



FAGOR 

REF. 0601

EXAMPLES MANUAL

Ref. 0601

All rights reserved. No part of this documentation may be copied, transcribed, stored in a data backup system or translated into any language without Fagor Automation's explicit consent.

The information described in this manual may be modified for technical reasons. FAGOR AUTOMATION S. COOP. Reserves the right to modify the contents of this manual without having to communicate such modifications.

Microsoft and Windows are registered trademarks of Microsoft Corporation USA. The other commercial trademarks belong to their respective owners.

The content of this manual and its validity for the product described here has been verified. Even so, involuntary errors are possible, thus no absolute match is guaranteed. Anyway, the contents of the manual is periodically checked making and including the necessary corrections in a future edition.

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

CHAPTER 1

CONTOURS

1.1	INTRODUCTION	1
1.2	CREATING A PART-PROGRAM	1
1.3	EXAMPLE: PROFILE MILLING USING 12 POINTS	2
1.3.1	POSITIONING 1	3
1.3.2	PROFILE 1	4
1.3.3	PROGRAM SIMULATION	5
1.4	EXAMPLE: PROFILE MILLING USING THE PROFILE EDITOR	6
1.4.1	PROFILE MILLING	7
1.4.2	PROGRAM SIMULATION	8

CHAPTER 2

MACHINING CANNED CYCLES

2.1	CREATING PROGRAMS WITH MACHINING CANNED CYCLES	9
2.2	EXAMPLE: USING MACHINING CANNED CYCLES	10
2.2.1	PROFILE MILLING	11
2.2.2	PROFILE MILLING (milling prior to drilling)	13
2.2.3	SIMPLE POCKET (top pocket)	15
2.2.4	SIMPLE POCKET (bottom pocket)	16
2.2.5	CIRCULAR POCKET 1	17
2.2.6	DRILLING DEFINITION	18
2.2.7	PROGRAM SIMULATION	21

CHAPTER 3

COORDINATE TRANSFORMATION CYCLES

3.1	EXAMPLE: USING THE MIRROR IMAGE CYCLE	24
3.1.1	PROFILE MILLING	25
3.1.2	RECTANGULAR POCKET	27
3.1.3	DRILLING 1 (Angular repetition)	28
3.1.4	DRILLING 1 (Linear repetition)	30
3.1.5	DRILLING 1 (angular repetition)	32
3.1.6	MIRROR IMAGE	34
3.1.7	PROGRAM SIMULATION	35
3.2	EXAMPLE: USING THE THREE MIRROR IMAGE COMMANDS	36
3.2.1	PROFILE MILLING	37
3.2.2	MIRROR IMAGES	39
3.2.3	PATTERN ROTATION	40
3.2.4	PROFILE MILLING	41
3.2.5	PROGRAM SIMULATION	43
3.3	EXAMPLE: PATTERN ROTATION	44
3.3.1	PROFILE MILLING	45
3.3.2	PATTERN ROTATION	47
3.3.3	CIRCULAR POCKETS	48
3.3.4	DRILLING WITH LINEAR REPETITION	49
3.3.5	PATTERN ROTATION	51
3.3.6	PROGRAM SIMULATION	52

CHAPTER 4

2D CYCLES

4.1	EXAMPLE: JOINT	53
4.1.1	2D PROFILE POCKET	54
4.1.2	CIRCULAR POCKETS	56
4.1.3	PROGRAM SIMULATION	57
4.2	EXAMPLE: CAM	58
4.2.1	2D PROFILE POCKETS	59
4.2.2	DRILLING	64
4.2.3	CIRCULAR POCKET 1	65
4.2.4	PROGRAM SIMULATION	66



CNC 8055
MC

REF. 0601

CHAPTER 5

3D CYCLES

5.1	INTRODUCTION	67
5.2	EXAMPLE: TOROID.....	68
5.2.1	3D PROFILE POCKETS.....	69
5.2.2	PROGRAM SIMULATION.....	73
5.3	EXAMPLE: IGLOO	74
5.3.1	3D PROFILE POCKETS.....	75
5.3.2	PROGRAM SIMULATION.....	78



CNC 8055
MC

REF. 0601

CONTOURS

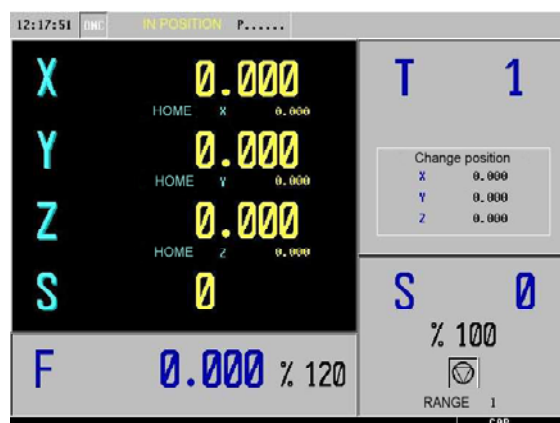
1

1.1 INTRODUCTION

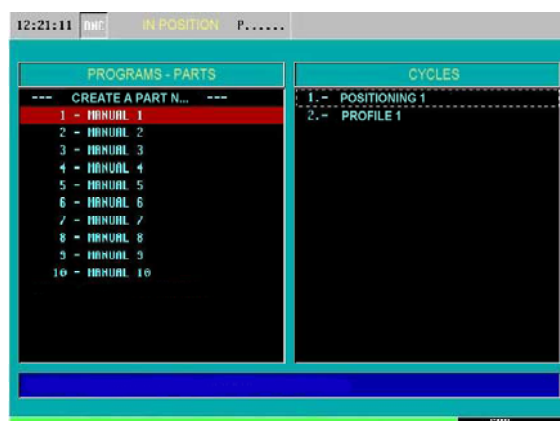
Programming in conversational mode consists in filling in a number of cycles depending the machining operations to perform. These cycles may be stored in a program or executed without storing them. Here are some examples of machining operations normally used with this method.

1.2 CREATING A PART-PROGRAM

To create a part-program, we will proceed as follows starting from the main screen:



Pressing the [PPROG] key gives access to file management. Once on this screen, we move the red cursor to "CREATE NEW PART" and press [PPROG]. At this time, the CNC requests a part number and a comment. We confirm both data with the [ENTER] key.

