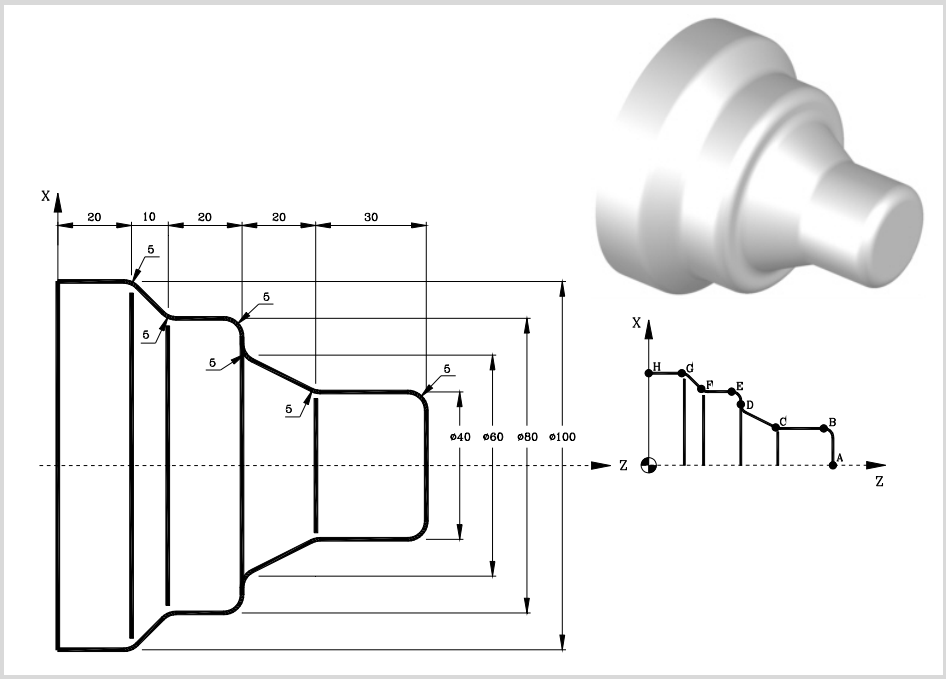
Programming with the arc center.

Absolute coordinates (G90).	Incremental coordinates (G91).
G90 G95 G96 F0.15 S180 T2 D1 M4 G0 X120 Z120 G1 X0 Z90 ; Point A G3 X40 Z70 I0 K-20 ; A-B section G1 Z60 ; B-C section G2 X60 Z30 I50 K0 ; C-D section G1 X80 ; D-E section G3 X100 Z10 I-19.9 K-22.45 ; E-F section G1 Z0 ; F-G section G0 X120 Z120 M30	G90 G95 G96 F0.15 S180 T2 D1 M4 G0 X120 Z120 G1 X0 Z90 ; Point A G91 G3 X40 Z-20 I0 K-20 ; A-B section G1 Z-10 ; B-C section G2 X20 Z-30 I50 K0 ; C-D section G1 X20 ; D-E section G3 X20 Z-20 I-19.9 K-22.45 ; E-F section G1 Z-10 ; F-G section G0 G90 X120 Z120 M30

Programming with the arc center.

Absolute coordinates (G90).	Incremental coordinates (G91).
G90 G95 G96 F0.15 S180 T2 D1 M4 G0 X120 Z120 G1 X0 Z90 ; Point A G3 X40 Z70 R20 ; A-B section G1 Z60 ; B-C section G2 X60 Z30 R50 ; C-D section G1 X80 ; D-E section G3 X100 Z10 R30 ; E-F section G1 Z0 ; F-G section G0 X120 Z120 M30	G90 G95 G96 F0.15 S180 T2 D1 M4 G0 X120 Z120 G1 X0 Z90 ; Point A G91 G3 X40 Z-20 R20 ; A-B section G1 Z-10 ; B-C section G2 X20 Z-30 R50 ; C-D section G1 X20 ; D-E section G3 X20 Z-20 R30 ; E-F section G1 Z-10 ; F-G section G0 G90 X120 Z120 M30

1.6.2 Example. Tangential entry/exit "G37/G38" and corner rounding (radius blend) "G36".



Programming with the arc center.

Absolute coordinates (G90).
G90 G95 G96 F0.15 S180 T2 D1 M4 G0 X120 Z120 G42 X0 ; Begin tool radius compensation. G01 X0 Z100 G37 I4 ; Tangential entry at point A. G01 X40 ; A-B section G36 I5 ; Rounding B G01 Z70 ; B-C section G36 ; Rounding C (I radius stays active) G01 X60 Z50 ; C-D section G36 ; Rounding D G01 X80 ; D-E section G36 ; Rounding E G01 Z30 ; E-F section G36 ; Rounding F G01 X100 Z20 ; F-G section G36 ; Rounding G G01 Z0 ; G-H section G38 I4 ; Tangential exit. G0 X120 G40 Z120 ; End of tool radius compensation. M30

1.

BASIC CONCEPTS.
Tool path programming

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

CANNED CYCLE PROGRAMMING. 2

2.1 Introduction

The canned cycles edited in ISO code are defined using a "G" function and its relevant parameters.

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

Machining canned cycles with a live tool:

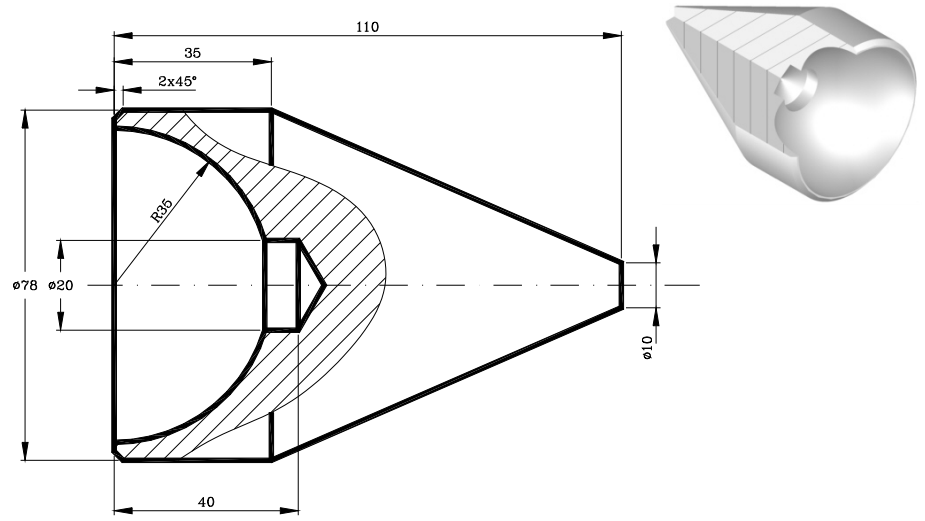
G160	Drilling / tapping canned cycle on the face of the part.
G161	Drilling / tapping canned cycle on the side of the part.
G162	Slot milling canned cycle along the side of the part.
G163	Slot milling canned cycle along the face of the part.

A canned cycle may be defined anywhere in the program, that is, in the main program as well as in a subroutine.

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

Parameter	Z-X plane	W-X plane	A-B plane
Parameter Z and all related to it, with the abscissa axis	Z axis	W axis	A axis
Parameter X and all related to it, with the ordinate axis	X axis	X axis	B axis

2.2 Example. Inside turning of curved and straight sections.



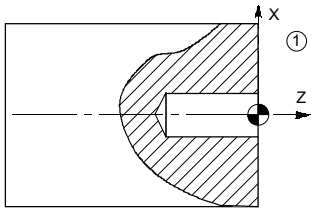
Raw dimensions Ø80x114 mm.

; First fixture.

; Set part zero:

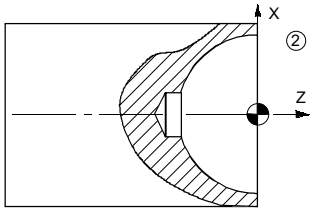
```
V.A.ORGT[1].X=0 V.A.ORGT[1].Z=112
G54
G192 S2200
```

; Operation 1 (Drilling)



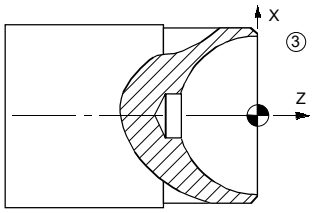

```
G94 G97 F90 S600 M4
G0 Z150
T9 D1
G0 X0 Z8
G83 X0 Z0 I45.773 B9 D4 K0 H0 C1
```

; Operation 2 (Inside curved turning)




```
G95 G96 F0.2 S120 M4
T8 D1
G0 X20 Z20
G1 G41 X18 Z5
G84 X70 Z0 Q20 R-33.541 C2 L0.3 M0.3 H0.1 I-35 K0
G0 G40 Z150
```

; Operation 3 (Facing and outside turning)



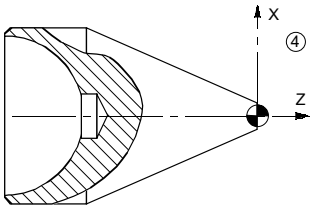

```
G95 G96 F0.2 S180 M4
T2 D1
G0 X90 Z20
G1 X78 Z5
G1 Z-40
G1 X85
G0 Z0
G1 X66
G1 Z5
G1 G42 X72 Z1
G1 X80 Z-3
G0 G40 Z150
```

; Second fixture

; Set new part zero:

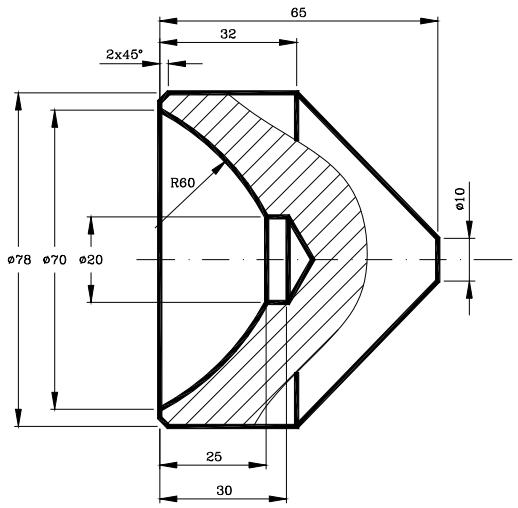
```
#MSG ["NEW FIXTURE - INVERT PART"]
M0 M5
#MSG [""]
V.A.ORGT[1].X=0 V.A.ORGT[1].Z=110
G54
G192 S2200
```

; Operation 4 (Taper turning and facing)




```
G95 G96 F0.2 S180 M4
G0 X90 Z20
G1 G42 X84 Z5
G81 X10 Z0 Q78 R-75 C2 L0.3 M0.3 H0.1
G0 G40 X14 Z0
G1 X-0.4
G0 Z150
M30
```

2.3 Example. Facing of inside curved sections and outside straight sections.



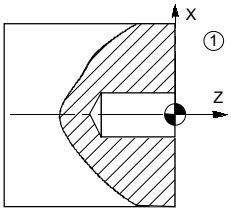
Raw dimensions Ø80x69 mm.

; First fixture.