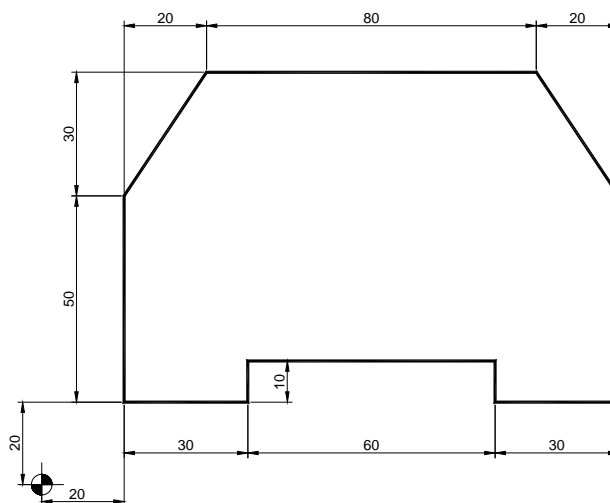


CNC 8055
MC

REF. 0601

1.3 EXAMPLE: PROFILE MILLING USING 12 POINTS

To machine the contour of this figure:



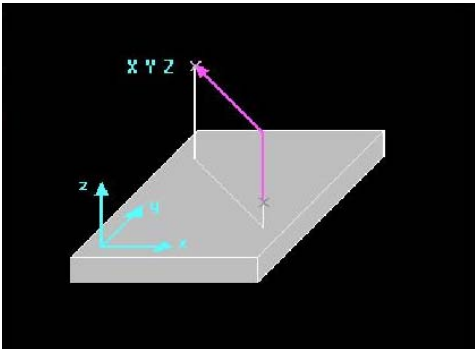
We will use the following tools:

Operations	Tools
Positioning	
Outside profile machining (roughing)	Flat endmill Ø6 T1 D1
Outside profile machining (finishing)	Flat endmill Ø4 T2 D2

After creating the part-program, we will make this part following these steps:

1.3.1 POSITIONING 1

To go into the POSITIONING cycle. we press:



There are two levels in the POSITIONING CYCLE, they may be toggled pressing [LEVEL CYCLE] or [PAGEUP/PAGEDOWN]. The first level offers the choice of programming the positioning in two movements:

- 1. Z XY
- 2. XY Z
- 3. STRAIGHT XYZ

These three options may be changed using the [TWO-COLOR] key and are confirmed with the [ENTER] key. Once the movement type has been chosen, we indicate each coordinate value in the corresponding box always confirming each value with [ENTER]:

GENERAL CONDITIONS		
	Positioning first in Z and then in the XY plane.	
	Rapid feedrate	
X	Positioning on the X axis	0
Y	Positioning on the Y axis	0
Z	Positioning on the Z axis	20

MACHINING CONDITIONS		
F	Machining feedrate	1000
S	RPM	1000
	Spindle clockwise	
T	Number of the tool to be used	1
D	Tool offset	1

Once the cycle has been filled in, we press [PPROG] to insert it in the previously created program. Once on the PART-PROGRAMS screen, the bottom of the CNC screen shows the following message:

SELECT POSITION TO INSERT POSITIONING 1

We press [ENTER] to insert the cycle in the program:



1.

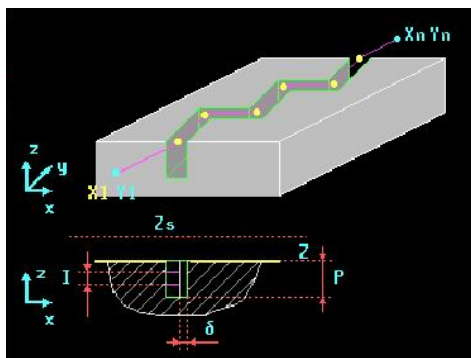
CONTOURS
EXAMPLE: PROFILE MILLING USING 12 POINTS



CNC 8055
MC

1.3.2 PROFILE 1

After inserting the POSITIONING 1 cycle, we fill in the PROFILE 1 cycle. To do that, we must press [F3].



The first data we must enter are the starting (initial) X and Y. Press [ENTER] to confirm each data:

STARTING POSITION		
X1	Starting point in X	80
Y1	Starting point in Y	0

This starting point corresponds to the positioning prior to the contour's entry coordinate where the radius of the tool being used is compensated for.

In the next window, we fill in the different geometry points, up to 12 coordinates; we could add a corner rounding, or a chamfer at each point using the [TWO-COLOR] key:

PROFILE PROGRAMMING

P1	X80	Y30	r 10	P7	X40	Y100	
P2	X110	Y30		P8	X20	Y70	
P3	X110	Y20		P9	X20	Y20	
P4	X140	Y20		P10	X50	Y20	
P5	X140	Y70		P11	X50	Y30	
P6	X120	Y100		P12	X80	Y30	r 10

When using less than 12 geometry points, we must repeat the last coordinate to indicate to the control that the contour has no more points.

After defining the geometry points, we fill in the section for the general conditions of the cycle:

GENERAL CONDITIONS		
Xn	Last point where the tool must return along the X axis.	80
Yn	Last point where the tool must return along the Y axis.	0
Zs	Safety coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	20
I	Value of each pass	3
Fz	In feed	100

In the third part of the cycle, we program the machining conditions, both for roughing and finishing also including the tools:




1.

CONTOURS
EXAMPLE: PROFILE MILLING USING 12 POINTS



CNC 8055
MC

REF. 0601

ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	1
D	Tool offset	1
	Right hand tool radius compensation	
FINISHING		
F	Finishing feedrate	1000
S	RPM	1500
	Clockwise turning direction	
T	Number of the tool to be used	2
D	Tool offset	2
δ	Finishing stock	1

If we do not wish to perform any of these two operations, we just program zero values in the corresponding section. Tool compensation is applied in the roughing section. To change this window, we can press the [TWO-COLOR] key.

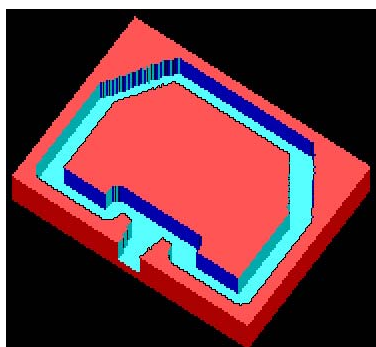
Once the cycle is completed, we proceed the same way as with the positioning cycle; i.e. pressing [PPROG] to insert it in the desired position of the program and pressing [ENTER].

1.3.3 PROGRAM SIMULATION

1. We position the red cursor on the first cycle of the program, in this case POSITIONING 1, and then press [GRAPHICS]. It then displays the simulation screen:



2. We press [RESET] and then [START].



1.

CONTOURS
EXAMPLE: PROFILE MILLING USING 12 POINTS

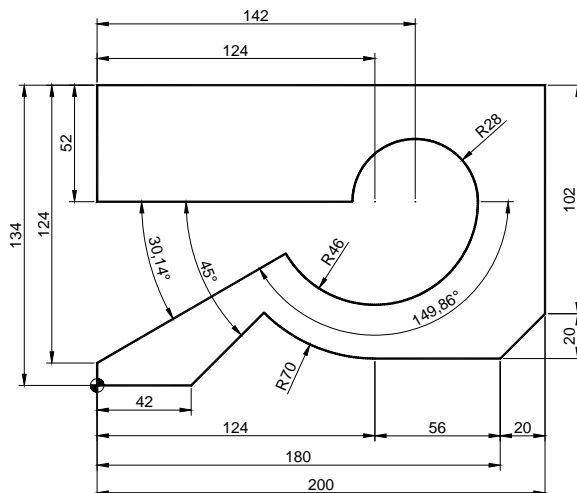
FAGOR 

CNC 8055
MC

REF. 0601

1.4 EXAMPLE: PROFILE MILLING USING THE PROFILE EDITOR

We will carry out a machining operation similar to the previous one using the PROFILE MILLING cycle



We will use the following tools:

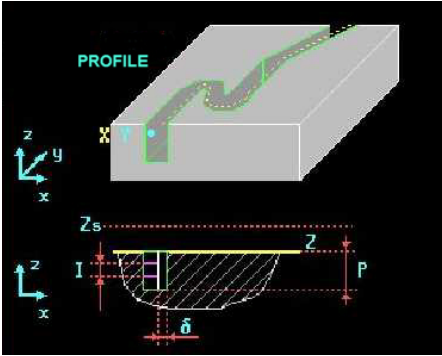
Operations	Tools
Positioning	
Profile milling (roughing)	Flat endmill Ø6 T1 D1
Profile milling (finishing)	Flat endmill Ø4 T2 D2

After creating the part-program and the positioning cycle like in the previous example, we make this part by following these steps:

1.4.1 PROFILE MILLING

After inserting the POSITIONING 1 cycle, we fill in the PROFILE MILLING cycle. To do that, we must press [F3]. Then, we press [LEVEL CYCLE] to switch from level 1 to level 2, where we can draw the profile to be contoured using the drawing assistant PROFILE EDITOR.