; Set part zero:

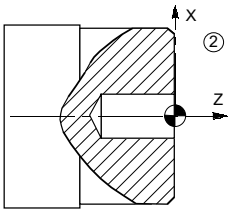
```
V.A.ORGT[1].X=0 V.A.ORGT[1].Z=67
G54
G192 S2200
```

; Operation 1 (Drilling)



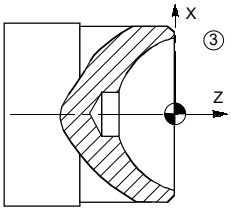

```
G94 G97 F90 S600 M4
G0 Z150
T9 D1
G0 X0 Z8
G83 X0 Z0 I38.773 B3 D7 K0 H0 C4
```

; Operation 2 (Facing and outside turning)




```
G95 G96 F0.2 S180 M4
T2 D1
G0 X90 Z20
G1 X85 Z0
G1 X18
G1 Z5
G0 G42 X72 Z1
G1 X78 Z-2
Z-40
X85
G0 Z150
```

; Operation 3 (Inside curved facing)



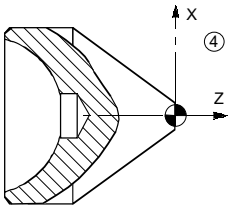

```
G95 G96 F0.2 S100 M4
T8 D1
G0 X20 Z20
G1 G42 X17 Z2
G85 X20 Z-25 Q70 R0 C1.4 L0.3 M0.3 H0.1 I-28.043
K53.043
G0 G40 Z150
```

; Second fixture

; Set new part zero:

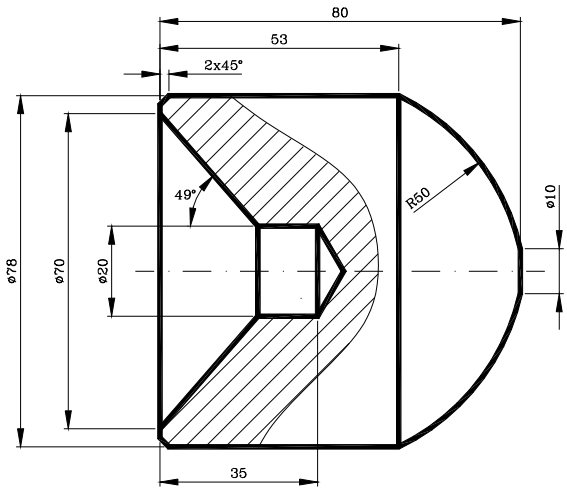
```
#MSG ["NEW FIXTURE - INVERT PART"]
M0 M5
#MSG [""]
V.A.ORGT[1].X=0 V.A.ORGT[1].Z=65
G54
G192 S2200
```

; Operation 4 (Outside taper facing)




```
G95 G96 F0.2 S180 M4
T2 D1
G0 X90 Z20
G1 G41 X83 Z5
G82 X78 Z-33 Q10 R0 C2 L0.3 M0.3 H0.1
G0 G40 X14 Z0
G1 X-0.4
G0 Z150
M30
```

2.4 Example. Facing of inside straight sections and outside curved sections.



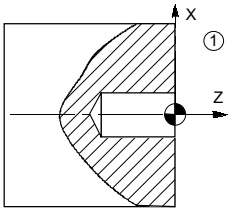
Raw dimensions Ø80x84 mm.

; First fixture.

; Set part zero:

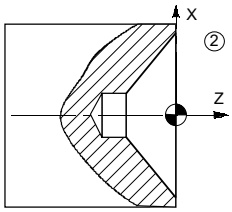
```
V.A.ORGT[1].X=0 V.A.ORGT[1].Z=82
G54
G192 S2200
```

; Operation 1 (Drilling)



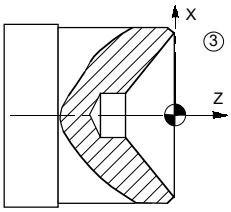

```
G95 G97 F0.15 S600 M4
G0 Z150
T9 D1
G0 X0 Z8
G83 X0 Z0 I40.773 B3 D7 K10 H0 C4
G0 Z150
```

; Operation 2 (Inside taper turning)




```
G95 G96 F0.2 S100 M4
T8 D1
G0 X20 Z20
G1 G42 X18 Z5
G82 X20 Z-21.732 Q70 R0 C2 L0.2 M0.2 F0.15 H0.1
G0 G40 Z150
```

; Operation 3 (Facing and outside turning)



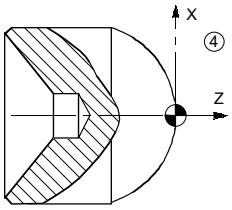

```
G95 G96 F0.2 S180 M4
T2 D1
G0 X90 Z20
G1 X78 Z5
G1 Z-40
G1 X85
G0 Z0
G1 X66
G1 Z5
G1 G42 X72 Z1
G1 X80 Z-3
G0 G40 Z15
```

; Second fixture

; Set new part zero:

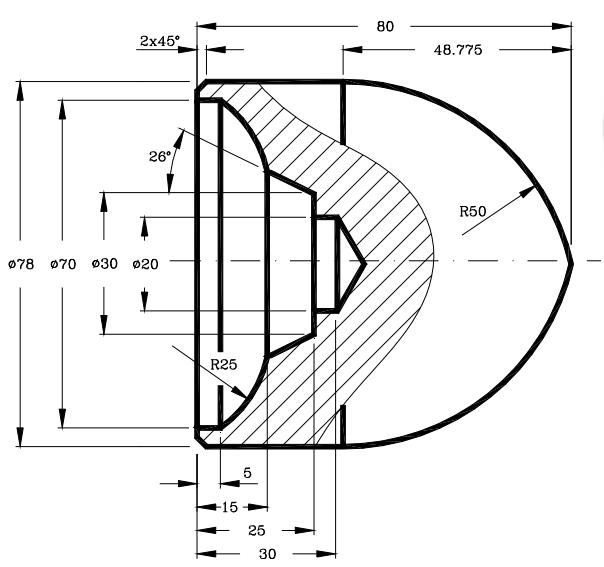
```
#MSG ["NEW FIXTURE - INVERT PART"]
M0 M5
#MSG [""]
V.A.ORGT[1].X=0 V.A.ORGT[1].Z=80
G54
G192 S2200
```

; Operation 4 (Outside curved facing)




```
G95 G96 F0.2 S180 M4
T2 D1
G0 X90 Z20
G41 X84 Z5
G85 X78 Z-27 Q10 R0 C1.5 L0.3 M0.3 H0.1 I-45.011
K-21.772
G0 G40 X14 Z0
G1 X-0.4 F0.2
G0 Z150
M30
```

2.5 Example. Inside roughing along Z axis and outside turning of curved sections.



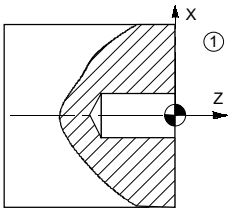
Raw dimensions Ø80x84 mm.

; First fixture.

; Set part zero:

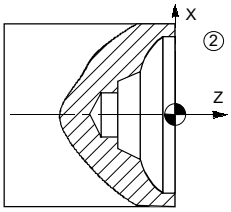
```
V.A.ORGT[1].X=0 V.A.ORGT[1].Z=82
G54
G192 S2200
```

; Operation 1 (Drilling)



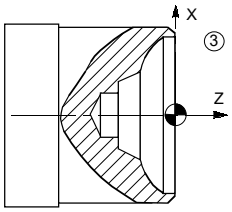

```
G94 G97 F90 S600 M4
G0 Z150
T9 D1
G0 X0 Z8
G83 X0 Z0 I35.773 B5 D5 K15 H0 C1.5
G0 Z150
```

; Operation 2 (Inside profile facing)




```
G95 G96 F0.2 S100 M4
T8 D1
G0 X20 Z20
G1 X16 Z5
G69 X20 Z-25 C1.5 L0.3 H0.1 S100 E110
$GOTO N120
N100 G1 X30 Z-25
X39.755 Z-15
G2 X70 Z-5 I-5.29 K24.434
N110 G1 X70 Z4
N120: G0 Z150 ; Jump target blocks followed by:
G0 G40 Z150
```

; Operation 3 (Facing and outside turning)



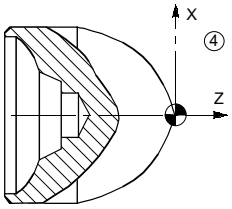

```
G95 G96 F0.2 S180 M4
T2 D1
G0 X90 Z20
G1 X78 Z5
G1 Z-40
G1 X85
G0 Z0
G1 X66
G1 Z5
G1 G42 X72 Z1
G1 X80 Z-3
G0 G40 Z150
```

; Second fixture

; Set new part zero:

```
#MSG ["NEW FIXTURE - INVERT PART"]
M0 M5
#MSG [""]
V.A.ORGT[1].X=0 V.A.ORGT[1].Z=80
G54
G192 S2200
```

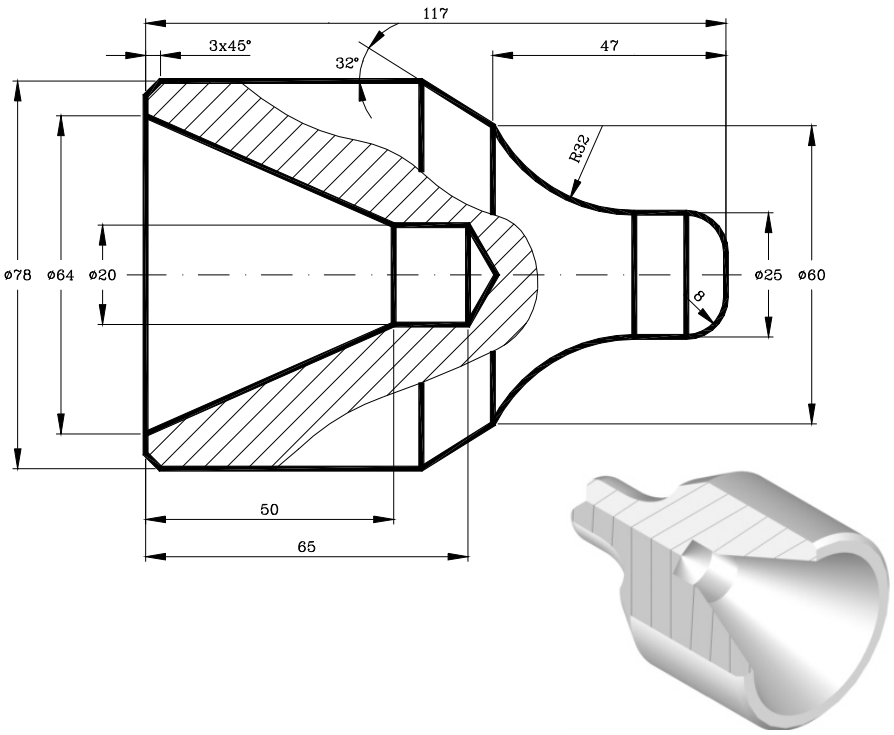
; Operation 4 (Outside CURVED facing)




```
G95 G96 F0.2 S180 M4
T2 D1
G0 X90 Z20
G1 G42 X84 Z5
G84 X0 Z0 Q78 R-48.775 C2 L0.3 M0.3 H0.1 I-11 K-48.775
G0 G40 Z150
M30
```


FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

2.6 Example. Inside turning of straight sections and outside roughing along the Z axis.



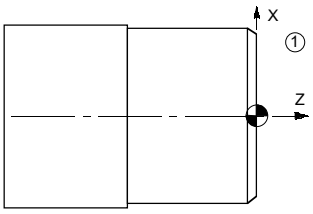
Raw dimensions Ø80x121 mm.

; First fixture.

; Set part zero:

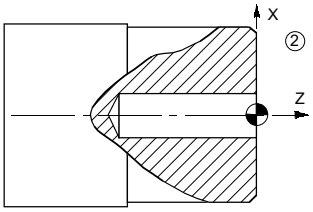
```
V.A.ORGT[1].X=0 V.A.ORGT[1].Z=119
G54
G192 S2200
```

; Operation 1 (Facing and outside turning)



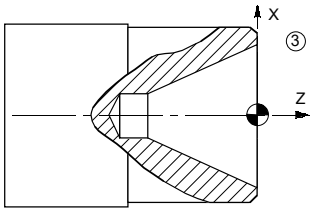

```
G95 G96 F0.2 S180 M4
G0 Z150
T2 D1
G0 X90 Z20
G1 X85 Z0
G1 X-0.4
Z5
G0 X78 Z2
G1 Z-60
X85
G0 G41 X80 Z-4
G1 X70 Z1
G0 Z150
```

; Operation 2 (Drilling)




```
G94 G97 F90 S600 M4
T9 D1
G0 X0 Z5
G83 X0 Z0 I70.773 B8 D4 K1 H0 C1
G0 Z150
```

; Operation 3 (Inside taper turning)



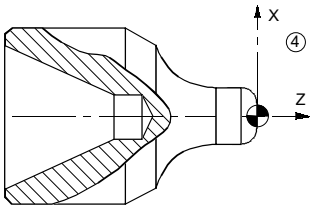

```
G95 G96 F0.2 S120 M4
T8 D1
G0 G41 X18 Z5
G81 X64 Z0 Q20 R-50 C1.5 L0.3 M0.25 H0.1
G0 Z150
```

; Second fixture

; Set new part zero:

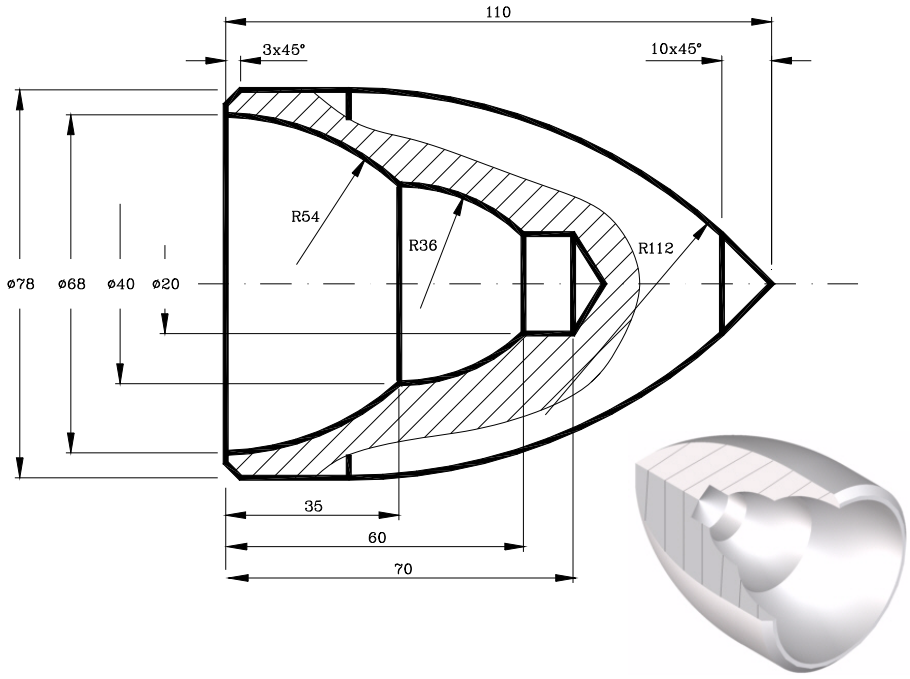
```
#MSG ["NEW FIXTURE - INVERT PART"]
M0 M5
#MSG [""]
V.A.ORGT[1].X=0 V.A.ORGT[1].Z=117
G54
G192 S2200
```

; Operation 4 (Taper turning and facing)




```
G95 G96 F0.2 S180 M4
T2 D1
G0 X90 Z20
G1 X85 Z5
G69 X78 Z-61.403 C1 L0.3 H0.1 S100 E110
$GOTO N120:
N100 G1 G5 X60 Z-47
G3 X25 Z-18.474 I14.5 K28.526
G1 G36 I8 X25 Z0
N110 X-0.4 Z0
N120: G0 Z150
M30
```

2.7 Example. Inside and outside roughing along the X axis.



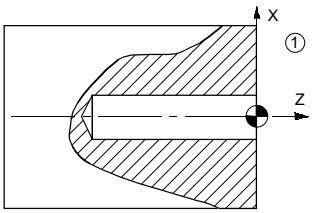
Raw dimensions Ø80x121 mm.

; First fixture.

; Set part zero:

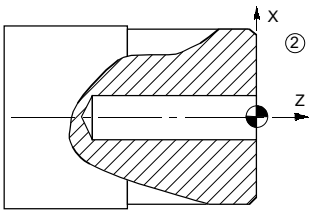
```
V.A.ORGT[1].X=0 V.A.ORGT[1].Z=112
G54
G192 S2200
```

; Operation 1 (Drilling)



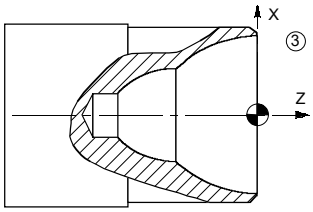

```
G94 G97 F90 S600 M4
G0 Z150
T9 D1
G0 X0 Z10
G83 X0 Z0 I75.773 B8 D2 K50 H0 C5
G0 Z150
```

; Operation 2 (Facing and outside turning)




```
G95 G96 F0.2 S180 M4
T2 D1
G0 X78 Z5
G1 Z-60
X85
G0 Z0
G1 X18
G0 Z5
G0 G42 X70 Z1
G1 X80 Z-4
G0 G40 X85 Z150
```

; Operation 3 (Inside profile turning)



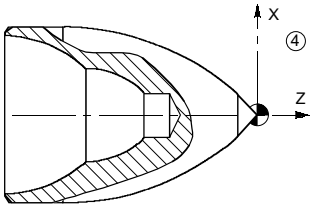

```
G95 G96 F0.2 S120 M4
T8 D1
G0 X18 Z20
G1 Z5
G68 X68 Z0 C1.5 L0.4 H0 S100 E110
G0 G41 X68 Z1
G5 G1 Z0 F0.1
N100 G3 X40 Z-35 I-53.985 K1.293
N110 G3 X20 Z-60 R36
G1 X18
G1 Z5
G0 G40 G7 Z150
```

; Second fixture

; Set new part zero:

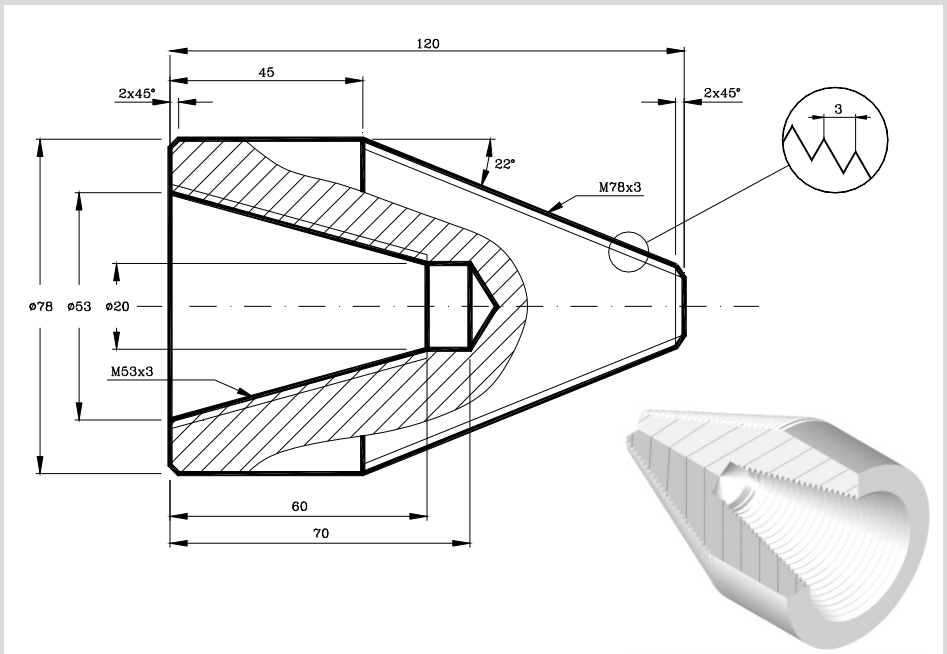
```
#MSG ["NEW FIXTURE - INVERT PART"]
M0 M5
#MSG [""]
V.A.ORGT[1].X=0 V.A.ORGT[1].Z=110
G54
G192 S2200
```

; Operation 4 (Taper turning and facing)




```
G95 G96 F0.2 S180 M4
T2 D1
G0 X85 Z20
G1 Z5
G68 X0 Z0 C1.5 L0.4 H0 S150 E160
G0 G42 X0 Z10
G1 G5 Z0 F0.1
N150 G1 X20 Z-10
N160 G3 X78 Z-85.2 I-83 K-75.2
G1 X80
G0 G40 G7 Z150
M30
```

2.8 Example. Inside and outside taper threading



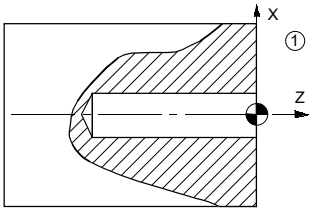
Raw dimensions Ø80x121 mm.

; First fixture.