The advantage of the PROFILE MILLING cycle is that we draw the geometry directly and, therefore, the number of points is unlimited, which does not happen with the first level that only allows a maximum of 12 points.



The first data we must enter is the starting (initial) X and Y, in this case:

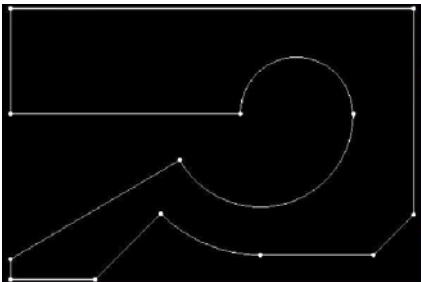
STARTING POSITION		
X	Starting point on the X axis	-25
Y	Starting point on the Y axis	-25
	Profile	1

We assign a number to the drawing to be made in the "PROFILE" box and press [RECALL] This key gives access to the drawing screen PROFILE EDITOR. Once on this screen, we draw the desired geometry.

PROFILE PROGRAMMING

STARTING POINT	X1: 0	Y1: 0		
STRAIGHT	X2: 42	Y2: 0		
STRAIGHT	α : 45			
COUNTERCLOCKWISE ARC	X2: 124	Y2: 12	XC: 124	YC: 82
STRAIGHT	X2: 180	Y2: 12		
STRAIGHT	X2: 200	Y2: 32		
STRAIGHT	X2: 200	Y2: 134		
STRAIGHT	X2: 0	Y2: 134		
STRAIGHT	X2: 0	Y2: 82		
STRAIGHT	X2: 114	Y2: 82		
CLOCKWISE ARC	X2: 170	Y2: 82	XC: 142	YC: 82
CLOCKWISE ARC	XC: 124	YC: 82	R: 46	TANG: YES
STRAIGHT	X2: 0	Y2: 10	α : 30.14	
STRAIGHT	X2: 0	Y2: 0		

Geometry of the example in the PROFILE EDITOR



1.

CONTOURS
EXAMPLE: PROFILE MILLING USING THE PROFILE EDITOR






CNC 8055
MC

REF. 0601

When the drawing is completed, we press [END] and the CNC will request a comment for that drawing. Once we confirmed this comment pressing [ENTER], we go back to the PROFILE MILLING cycle to fill in the rest of the data.

GENERAL CONDITIONS		
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	20
I	Value of each pass	3
Fz	In feed	100

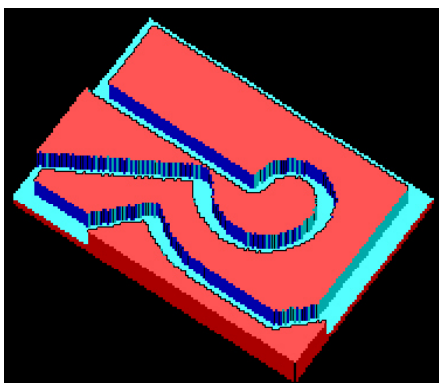
ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	1
D	Tool offset	1
	Right hand tool radius compensation	

FINISHING		
F	Finishing feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	2
D	Tool offset	2
δ	Finishing stock	0.2

Once the cycle is completed, we proceed the same way as with the positioning cycle; i.e. pressing [PPROG] to insert it in the desired position of the program and pressing [ENTER].

1.4.2 PROGRAM SIMULATION

1. We position the red cursor on the first cycle of the program, in this case POSITIONING 1, and then press [GRAPHICS].
2. Then, we press [RESET] and then [START].



MACHINING CANNED CYCLES

2

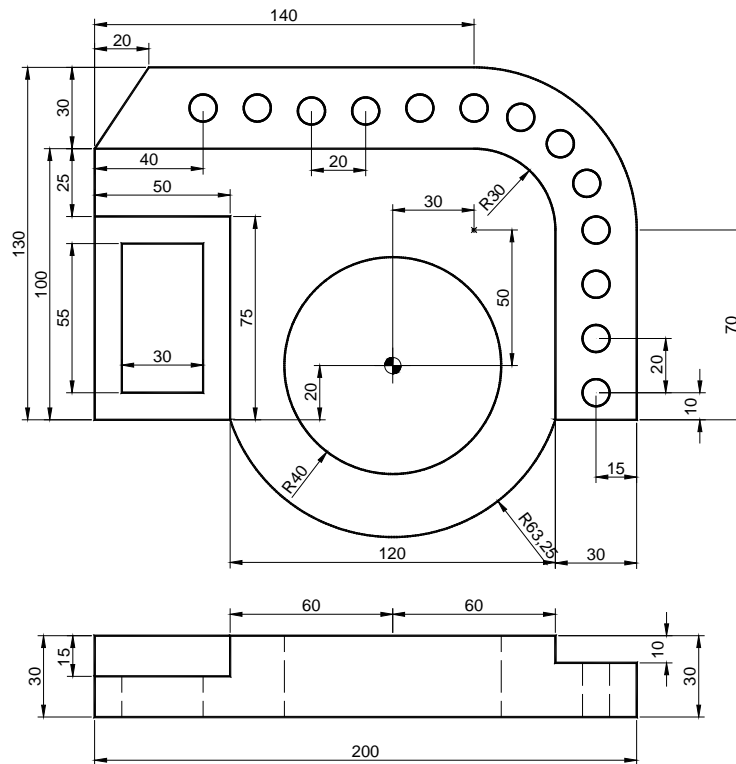
2.1 CREATING PROGRAMS WITH MACHINING CANNED CYCLES

A canned cycle is a machining operation that is inside the outside geometry of the part, requires a roughing operation and sometimes also a finishing operation. The conversational mode offers the following cycles:

- Surface milling
- Slot milling
- 2D and 3D pockets
- Rectangular and circular boss
- Rectangular and circular pocket
- Boring
- Reaming
- Tapping
- Drilling

2.2 EXAMPLE: USING MACHINING CANNED CYCLES

To make the following part:



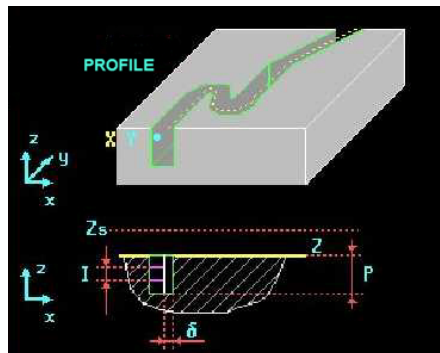
We will use the following tools:

Operations	Tools
Outside profile machining (roughing)	Table Ø50 T3 D3
Outside profile machining (finishing)	Flat endmill Ø4 T2 D2
Milling before drilling	End mill Ø30 T6 D6
Machining of the pockets.	Flat endmill Ø6 T1 D1
Drilling	Drill bit Ø10 T4 D4

After creating the part-program and performing the positioning, we make this part by following these steps:

2.2.1 PROFILE MILLING

To fill in the PROFILE MILLING cycle, we press [F3]. Then, we press [LEVEL CYCLE] to switch from level 1 to level 2, where we can draw the profile to be contoured using the PROFILE EDITOR.



We select the starting point in XY coordinates, assign a number to the drawing and press [RECALL]:

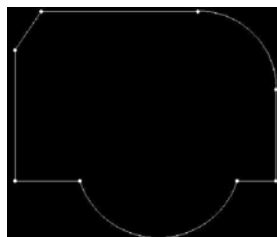
STARTING POSITION		
X	Starting point on the X axis	-140
Y	Starting point on the Y axis	-50
	Profile	2

Once on the screen of the PROFILE EDITOR, we draw the desired geometry:

PROFILE PROGRAMMING

STARTING POINT	X1: -110	Y1: -20		
STRAIGHT	X2: -60	Y2: -20		
COUNTERCLOCKWISE	X2: 60	Y2: -20	XC: 0	YC: 0
ARC				
STRAIGHT	X2: 90	Y2: -20		
STRAIGHT	X2: 90	Y2: 50		
COUNTERCLOCKWISE	X2: 30	Y2: 110	XC: 30	YC: 50
ARC				
STRAIGHT	X2: -90	Y2: 110		
STRAIGHT	X2: -110	Y2: 80		
STRAIGHT	X2: -110	Y2: -20		

Geometry of the example in the PROFILE EDITOR



After drawing the contour of the figure, we save the drawing in the PROFILE MILLING cycle and fill in the rest of the data:

GENERAL CONDITIONS		
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	30
I	Value of each pass	3
Fz	In feed	100

2.

MACHINING CANNED CYCLES
EXAMPLE: USING MACHINING CANNED CYCLES




FAGOR

CNC 8055
MC

REF. 0601

2.

MACHINING CANNED CYCLES
EXAMPLE: USING MACHINING CANNED CYCLES

ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	3
D	Tool offset	3
	Right hand tool radius compensation	
FINISHING		
F	Finishing feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	2
D	Tool offset	2
δ	Finishing stock	0.2



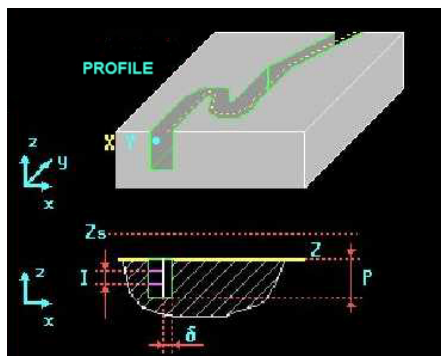
CNC 8055
MC

REF. 0601

2.2.2 PROFILE MILLING (milling prior to drilling)

To mill for drilling, we use the same cycle as for the outside contour, but this time you draw the path of the tool center for that previous milling.

Same as before, to fill in the PROFILE MILLING cycle, we press [F3]. Then, we press [LEVEL CYCLE] to switch from level 1 to level 2, where we can draw the profile to be contoured using the PROFILE EDITOR.



We select the starting point in XY coordinates, assign a number to the drawing and press [RECALL]:

STARTING POSITION		
X	Starting point on the X axis	-140
Y	Starting point on the Y axis	95
	Profile	3

Once on the screen of the PROFILE EDITOR, we draw the desired geometry:

PROFILE PROGRAMMING

STARTING POINT	X1: -110	Y1: 95		
STRAIGHT	X2: 30	Y2: 95		
CLOCKWISE ARC	X2: 75	Y2: 50	XC: 30	YC: 50
STRAIGHT	X2: 75	Y2: -20		

Geometry of the example in the PROFILE EDITOR



After drawing the contour of the figure, we save the drawing in the PROFILE MILLING cycle. Then, besides filling in the rest of data, we will have to indicate that the tool is not to be compensated:

GENERAL CONDITIONS		
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	10
I	Value of each pass	3
Fz	In feed	100

2.

MACHINING CANNED CYCLES
EXAMPLE: USING MACHINING CANNED CYCLES



FAGOR


CNC 8055
MC

REF. 0601

2.

MACHINING CANNED CYCLES
EXAMPLE: USING MACHINING CANNED CYCLES

ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	6
D	Tool offset	6
	Without tool compensation	

FINISHING		
F	Finishing feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	6
D	Tool offset	6
δ	Finishing stock	0.2

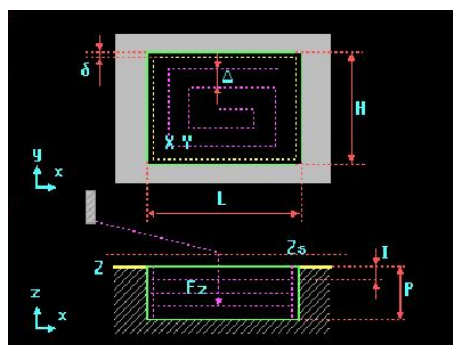


CNC 8055
MC

REF. 0601

2.2.3 SIMPLE POCKET (top pocket)

To fill in the SIMPLE POCKET cycle, we press [F7].



The position of the pocket may be defined in two ways, from the bottom left corner, or from the center; the [TWO-COLOR] key toggles this option.

GENERAL CONDITIONS		
	Corner where the machining operation begins	
X	Starting point on the X axis	-110
Y	Starting point on the Y axis	-20
L	Total length in X	50
H	Total length in Y	75
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	15
I	Value of each pass	3
Fz	In feed	100

MACHINING CONDITIONS		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	1
D	Tool offset	1
	Clockwise machining	
Δ	Finishing pass	0
δ	Finishing stock	0.2

2.

MACHINING CANNED CYCLES
EXAMPLE: USING MACHINING CANNED CYCLES

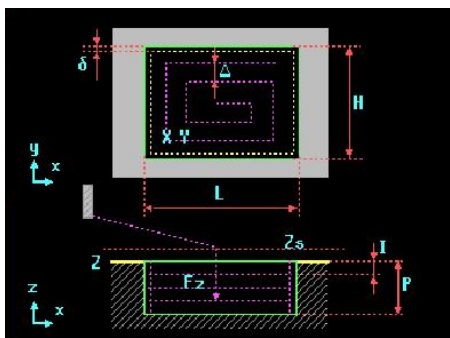
FAGOR

CNC 8055
MC

REF. 0601

2.2.4 SIMPLE POCKET (bottom pocket)

This second pocket is programmed the same way as the previous one.