; Set part zero:

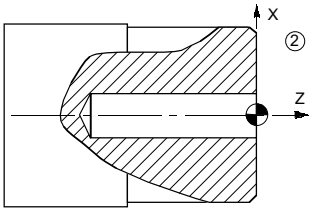
```
V.A.ORGT[1].X=0 V.A.ORGT[1].Z=122
G54
G192 S2200
```

; Operation 1 (Drilling)



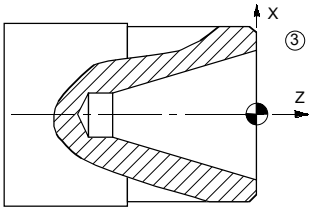

```
G95 G97 F0.15 S600 M4
G0 Z150
T9 D1
G0 X0 Z5
G83 X0 Z0 I75.773 B5 D5 K130 H0 C2
G0 Z150
```

; Operation 2 (Facing and outside turning)



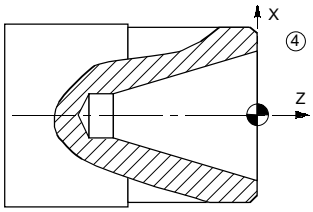

```
G95 G96 F0.2 S180 M4
T2 D1
G0 X78 Z5
G1 Z-50
X86
G0 G41 X78 Z-2.5
G1 X74 Z0
X16
G0 G40 Z150
```

; Operation 3 (Inside taper turning)




```
G95 G96 F0.2 S120 M4
T8 D1
G0 X20 Z20
G1 G41 X16 Z1.5
G81 X53 Z0 Q20 R-60 C1.5 L0.3 M0.25 H0.1
G0 G40 Z150
```

; Operation 4 (Inside taper threading)



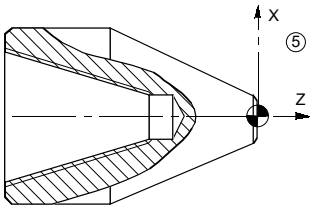

```
G95 G96 F0.15 S60 M4
T10 D1
G0 X20 Z20
G1 X16 Z1.5
G86 X53 Z0 Q20 R-60 I-1 B0.4 D-2 L0 C-3 J5 A29.5
G0 Z150
```

; Second fixture

; Set new part zero:

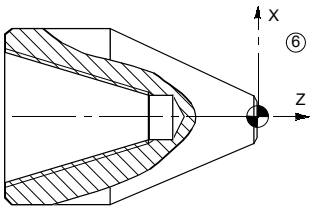
```
#MSG ["NEW FIXTURE - INVERT PART"]
M0 M5
#MSG [""]
V.A.ORGT[1].X=0 V.A.ORGT[1].Z=120
G54
G192 S2200
```

; Operation 5 (Outside taper turning)




```
G95 G96 F0.2 S180 M4
T2 D1
G0 X90 Z20
G1 G42 X85 Z5
G81 X17.396 Z0 Q78 R-75 C2 L0.3 M0.3 H0.1
G0 G40 X20.396 Z0
G1 X-0.4
G1 Z5
G0 Z150
```

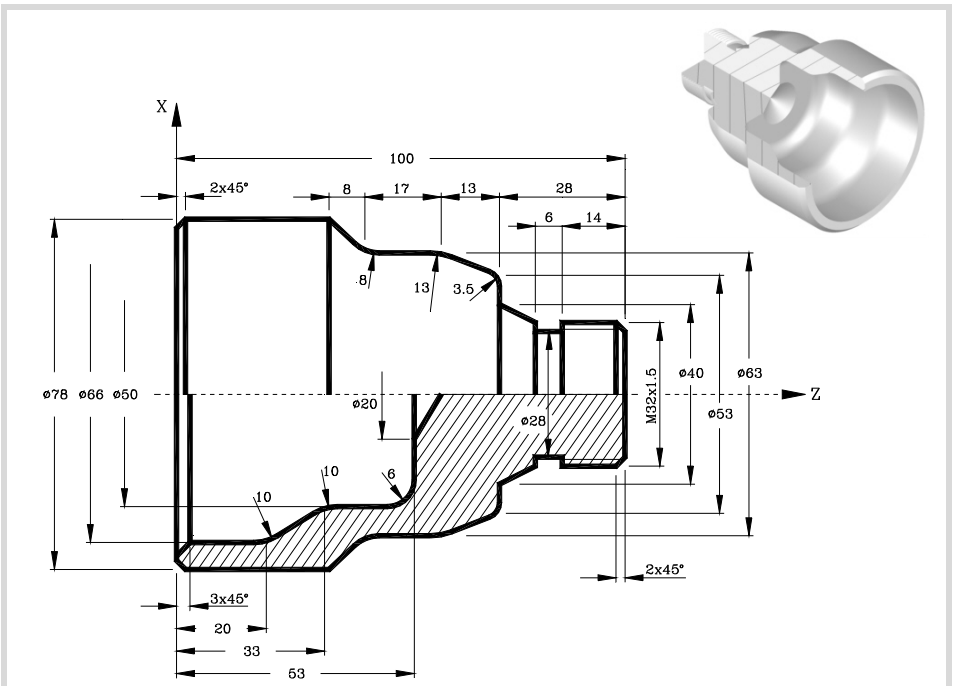
; Operation 6 (Outside taper threading)




```
G95 G96 F0.15 S60 M4
T11 D1
G0 X80 Z1.5
G86 X17.396 Z0 Q78 R-75 I2 B0.4 D-2 L0 C-3 J5
A29.5
G0 Z150
M30
```


FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

2.9 Example. Roughing on the X axis. Outside grooving and threading.



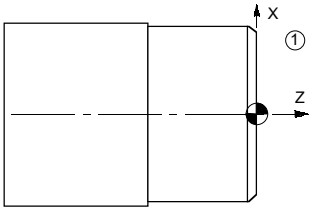
Raw dimensions Ø80x104 mm.

; First fixture.

; Set part zero:

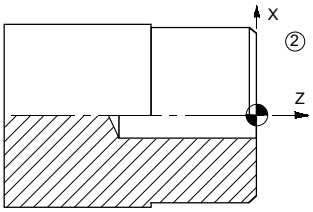
```
V.A.ORGT[1].X=0 V.A.ORGT[1].Z=102
G54
G192 S2200
```

; Operation 1 (Facing and outside turning)



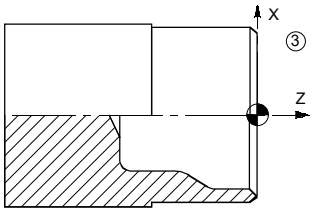

```
G95 G96 F0.2 S180 M4
G0 Z150
T2 D1
G0 X90 Z20
G1 X78 Z5
Z-38
X82
G0 Z0
G1 X-0.4
G1 Z5
G0 G42 X72 Z1
G1 X80 Z-3
X85
G0 G40 X60 Z150
```

; Operation 2 (Drilling)




```
G94 G97 F90 S600 M4
T9 D1
G0 X0 Z10
G83 X0 Z1 I58.773 B5 D2 K5 H0 C1
G0 Z150
```

; Operation 3 (Inside profile turning)



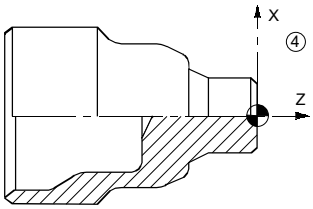

```
G95 G96 F0.4 S120 M4
T8 D1
G0 X18.2 Z10
G68 X74 Z1 C1 L0.3 H0 S100 E110
G0 G41 X74 Z1
N100 G1 G5 X66 Z-3
Z-17.169
G3 X63.033 Z-22.411 I-10 K0
G1 X50 Z-33
G36 I10
X50 Z-47
G3 X38 Z-53 I-6 K0
N110 G1 X19 Z-53
G0 G40 G7 Z150
```

; Second fixture

; Set new part zero:

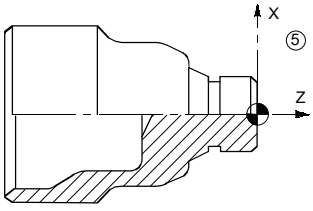
```
#MSG ["NEW FIXTURE - INVERT PART"]
M0 M5
#MSG [""]
V.A.ORGT[1].X=0 V.A.ORGT[1].Z=100
G54
G192 S2200
```

; Operation 4 (Outside profile turning)



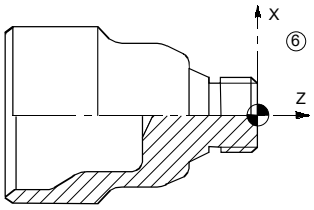

```
G95 G96 F0.2 S180 M4
T2 D1
G0 X90 Z20
G1 X82 Z0
G1 X-0.4
G1 Z5
G0 X82.5 Z4
G68 X27 Z0.5 C1 L0.3 H0 S120 E130
G1 G42 X27 Z0.5
N120 G1 G5 X32 Z-2
X32 Z-20
X40 Z-28
X53 Z-28
G36 I3.5
X63 Z-41
G36 I13
X63 Z-54.836
G2 X67.327 Z-60.308 I8 K0
G1 X78 Z-66
N130 X81 Z-67
G0 G40 X80 Z150
```

; Operation 5 (Grooving)



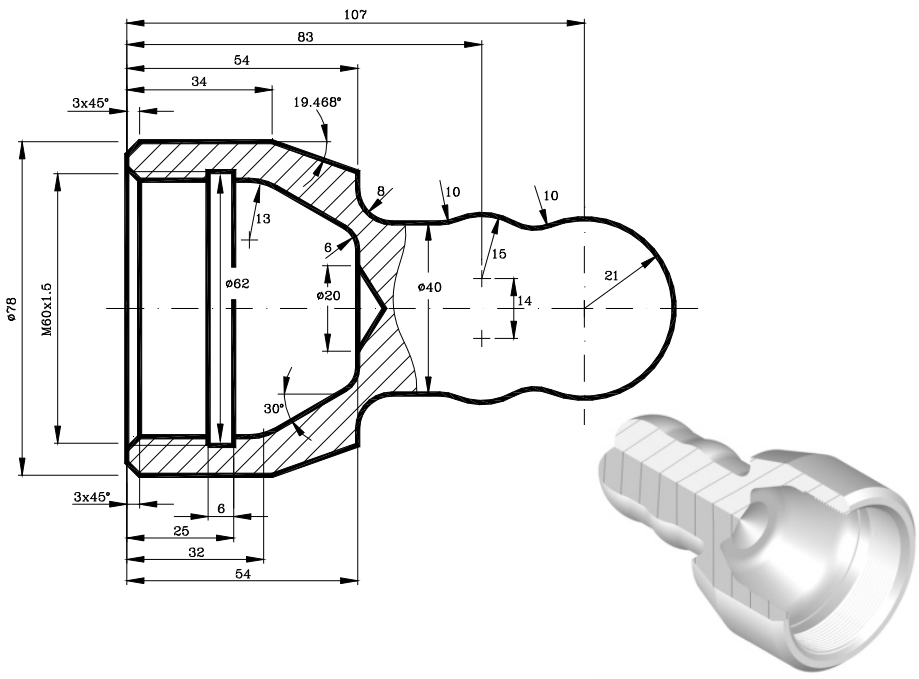

```
G95 G96 F0.08 S50 M4
T12 D1
G0 G41 X34 Z-17
G88 X32 Z-20 Q28 R-14 D1 K2
G0 G40 X80 Z150
```

; Operation 6 (Outside threading)




```
G95 G96 F0.15 S60 M4
T11 D1
G0 X35 Z5
G86 X32 Z3 Q32 R-16 I0.8 B0.1 D1 L0 C1.5 J0 A29.5
G0 X80 Z150
M30
```

2.10 Example. Outside pattern repeat. Internal grooving and threading.



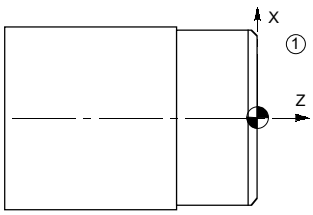
Raw dimensions Ø80x104 mm.

; First fixture.

; Set part zero:

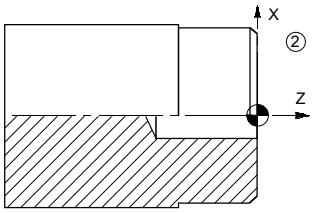
```
V.A.ORGT[1].X=0 V.A.ORGT[1].Z=130
G54
G192 S2200
```

; Operation 1 (Facing and outside turning)



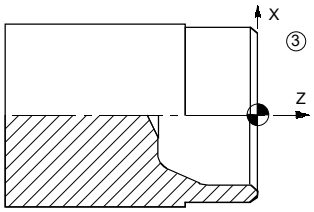

```
G95 G96 F0.2 S180 M4
G0 Z150
T2 D1
G0 X90 Z20
G1 X78 Z5
G1 Z-36
G1 X82
G0 Z0
G1 X-0.4
G1 Z5
G0 G42 X70 Z1
G1 X80 Z-4
G0 G40 X90 Z150
```

; Operation 2 (Drilling)



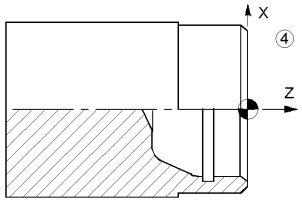

```
G94 G97 F90 S600 M4
T9 D1
G0 X0 Z10
G83 X0 Z1 I59.773 B13 D2 K1 H0 C1
G0 Z150
```

; Operation 3 (Inside profile turning)



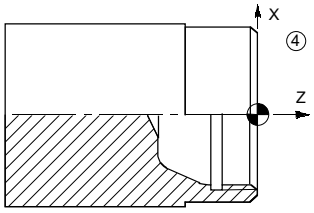

```
G95 G96 F0.2 S120 M4
T8 D1
G0 X16 Z5
G68 X64.35 Z0 C1 L0.5 H0 S100 E110
G0 G41 X65.35 Z0.5
N100 G1 G5 X58.35 Z-3
X58.35 Z-32
G36 I13
G1 X32.946 Z-54
G36 I6
N110 G1 X18 Z-54
G0 G40 G7 Z150
```

; Operation 4 (Inside grooving)




```
G95 G96 F0.08 S50 M4
T13 D1
G0 G41 X40 Z-15
G88 X60 Z-19 Q62 R-25 K5
G0 Z150
```

; Operation 5 (Inside threading)




```
G95 G96 F0.15 S60 M4
T10 D1
G0 X40 Z1.5
G86 X[60-2*0.8119] Z2 Q[60-2*0.8119] R-20
  I-0.8119 B0.3068 D1 L0 C-1.5 J0 A2.95
G0 Z150
```

Internal threading: M60*1.5.

- Thread pitch (C):

  C=1.5

- Thread depth (I):

  I= 0.5413*C; I=0.5413*1.5; I=-0.81195 mm (negative because it is internal threading).

- Depth of pass (B). For internal thread of p=1.5, number of passes = 7.

  B= I / √(no. passes); B=+0.81195 / √7; B= +0.3068 mm

- Penetration angle (A).

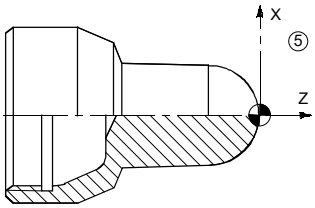
  A = Thread angle - 0.5º; A=29.5

; Second fixture

; Set new part zero:

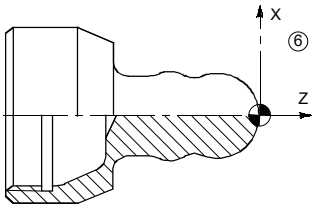
```
#MSG ["NEW FIXTURE - INVERT PART"]
M0 M5
#MSG [""]
V.A.ORGT[1].X=0 V.A.ORGT[1].Z=128
G54
G192 S2200
```

; Operation 6 (Outside profile roughing)




```
G95 G96 F0.2 S120 M4
T2 D1
G0 X85 Z5
G68 X0 Z0 C1.5 L0.5 H0 S120 E130
$GOTO N140:
N120 G3 X42 Z-21 I0 K-21
G1 X44 Z-45
X44 Z-69.5
X66 Z-73
N130 X80 Z-94
N140: G0 Z20
```

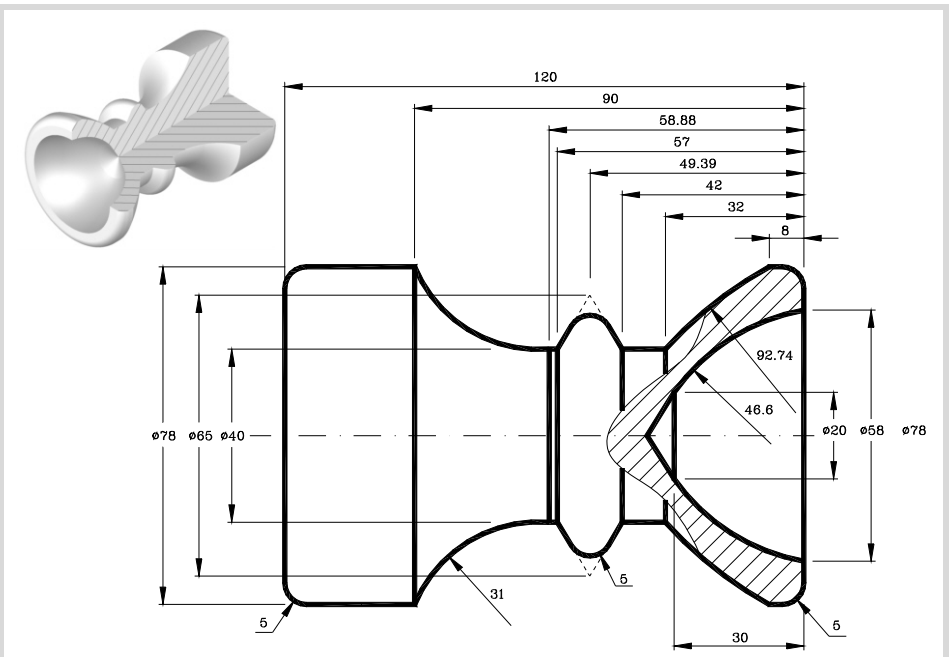
; Operation 7 (Outside profile finishing)




```
G95 G96 T4 D1 F0.2 S120 M4
G0 G90 X85 Z20
G1 X85 Z5
G66 X0 Z0 I2.5 C0.5 L0.2 H0.1 S150 E160
$GOTO N170:
N150 G5 G3 X33.56 Z-33.63 R21
G36 I10
G3 X40 Z-52.48 R15
G36 I10
G1 X40 Z-74
G36 I8
G1 X63.86 Z-74
N160 G7 X78 Z-94
N170 G90 G0 Z150
M30
```


FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

2.11 Example. Inside and outside roughing along the X axis.



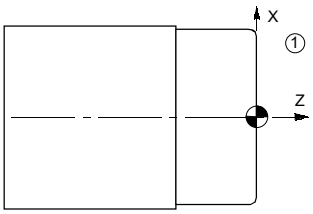
Raw dimensions Ø80x124 mm.

; First fixture.

; Set part zero:

```
V.A.ORGT[1].X=0 V.A.ORGT[1].Z=122
G54
G192 S2200
```

; Operation 1 (Facing and outside turning)



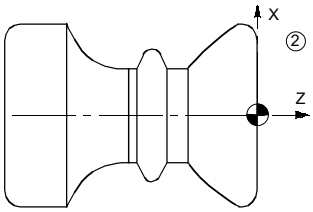

```
G95 G96 F0.2 S180 M4
G0 Z150
T2 D1
G0 X90 Z20
G1 X85 Z0
X -0.4
Z5
G1 G42 X0 Z0
X78 Z0
G36 I5
Z-35
X85
G0 G40 X90 Z150
```

; Second fixture

; Set new part zero:

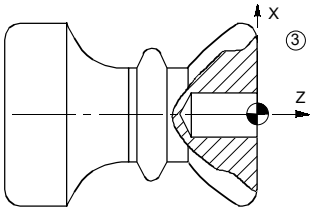
```
#MSG ["NEW FIXTURE - INVERT PART"]
M0 M5
#MSG [""]
V.A.ORGT[1].X=0 V.A.ORGT[1].Z=120
G54
G192 S2200
```

; Operation 2 (Outside profile turning)



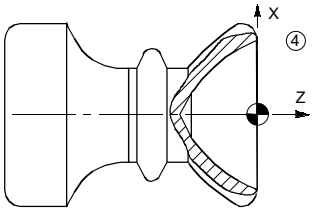

```
G95 G96 F0.2 S180 M4
T2 D1
G0 X85 Z0
G1 X-0.4
G0 Z5
G1 G42 X0 Z0
X78 Z0
G36 I5
Z-10
F0.1 T17 D1
G0 X90 Z-4
G68 X78 Z-8 C1 L0.5 H0.1 S100 E110
$GOTO N120:
N100 G3 X40 Z-32 R92.74
G1 Z-42
X65 Z-49.39
G36 I5
X40 Z-57
N110 G2 X78 Z-90 R31
N120: G0 Z150
```

; Operation 3 (Drilling)




```
G94 G97 F90 S600 M4
T9 D1
G0 X0 Z10
G83 X0 Z0 I35.773 B10 D2 H5 C2
G0 Z150
```

; Operation 4 (Inside profile turning)




```
G95 G96 F0.1 S120 M4
T8 D1
G0 X16 Z20
G1 Z5
G68 X58 Z0 C1 L0.5 H0.1 S150 E160
$GOTO N170:
N150 G3 X20 Z-30 R46.6
N160 G1 X19
N170: G1 Z20
G0 X85 Z150
M30
```

2.

CANNED CYCLE PROGRAMMING.

Example. Inside and outside roughing along the X axis.

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

C AXIS PROGRAMMING

3

3.1 Introduction.

The CNC allows activating axes and spindles as C axis, that interpolated with a linear axis makes it possible to mill the cylindrical surface or the face of a turning part. Although the machine may have several axes or spindle defined as "C" axis, only one of them may be active.

On a lathe, it is most common to activate the spindle as ·C· axis and use a live tool to machine the part.

Activating the spindle as ·C· axis.

To use a spindle as ·C· axis, it must be enabled as such first. The #CAX instruction activates a spindle as ·C· axis. Once this is done, it will be possible to program machining operation on the face or on the side using the instructions #FACE or #CYL respectively.

#CAX [<{spdl}><,{name}>]

{spdl}	Optional. Spindle to be activated as C axis.
{name}	Optional. Name of the C axis.

```
#CAX
#CAX [S1]
#CAX [S,C]
```

Activating the machining of the face of the part.

The #FACE instruction activates the machining of the face and it also defines the work plane. The axis to be activated as "C" axis will be determined by the work plane defined.

#FACE [{abs},{ord}<,{long}>]<[{kin}]>

{abs}	Abscissa axis of the work plane.
{ord}	Ordinate axis of the work plane.
{long}	Optional. Longitudinal axis of the tool.
{kin}	Optional. Number of the kinematics.

```
#FACE [X,C]
#FACE [X,C][1]
#FACE [X,C,Z]
#FACE [X,C,Z][1]
```

Programming the kinematics is optional; if not programmed, the CNC applies the first kinematics that has been defined in the machine parameters and is valid for this type of machining.

Activating the machining of the side of the part.

The #CYL instruction activates the machining of the side and it also defines the work plane. The axis to be activated as "C" axis will be determined by the work plane defined.

#CYL [{abs},{ord},{long}{radius}]<[{kin}]>

{abs}	Abscissa axis of the work plane.
{ord}	Ordinate axis of the work plane.
{long}	Longitudinal axis of the tool.
{radius}	Radius of the cylinder that will be machined.
{kin}	Optional. Number of the kinematics.