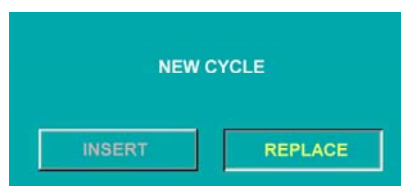


The dimensions of this pocket are different from the previous one, therefore, the previous one may be changed by simply recalling it from the program using the [RECALL] key and changing the starting X and Y, the length L and the width H:

GENERAL CONDITIONS		
	Corner where the machining operation begins	
X	Starting point on the X axis	-100
Y	Starting point on the Y axis	-10
L	Total length in X	30
H	Total length in Y	55
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	30
I	Value of each pass	3
Fz	In feed	100

MACHINING CONDITIONS		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	1
D	Tool offset	1
	Clockwise machining	
Δ	Finishing pass	0
δ	Finishing stock	0.2

After this, we press the [PROG] key and the CNC will offer the choice to replace the cycle or inserted underneath. To change sides, we press the cursor arrows:



2.

MACHINING CANNED CYCLES
EXAMPLE: USING MACHINING CANNED CYCLES

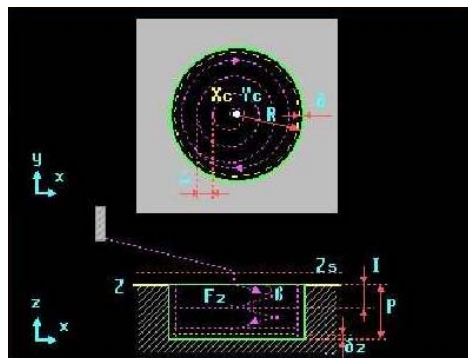


CNC 8055
MC

REF. 0601



2.2.5 CIRCULAR POCKET 1



To go into the CIRCULAR POCKET 1 cycle, we press [F7]. Then, we press [LEVEL CYCLE] until reaching level 3.



The CIRCULAR POCKET 1 cycle always begins at its center. We filled in the cycle data the same way as in the previous cycles:

GENERAL CONDITIONS		
Xc	Pocket center in X	0
Yc	Pocket center in Y	0
R	Pocket radius	40
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	30
I	Value of each pass	3
Fz	In feed	100

ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	1
D	Tool offset	1
β	Penetrating angle	90
	Clockwise machining	
Δ	Roughing pass	0

FINISHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	2
D	Tool offset	2
θ	Tool penetrating angle when finishing	90
	Clockwise machining	
δ	Finishing stock	0.2
δz	Excess material in Z	0.1
N	Number of finishing passes in Z	1

2.

MACHINING CANNED CYCLES
EXAMPLE: USING MACHINING CANNED CYCLES

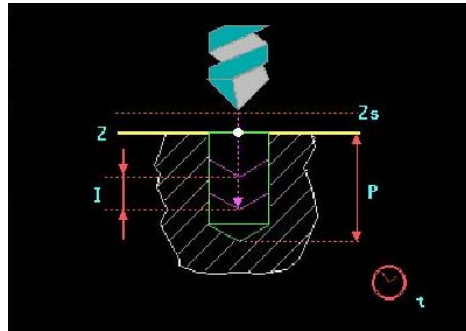
FAGOR 

CNC 8055
MC

REF. 0601


2.2.6 DRILLING DEFINITION

To go into the DRILLING 1 cycle, we press:



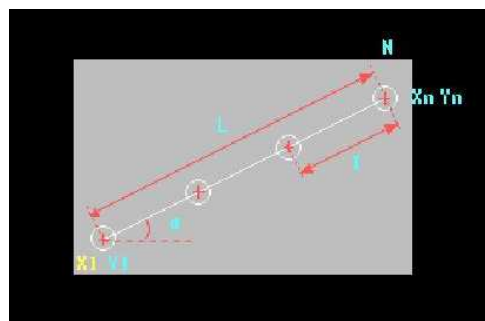
When defining the drilling, we begin from the holes at the bottom right side of the part. The first hole is drilled at X75 Y-10

GENERAL CONDITIONS		
X	X coordinate of the first hole	75
Y	Y coordinate of the first hole	-10
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	30
I	Value of each pass	5
t	Dwell at the bottom	0


PENETRATION		
F	Machining feedrate	600
S	RPM	750
	Clockwise turning direction	
T	Number of the tool to be used	4
D	Tool offset	4




Once the first hole is programmed, we repeat it linearly pressing the [LINEAR REPETITION] key.



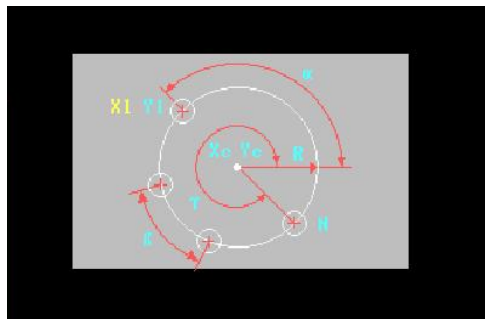
In this cycle, we can enter the necessary data for the linear repetition of different shapes by simply changing the window mode. To toggle this window, we press the [TWO-COLOR] key:

GENERAL CONDITIONS		
X1	X coordinate of the first hole	75
Y1	Y coordinate of the first hole	-10
	Linear repetition method 1	
Xn	X coordinate of the last hole	75
Yn	Y coordinate of the last hole	30
N	Number of positions	3


DRILLING		
Zs	Safety Z coordinate	2
Z	Surface Z coordinate	0
P	Total pocket depth	30
t	Dwell at the bottom (seconds)	0
I	Penetration step	5
F	Machining feedrate	600
S	RPM	750
	Clockwise turning direction	
T	Number of the tool to be used	4
D	Tool offset	4



Once the linear drilling has been programmed, we program the angular drilling. To do this, we press the [POLAR REPETITION] key.



In this cycle, we can also change the data that the CNC needs depending on what we (the programmers) have, using the [TWO-COLOR] key.

GENERAL CONDITIONS		
X1	X coordinate of the first hole	75
Y1	Y coordinate of the first hole	50
	Angular repetition method 1	
Xc	X coord. of the arc center	30
Yc	Y coord. of the arc center	50
N	Number of positions	4
τ	Angular distance of the last cycle, referred to the horizontal axis X	67.5

2.

MACHINING CANNED CYCLES
EXAMPLE: USING MACHINING CANNED CYCLES


FAGOR 

CNC 8055
MC

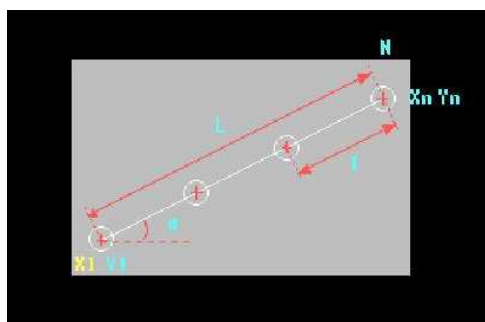
REF. 0601


2.