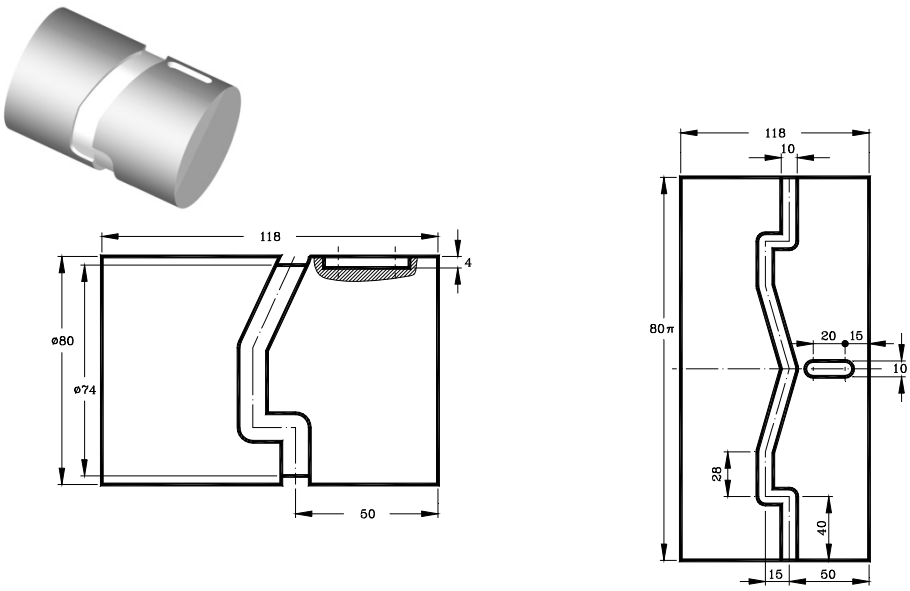
```
#CYL [X,C,Z45]
#CYL [C,Y,Z30]
#CYL [X,C,Z45][3]
```

Cancelling the machining operations.

The spindle active as C axis is deactivated with the #CAX OFF instruction and the spindle goes back to working as a normal spindle.

Instructions #FACE OFF/#CYL OFF cancel the machining operations defined with #FACE and #CYL.

3.2 Example. Machining of a profile in the ZC plane.



Raw dimensions Ø80x84 mm.

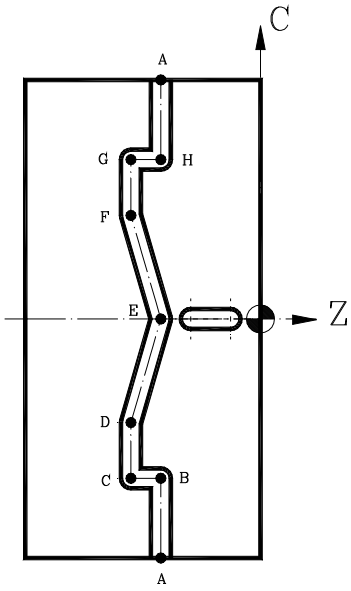
; Selecting a live tool:

```
G0 X100 Z150
T15 D1
M45 S600
```

; Operation 1 (Slot machining)

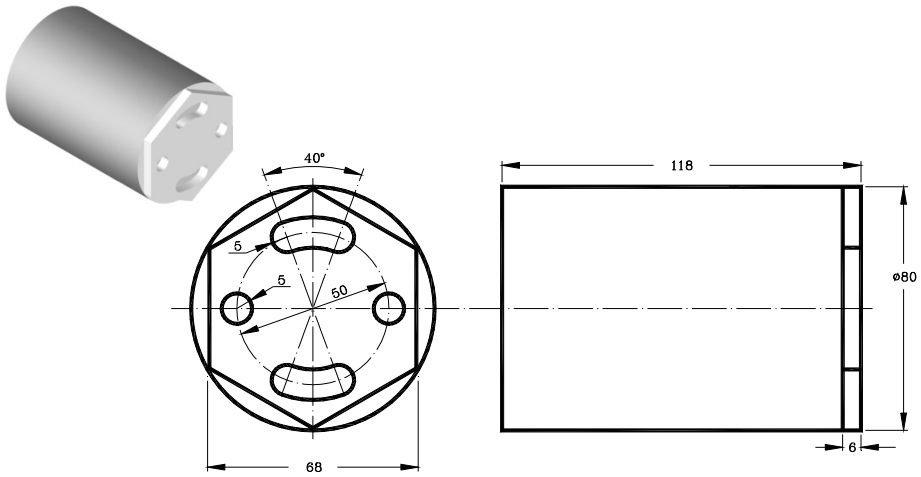
```
#CAX
    ; C axis selection.

#CYL [Z, C, X36]
    ; Work plane selection

G0 X90
Z-15 C0
G1 G94 X72 F100 M13
Z-35
G1 X90
```

; Operation 2 (grooving)




```
#CYL[Z, C, X37]
G0 Z-50 C-125.664
    ; Positioning in point A

G1 X74 F100
G91 C40 F50
    ; A-B section

Z-15
    ; B-C section

C28
    ; C-D section

Z15 C57.664
    ; D-E section

Z-15 C57.664
    ; E-F section

C28
    ; F-G section

Z15
    ; G-H section

C40
    ; H-A section

G90 X90
G0 Z10
M30
```

3.3 Example. Machining of a profile in the XC plane.

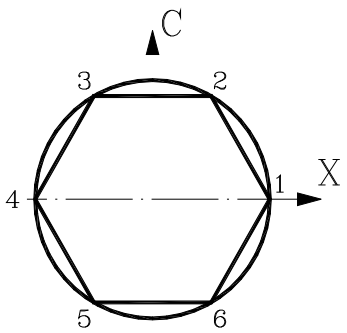


Raw dimensions Ø80x84 mm.

; Selecting a live tool:

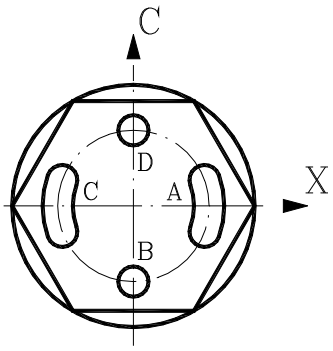
```
G0 X100 Z150
T16 D1
M45 S600
```

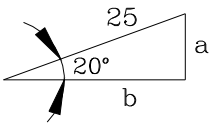
; Operation 1 (Hexagon machining)




```
#CAX
    ; C axis selection.
#FACE [X, C, Z]
    ; Work plane selection.

G94 Z10 C0 F100
G1 Z-6
G1 G42 X39.26 C0
    ; Positioning in point 1.

X19.63 C34
    ; 1-2 section

X-19.63 C34
    ; 2-3 section

X-39.26 C0
    ; 3-4 section

X-19.63 C-34
    ; 4-5 section

X19.63 C-34
    ; 5-6 section

X39.26 C0
    ; 6-1 section

G0 G40 X50
Z10
```

; Operation 2 (grooving and hole machining)





a=25sen20°
b=25cos20°

```
X23.492 C8.55
G1 Z-5 F50
    ; Positioning in point A.

G2 X23.492 C-8.55 R25
    ; Machining of groove A.

G0 Z5
X0 C-25
    ; Positioning in point B.

G1 Z-5
G1 Z5
G0 X-23.492 C-8.55
G1 Z-5
G2 X-23.492 C8.55 R25
    ; Machining of groove C.

G0 Z5
X0 C25
    ; Positioning in point D.

G1 Z-5
G0 Z5
M30
```

PROFILE EDITOR

4

The profile editor is used to edit quickly and easily simple rectangular, circular profiles and any type of profile consisting of straight and curved sections. As the profile data is entered, the editor shows a graphic representation of the profile.

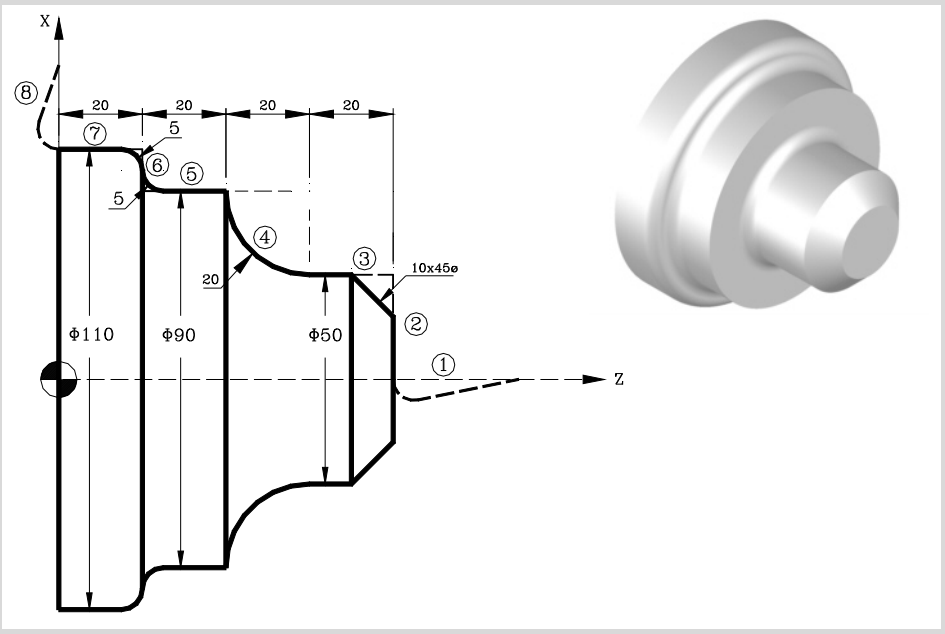
After defining the profile data, the CNC generates the necessary blocks and it will add them to the program inserting them after the block indicated by the cursor.

How to use the profile editor.

Several profiles may be edited without having to exit the profile editor. To edit a profile, proceed as follows:

1. Define the work plane at the profile editor.
2. Select the type of profile to be edited, such as a circular or rectangular profile or any profile.
3. For a rectangular or circular profile, define its data and insert it. For any profile, first select the starting point of the profile. Once the first point has been selected, draw the profile, which will be made up of straight and curved sections. If it has corner rounding, chamfering or tangential entries and exits, use one of these methods:
   - Treat them as individual sections when having enough information to define them.
   - Ignore them while defining the profile and, once it has been defined, select the corners that have those characteristics and insert them.
4. End the profile editing session by inserting them into the program. The portion of ISO-code program corresponding to the edited profile will be identified with the line "(#PROFILE)" or it will appear framed between the lines "(#PROFILE BEGIN)" and "(#PROFILE END)".

4.1 Example. Profile editor.



Profile definition without rounding, chamfers or tangential entries and exits.

Section.	Geometry.		
Starting point	Z = 100 X = 0		
Straight (1)	Z = 80 X = 0		
Straight (2)	Z = 80 X = 50		
Straight (3)	Z = 60 X = 50		
Clockwise arc (4)	Z = 40 X = 90	Center Z = 60 Center X = 90	Radius = 20
Straight (5)	Z = 20 X = 90		
Straight (6)	Z = 20 X = 110		
Straight (7)	Z = 0 X = 110		
Straight (8)	Z = 0 X = 150		

Definition of rounding, chamfers and tangential entries and exits

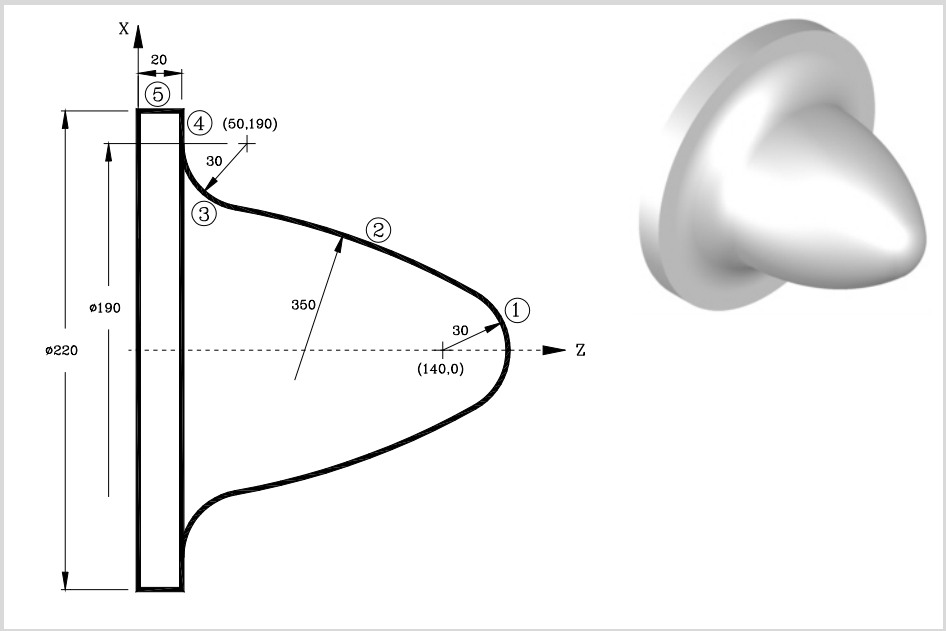
Select the "Corners" option. Press [ESC] to quit the "Corners" option.

Corners.		
Tangential entry	Select the corner "1-2"	Assign radius = 5
Chamfer	Select the corner "2-3"	Assign size = 10
Rounding	Select the corner "5-6"	Assign radius = 5
Rounding	Select the corner "6-7"	Assign radius = 5
Tangential exit	Select the corner "7-8"	Assign radius = 5

End of editing

Select the "END" option and save the profile. The CNC quits the profile editor and inserts the profile in the part-program.

4.2 Example. Profile editor.



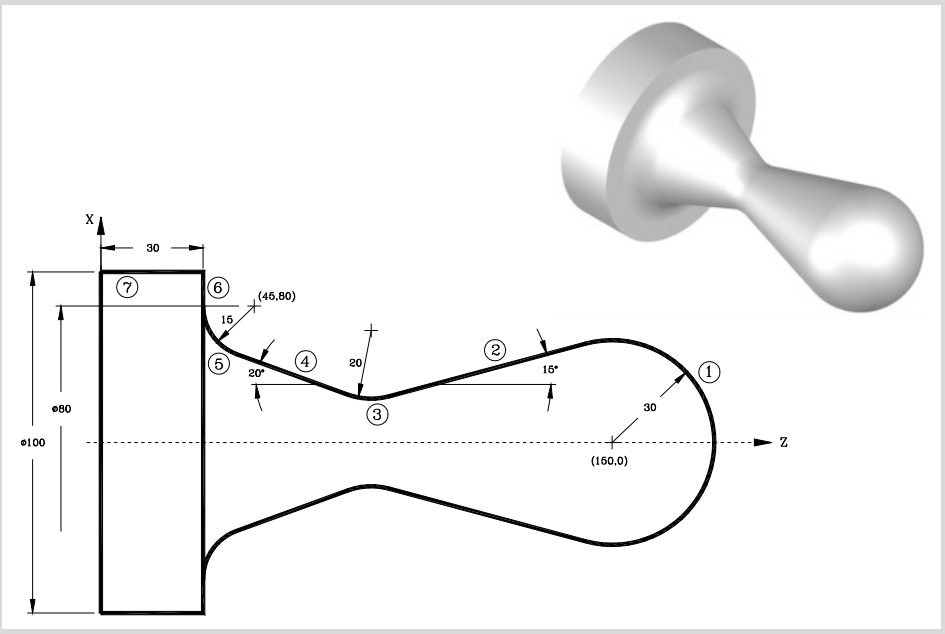
Profile definition.

Section.	Geometry.		
Starting point	Z = 170 X = 0		
Counterclockwise arc (1)	Center Z = 140 Center X = 0	Radius = 30	Tangency = Yes
Counterclockwise arc (2)	Radius = 350		Tangency = Yes
Clockwise arc (3)	Center Z = 50 Center X = 190	Radius = 30	Tangency = Yes
• The CNC shows all the possible options for section 2. Select the correct one.			
Straight (4)	Z = 20 X = 220		Tangency = Yes
• The CNC displays all the possible options between sections 3-4. Select the correct one.			
Straight (5)	Z = 0 X = 220		

End of editing

Select the "END" option and save the profile. The CNC quits the profile editor and inserts the profile in the part-program.

4.3 Example. Profile editor.



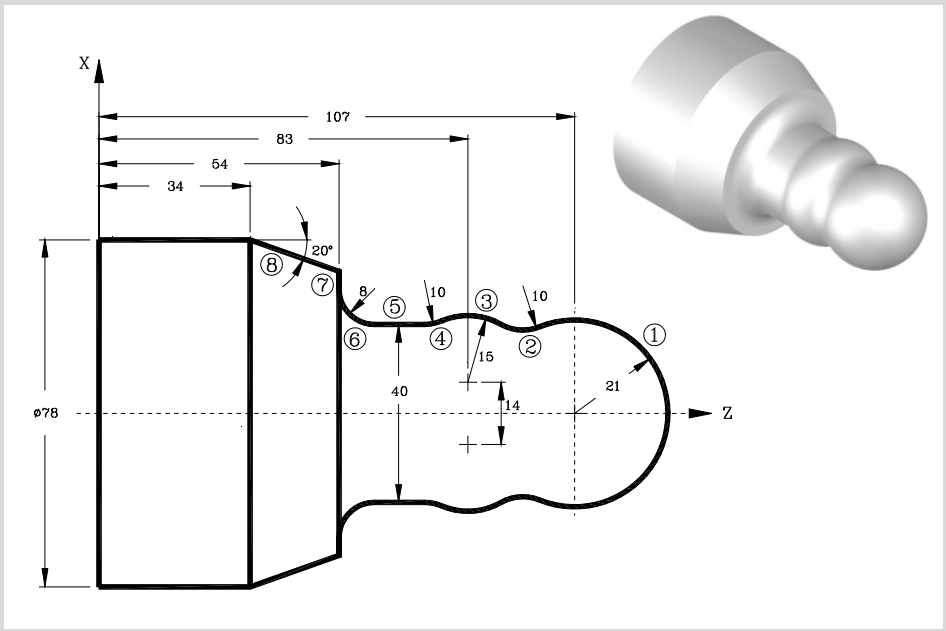
Profile definition.

Section.	Geometry.		
Starting point	Z = 180 X = 0		
Counterclockwise arc (1)	Center Z = 150 Center X = 0	Radius = 30	
Straight (2)		Angle = 195	Tangency = Yes
• The CNC displays all the possible options between sections 1-2. Select the correct one.			
Clockwise arc (3)		Radius = 20	Tangency = Yes
Straight (4)		Angle = 160	Tangency = Yes
Clockwise arc (5)	Z = 30 X = 80	Zcenter = 45 Xcenter = 80	Tangency = Yes
• The CNC displays all the possible options between sections 4-5. Select the correct one.			
• The CNC shows all the possible options for section 3. Select the correct one.			
Straight (6)	Z = 30 X = 100		
Straight (7)	Z = 0 X = 100		

End of editing

Select the "END" option and save the profile. The CNC quits the profile editor and inserts the profile in the part-program.

4.4 Example. Profile editor.



Profile definition.

Section.	Geometry.		
Starting point	Z = 128 X = 0		
Counterclockwise arc (1)	Center Z = 107 Center X = 0	Radius = 21	
Clockwise arc (2)	Radius = 10		Tangency = Yes
Counterclockwise arc (3)	Center Z = 83 Center X = 14	Radius = 15	Tangency = Yes
• The CNC shows all the possible options for section 2. Select the correct one.			
Clockwise arc (4)		Radius = 10	Tangency = Yes
Straight (5)	Z = 40	Angle = 180	Tangency = Yes
• The CNC shows all the possible options for section 4. Select the correct one.			
Clockwise arc (6)	Center Z = 74 Center X = 56	Radius = 8	Tangency = Yes
Straight (7)	Z = 54	Angle = 90	Tangency = Yes
Straight (8)	Z = 34 X = 78	Angle = 160	

End of editing

Select the "END" option and save the profile. The CNC quits the profile editor and inserts the profile in the part-program.

4.

PROFILE EDITOR

Example. Profile editor.

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

USE SUBROUTINES TO CREATE CANNED CYCLES.

5

5.1 Arithmetic subroutines and parameters.

A subroutine is a set of blocks that, once properly identified, may be called upon several times from another subroutine or from the program. Subroutines are normally used for defining a bunch of operations or movements that are repeated several times throughout the program.

Types of subroutines.

Local subroutines.

The local subroutine is defined as part of a program. This subroutine may only be called upon from the program where it has been defined.

Global subroutines.

The global subroutine is stored in CNC memory as an independent program. This subroutine may be called upon from any program or subroutine being executed.

OEM subroutines.

OEM subroutines, that are a special case of a global subroutine defined by the OEM. The machine manufacturer may define up to 30 subroutines per channel and associate them with functions G180 through G189 and G380 through G399 in such a way that when a channel executes one of these functions, it will execute the subroutine associated with that function for that channel.

Arithmetic parameters in the subroutines

Arithmetic parameters are general purpose variables that the user may utilize to create his/her own programs. Arithmetic parameters are programmed with the "P" code followed by the parameter number. The has some tables for consulting the value of these parameters; refer to the operating manual to learn how to handle these tables.

The CNC has global, local and common arithmetic parameters. The range of available parameters of each type is defined in the machine parameters.

- Local parameters.

  The CNC has global parameters (accessible from the program or any subroutine) and local parameters (accessible only from the program or subroutine where they have been programmed).

  The maximum range of local parameters is P0 to P99, the typical range being P0 to P25.

  Local parameters may be assigned to more than one subroutine up to 7 parameter nesting levels within the 20 subroutine nesting levels. Not all the subroutine call types change the nesting level; only the #CALL, #PCALL, #MCALL calls and functions G180 to G189 and G380 to G399.

- Global parameters. Global parameters will be shared by the program and the subroutines of the channel. They may be used in any block of the program and of the subroutine regardless of the nesting level they may be at.

  The maximum range of global parameters is P100 to P9999, the typical range being P100 to P299.

- Common parameters. Common parameters will be shared by the program and the subroutines of any channel. They may be used in any block of the program and of the subroutine regardless of the nesting level they may be at.

  The maximum range of common parameters is P10000 to P19999, the typical range being P10000 to P10999.

Location (path) of the global subroutines.

When calling a global subroutine its path (location) may be defined. When indicating the whole path, the CNC only looks for the subroutine in the indicated directory. If the path has not been indicated, the CNC looks for the subroutine in the following directories and in this order.