MACHINING CANNED CYCLES
EXAMPLE: USING MACHINING CANNED CYCLES

DRILLING		
Zs	Safety Z coordinate	2
Z	Surface Z coordinate	0
P	Total pocket depth	30
t	Dwell at the bottom (seconds)	0
I	Penetration step	5
F	Machining feedrate	600
S	RPM	750
	Clockwise turning direction	
T	Number of the tool to be used	4
D	Tool offset	4

The last row of holes is programmed the same way as the first one; i.e. using the [LINEAR REPETITION] key.



GENERAL CONDITIONS		
X1	X coordinate of the first hole	30
Y1	Y coordinate of the first hole	95
	Linear repetition method 1	
Xn	X coordinate of the last point	-70
Yn	Y coordinate of the last point	95
N	Number of positions	6

MACHINING		
Zs	Safety Z coordinate	2
Z	Surface Z coordinate	0
P	Total pocket depth	30
t	Dwell at the bottom (seconds)	0
I	Penetration step	5
F	Machining feedrate	600
S	RPM	750
	Clockwise turning direction	
T	Number of the tool to be used	4
D	Tool offset	4

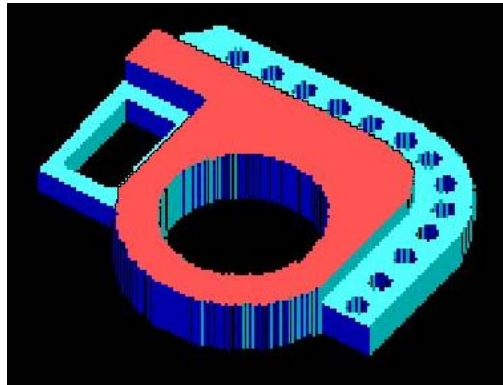


CNC 8055
MC

REF. 0601

2.2.7 PROGRAM SIMULATION

1. We position the red cursor on the first cycle of the program and then press [GRAPHICS].
2. Then, we press [RESET] and then [START].



2.

MACHINING CANNED CYCLES
EXAMPLE: USING MACHINING CANNED CYCLES

FAGOR 

**CNC 8055
MC**

REF. 0601

2.

MACHINING CANNED CYCLES

EXAMPLE: USING MACHINING CANNED CYCLES



CNC 8055
MC

REF. 0601

COORDINATE TRANSFORMATION CYCLES

3

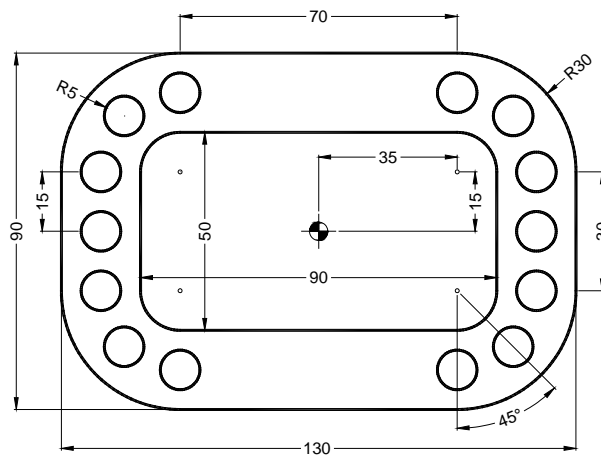
Coordinate transformation cycles are those where, based on an original program, a function is applied to repeat the same geometry differently. There are four types of functions for coordinate transformation:

- Mirror image.
- Scaling cycle
- Pattern rotation
- Part zero offset cycle

In conversational mode, these cycles are within the ISO cycle and may be toggled using the [LEVEL CYCLE] key.

3.1 EXAMPLE: USING THE MIRROR IMAGE CYCLE

The following example uses the mirror image cycle for drilling.



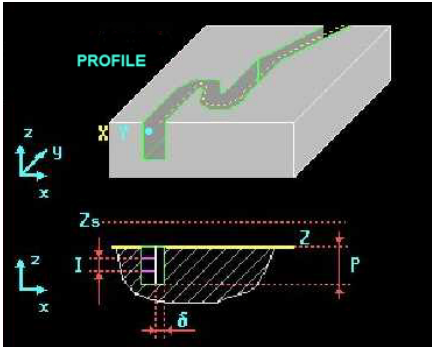
We will use the following tools:

Operations	Tools
Milling of the outside profile	Table Ø50 T3 D3
Inside rectangular pocket	End mill Ø12 T5 D5
Right holes	Drill bit Ø10 T4 D4

After creating the part-program and performing the positioning, we make this part by following these steps:

3.1.1 PROFILE MILLING

To fill in the PROFILE MILLING cycle, we press [F3]. Then, we press [LEVEL CYCLE] to switch from level 1 to level 2, where we can draw the profile to be contoured using the PROFILE EDITOR.



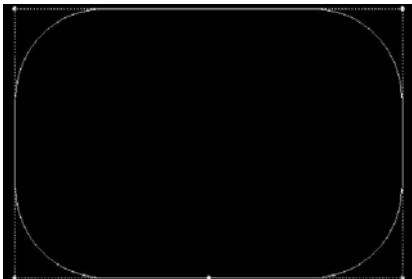
We select the starting point in XY coordinates, assign a number to the drawing and press [RECALL]:

STARTING POSITION		
X	Starting point on the X axis	0
Y	Starting point on the Y axis	-75
	Profile	4

Once on the screen of the PROFILE EDITOR, we draw the desired geometry:

PROFILE PROGRAMMING					
STARTING POINT	X1: 0	Y1: -45			
STRAIGHT	X2: 35	Y2: -45			
COUNTERCLOCKWISE ARC	X2: 65	Y2: -15	XC: 35	YC: -15	
STRAIGHT	X2: 65	Y2: 15			
COUNTERCLOCKWISE ARC	X2: 35	Y2: 45	XC: 35	YC: 15	
STRAIGHT	X2: -35	Y2: 45			
COUNTERCLOCKWISE ARC	X2: -65	Y2: 15	XC: -35	YC: 15	
STRAIGHT	X2: -65	Y2: -15			
COUNTERCLOCKWISE ARC	X2: -35	Y2: -45	XC: -35	YC: -15	
STRAIGHT	X2: 0	Y2: -45			

Geometry of the example in the PROFILE EDITOR



3.



COORDINATE TRANSFORMATION CYCLES
EXAMPLE: USING THE MIRROR IMAGE CYCLE




CNC 8055
MC

After drawing the contour of the figure, we save the drawing in the PROFILE MILLING cycle and fill in the rest of the data:

GENERAL CONDITIONS		
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	20
I	Value of each pass	2
Fz	In feed	100

ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	3
D	Tool offset	3
	Right hand tool radius compensation	

FINISHING		
F	Finishing feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	2
D	Tool offset	2
δ	Finishing stock	0.2

3.

COORDINATE TRANSFORMATION CYCLES
EXAMPLE: USING THE MIRROR IMAGE CYCLE

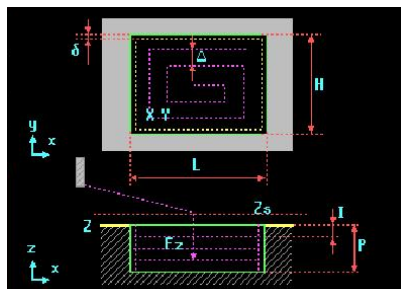


CNC 8055
MC







REF. 0601

3.1.2 RECTANGULAR POCKET

To fill in the RECTANGULAR POCKET cycle, we press [F7]. Then, press [LEVEL CYCLE] until reaching level 2.



This cycle offers the chance to round the corners with a radius larger than that of the tool being used or to chamfer them.

GENERAL CONDITIONS		
	Position where the machining operation begins	
X	Starting point on the X axis	0
Y	Starting point on the Y axis	0
L	Total length in X	90
H	Total length in Y	50
τo	Pocket inclination angle	0
	Corner finishing	
r	Corner blending radius	10
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	20
I	Value of each pass	3
Fz	In feed	100
ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	5
D	Tool offset	5
β	Penetrating angle	90
	Clockwise machining	
Δ	Roughing pass	0
FINISHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	2
D	Tool offset	2
θ	Tool penetrating angle when finishing	90
	Clockwise machining	
δ	Finishing stock	0.2
δz	Excess material in Z	0.1
N	Number of finishing passes in Z	1

3.

COORDINATE TRANSFORMATION CYCLES
EXAMPLE: USING THE MIRROR IMAGE CYCLE

FAGOR 

CNC 8055
MC

REF. 0601

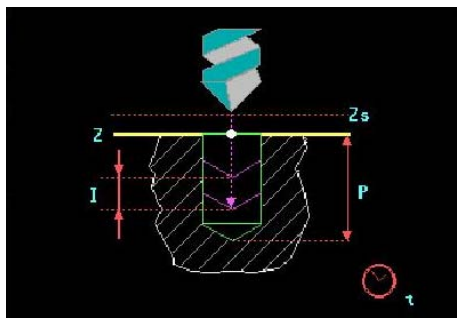
3.1.3 DRILLING 1 (Angular repetition)

To go into the DRILLING 1 cycle, we press:




3.

COORDINATE TRANSFORMATION CYCLES
EXAMPLE: USING THE MIRROR IMAGE CYCLE



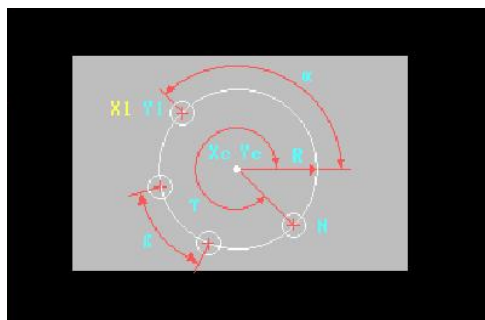
When defining the drilling, we begin from the hole at the bottom right side of the part. This hole will be positioned at X35 Y-35:

GENERAL CONDITIONS		
X	X coordinate of the first hole	35
Y	Y coordinate of the first hole	-35
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	20
I	Value of each pass	5
t	Dwell at the bottom	0

PENETRATION		
F	Machining feedrate	600
S	RPM	750
	Clockwise turning direction	
T	Number of the tool to be used	4
D	Tool offset	4




Once the first hole is programmed, we repeat it in an arc pressing the [ANGULAR REPETITION] key.




CNC 8055
MC

REF. 0601

In this cycle, we can enter the necessary data for the angular repetition of different shapes by simply changing the window mode. To toggle this window, we press the [TWO-COLOR] key:

GENERAL CONDITIONS		
X1	X coordinate of the first hole	35
Y1	Y coordinate of the first hole	-35
	Angular repetition method 1	
Xc	X coord. of the arc center	35
Yc	Y coord. of the arc center	-15
N	Number of positions	2
τ	Angular distance of the last cycle, referred to the horizontal axis X	-45

MACHINING		
Zs	Safety Z coordinate	2
Z	Surface Z coordinate	0
P	Total pocket depth	20
t	Dwell at the bottom (seconds)	0
I	Penetration step	5
F	Machining feedrate	600
S	RPM	750
	Clockwise turning direction	
T	Number of the tool to be used	4
D	Tool offset	4

3.

COORDINATE TRANSFORMATION CYCLES
EXAMPLE: USING THE MIRROR IMAGE CYCLE

FAGOR 

CNC 8055
MC

REF. 0601

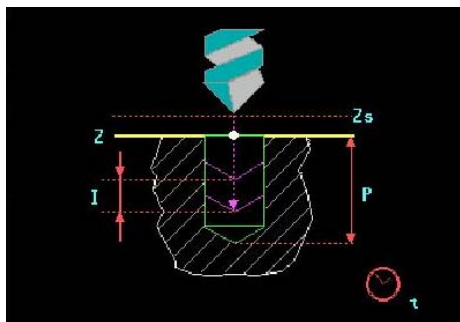
3.1.4 DRILLING 1 (Linear repetition)

Like before, to go into the DRILLING 1 cycle, we press:




3.

COORDINATE TRANSFORMATION CYCLES
EXAMPLE: USING THE MIRROR IMAGE CYCLE



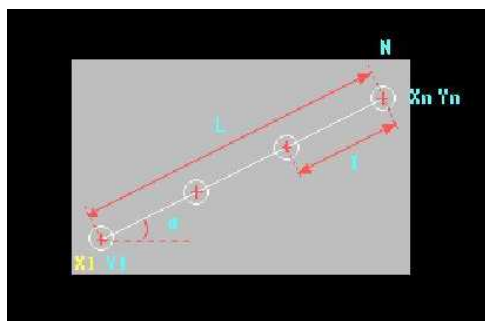
The drill bit will position at the first hole, X55 Y-15, and we will then do a linear repetition.

GENERAL CONDITIONS		
X	X coordinate of the first hole	55
Y	Y coordinate of the first hole	-15
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	20
I	Value of each pass	5
t	Dwell at the bottom	0

PENETRATION		
F	Machining feedrate	600
S	RPM	750
	Clockwise turning direction	
T	Number of the tool to be used	4
D	Tool offset	4




Then, we define a linear repetition by pressing the [LINEAR REPETITION] key.




CNC 8055
MC

REF. 0601

We program the linear repetition of this second hole by defining the XY coordinate of the last hole of the repetition and the number of holes:

GENERAL CONDITIONS		
X1	X coordinate of the first hole	55
Y1	Y coordinate of the first hole	-15
	Linear repetition method 1	
Xn	X coordinate of the last hole	55
Yn	Y coordinate of the last hole	0
N	Number of positions	2

DRILLING		
Zs	Safety Z coordinate	2
Z	Surface Z coordinate	0
P	Total pocket depth	20
t	Dwell at the bottom (seconds)	0
I	Penetration step	5
F	Machining feedrate	600
S	RPM	750
	Clockwise turning direction	
T	Number of the tool to be used	4
D	Tool offset	4

3.