1 Directory selected with the #PATH instruction.

2 Directory of the program being executed.

3 Directory defined by machine parameter SUBPATH.

5.2 Assistance for subroutines.

5.2.1 Subroutine help files.

Help files may be associated with each OEM subroutine and each global subroutine called upon using #MCALL or #PCALL and they will be displayed while editing. Each subroutine may have two help files; a text file (txt) and an image file (bmp).

The help window is displayed while editing, after a blank space or tab following a G180-G189, G380-399 or following the name of the subroutine. The help window is only informative, it cannot be accessed with the cursor nor browse through it. When the help file is displayed, its text may be inserted into the part-program using the [INS] key. 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.

How to create 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. The name of the files must follow the following rule:

Subroutine.	Name of the help files.
G180-G189 G380-G399	The name of the files must be the function it is associated with. For example *G180.txt* y *G180.bmp*.
#MCALL #PCALL	The name of the files must be the name of the subroutine. For example *subroutine.txt* and *subroutine.bmp*.

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.

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

- 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.

The format of the text file may be the following.

```
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
```

Where to save the help files.

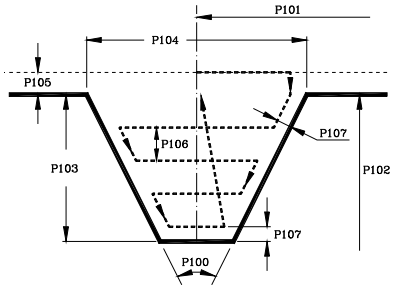
The machine manufacturer can save the help files in the folder ..\MTB\SUB\HELP\language. Since the modifications to the MTB directory in the "User" work mode disappear when turning the unit off, the user must save his help files in the folder ..\USERS\HELP\language. The CNC looks for the help files in the language that is selected; if the files are not there, the CNC will not show any help.

The CNC first looks for the files in the OEM folder and then in the user folder, that's why the user must not define subroutines and/or help files with the same name as those of the OEM. If both files have the same name, the CNC will first show the OEM files.

5.3 Example: Global subroutine. Machining of pulleys.

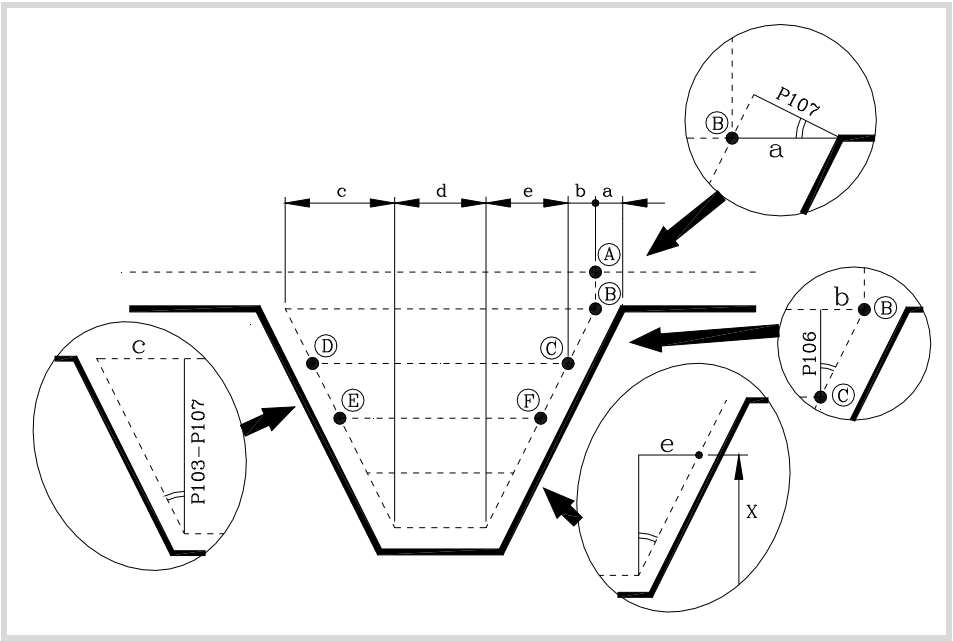
5.3.1 Define the subroutine.

The subroutine calling global parameters are:



P100 = Angle between sides of the pulley.
P101 = Absolute coordinate of the center of the pulley along Z.
P102 = Outside diameter of the pulley.
P103 = Depth of the groove (in radius)
^P104 = Width of the groove.
P105 = Safety distance.
P106 = Maximum depth of pass.
P107 = Finishing stock.
P108 = Cutting speed.
P109 = Roughing feedrate in mm/turn.
P110 = Finishing feedrate in mm/turn.

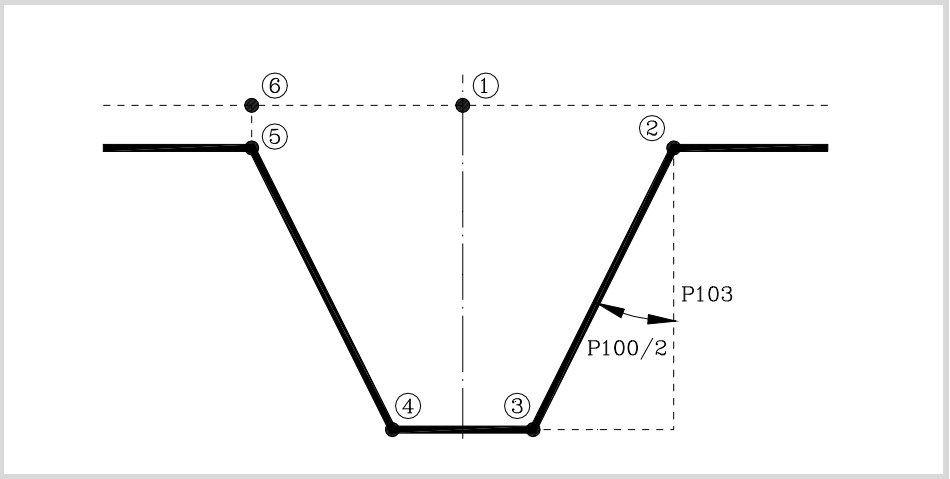
The necessary points for the roughing operation are:



Point.	X	Z
A	P102+2*P105	P101+(P104/2)-[P107/COS(P100/2)]
A-B	-2*P105	0
B-C	-2*P106	- P106*TANG(P100/2)
C-D	0	- (d+2e)
D-E	-2*P106	P106*TANG(P100/2)
E-F	0	d+2e

Section.	
a	P107/COS(P100/2)
b	P106*TANG(P100/2)
c	(P103 - P107)*TANG(P100/2)
d	P104 - 2a - 2c
e	[(x/2) - ((P102/2) - P103+P107)]*TANG(P100/2)

The necessary points for the finishing operation are:



Point.	X	Z
1	P102 + 2*P105	P101
2	P102	P101 + (P104/2)
3	P102 -2*P103	P101 + (P104/2) - P103*TANG(P100/2)
4	P102 -2*P103	P101 - (P104/2) + P103*TANG(P100/2)
5	P102	P101 - (P104/2)
6	P102 + 2*P105	P101 - (P104/2)

Program lines of the subroutine:

```
%PULLEY
$IF V.TM.NOSEW[1] > P104-2*[P107/COS[P100/2]]-2*[P103-P107]*TAN[P100/2]
$GOTO N10:
$ELSE
$GOTO N20:
$ENDIF
;
N10: #ERROR ["WRONG DATA"]
    (if cutter width > "d" => Error)
;
N20:
;————————————————————
; Roughing operation
;————————————————————
P115=FUP[[P103-P107]/P106]
    (Calculates Nr of passes (P115).)
P106=[[P103-P107]/P115]
    (Recalculates the pass (P106).)
G192 S500
G95 G96 FP109 SP108 T12 M4 M41
;
P1=P102+2*P105 P2=P101+[P104/2]-[P107/COS[P100/2]]-V.TM.NOSEW[1]
G0 G90 X P1 Z P2
    (Movement to point "A")

P1=2*P105
G1 G91 X-P1
    (Movement "A-B")

N50: P1=2*P106 P2=P106*TAN[P100/2]
X-P1 Z-P2
    (Movement "B-C")

P2=P104-2*P107/COS[P100/2]-2*[P103-P107]*TAN[P100/2]+2*[V.A.TIPTPOS.X/2-
[P102/2-P103+P107]]*TAN[P100/2]-V.TM.NOSEW[1]
Z-P2
    (Movement "C-D")
```


FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

```
P115=P115-1
    (Decrements number of passes)
$IF P115 <= 0
$GOTO N100
    (If all the passes have been run, finishing stage)
$ENDIF

P1=2*P106 P2=P106*TAN[P100/2]
X-P1 ZP2
    (Movement "D-E")

P2=P104-2*[P107/COS[P100/2]]-2*[P103-P107]*TAN[P100/2]+2*[V.A.TIPTPOS.X/2-
[P102/2-P103+P107]]*TAN[P100/2]-V.TM.NOSEW[1]
ZP2
    (Movement "E-F")

P115=P115-1
    (Decrements number of passes)
$IF P115 > 0
$GOTO N50
$ENDIF
    (If all the passes have been run, finishing stage)

;————————————————————
; FINISHING operation
;————————————————————
N100: G95 G96 FP110 SP108
P1=P102+2*P105
G0 G90 XP1 ZP101
    (Movement to point "1")

P2=P101+[P104/2]-V.TM.NOSEW[1]
G1 XP102 ZP2
    (Movement to point "2")

P1=P102-2*P103
P2=P101+[P104/2]-P103*TAN[P100/2]-V.TM.NOSEW[1]
X P1 Z P2
    (Movement to point "3")

P1=P102-2*P103
P2=P101-[P104/2]+P103*TAN[P100/2]
XP1 ZP2
    (Movement to point "4")

P2=P101-[P104/2]
XP102 ZP2
    (Movement to point "5")

P1=P102+2*P105 P2=P101-[P104/2]
X P1 Z P2
    (Movement to point "6")

P1=P102+2*P105
X P1 Z P101
    (Movement to point "1")

M17
```

5.3.2 Call to the subroutine from the part-program.

This examples assumes that the OEM has associated the global subroutine with function G180 using OEM parameters.

Program blocks.

```
%PROGRAM
T12 D1
G180 P100=100 P101=0 P102=80 P103=30 P104=80 P105=5 P106=3 P107=1
P108=100 P109=0.3 P110=0.1
#COMMENT BEGIN
---------- G180 ----------
P100 = Angle between sides of the pulley.
P101 = Absolute coordinate of the center of the pulley along Z.
P102 = Outside diameter of the pulley.
P103 = Depth of the groove (in radius)
^P104 = Width of the groove.
P105 = Safety distance.
P106 = Maximum depth of pass.
P107 = Finishing stock.
P108 = Cutting speed.
P109 = Roughing feedrate in mm/turn.
P110 = Finishing feedrate in mm/turn.
--------------------------------------
#COMMENT END
M30
```


FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

User notes:

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

User notes:

FAGOR

FAGOR AUTOMATION

Fagor Automation S. Coop.

Bº San Andrés, 19 - Apdo. 144
E-20500 Arrasate-Mondragón, Spain
Tel: +34 943 039 800
Fax: +34 943 791 712
E-mail: info@fagorautomation.es
www.fagorautomation.com

AENOR
ER
Empresa Registrada
ER-073/1994

CERTIFIED
IQNet
MANAGEMENT SYSTEM