COORDINATE TRANSFORMATION CYCLES
EXAMPLE: USING THE MIRROR IMAGE CYCLE

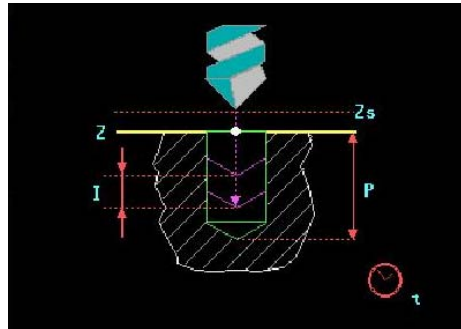
FAGOR 

CNC 8055
MC

REF. 0601


3.1.5 DRILLING 1 (angular repetition)

To go into the DRILLING 1 cycle, we press:



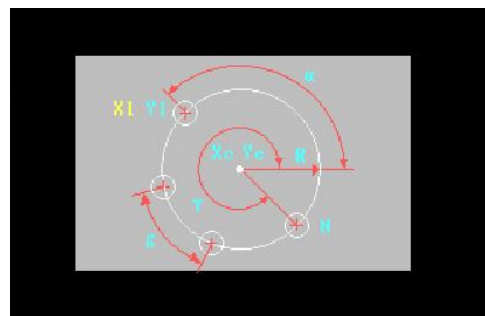
The drill bit will position at the third hole, X55 Y15, and we will then do an angular repetition.

GENERAL CONDITIONS		
X	X coordinate of the first hole	55
Y	Y coordinate of the first hole	15
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	20
I	Value of each pass	5
t	Dwell at the bottom	0

PENETRATION		
F	Machining feedrate	600
S	RPM	750
	Clockwise turning direction	
T	Number of the tool to be used	4
D	Tool offset	4



Once the cycle of the third hole is filled in, we define an angular repetition by pressing the [ANGULAR REPETITION] key.



3.


COORDINATE TRANSFORMATION CYCLES
EXAMPLE: USING THE MIRROR IMAGE CYCLE




CNC 8055
MC

REF. 0601

In this cycle, we can enter the necessary data for the angular repetition of different shapes by simply changing the window mode. To toggle this window, we press the [TWO-COLOR] key:

GENERAL CONDITIONS		
X1	X coordinate of the first hole	55
Y1	Y coordinate of the first hole	15
	Angular repetition method 1	
Xc	X coord. of the arc center	35
Yc	Y coord. of the arc center	15
N	Number of positions	3
τ	Angular distance of the last cycle, referred to the horizontal axis X	90

MACHINING		
Zs	Safety Z coordinate	2
Z	Surface Z coordinate	0
P	Total pocket depth	20
t	Dwell at the bottom (seconds)	0
I	Penetration step	5
F	Machining feedrate	600
S	RPM	750
	Clockwise turning direction	
T	Number of the tool to be used	4
D	Tool offset	4

3.

COORDINATE TRANSFORMATION CYCLES
EXAMPLE: USING THE MIRROR IMAGE CYCLE

FAGOR 

CNC 8055
MC

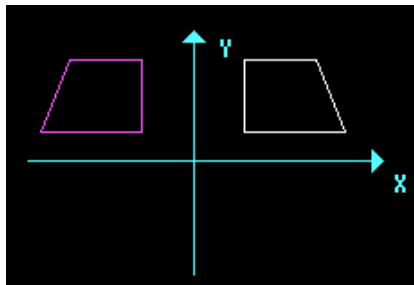
REF. 0601

3.1.6 MIRROR IMAGE

After all the holes have been programmed, we do a mirror image of them to obtain the left side. Doing that requires a mirror image on X



We press the [ISO] key and use the [LEVEL CYCLE] key to find the MIRROR IMAGE cycle (Level 5).



We apply the mirror image function by changing the sign of the coordinates of the selected axis.

CONDITIONS		
NEW	Axis upon which mirror image is applied.	X

We press the [PPROG] key and insert that cycle into the program by pressing [ENTER].

When applying a mirror image function, we must repeat the portion of the program to be copied; in other words, we must set some labels indicating the beginning and end of the repetition and then give the command RPT:



We select the window corresponding to the labels using the [BICOLOR] key and pressing [ENTER].

PROGRAM	
N1	Label located above the first machining operation to be repeated
N2	Label located under the last machining operation to be repeated
RTP	Command located under the MIRROR IMAGE cycle

Then, the program looks like this:

CYCLES
1.- POSITIONING 1 2.- PROFILE MILLING 3.- RECTANGULAR POCKET N1;} 4.- DRILLING 1 + POSIT. IN ARC 1 5.- DRILLING 1 + POSIT. LINEAR 6.- DRILLING 1 + POSIT. IN ARC 1 N2;} G10G11;} MIRROR IMAGE (RPT N1,N2)N1;} 7.- POSITIONING 1

3.

COORDINATE TRANSFORMATION CYCLES
EXAMPLE: USING THE MIRROR IMAGE CYCLE

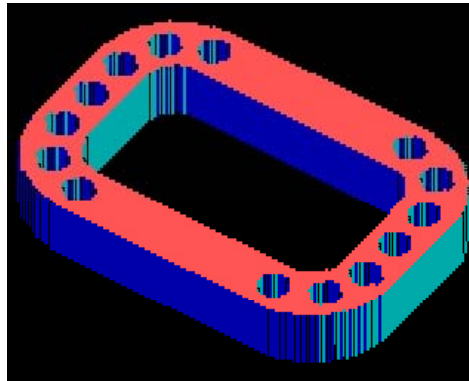


CNC 8055
MC

REF. 0601

3.1.7 PROGRAM SIMULATION

1. We position the red cursor on the first cycle of the program and then press [GRAPHICS].
2. Then, we press [RESET] and then [START].



3.

COORDINATE TRANSFORMATION CYCLES
EXAMPLE: USING THE MIRROR IMAGE CYCLE

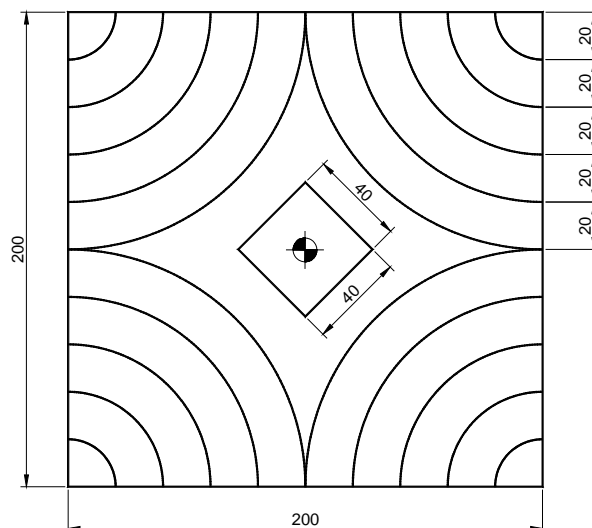
FAGOR 

**CNC 8055
MC**

REF. 0601

3.2 EXAMPLE: USING THE THREE MIRROR IMAGE COMMANDS

To make the following part:



The following example uses the three commands for mirror image and pattern rotation. We will use the following tools:

Operations	Tools
Profile milling	End mill Ø5 T8 D8

After creating the part-program and performing the positioning, we make this part by following these steps:

3.

COORDINATE TRANSFORMATION CYCLES
EXAMPLE: USING THE THREE MIRROR IMAGE COMMANDS

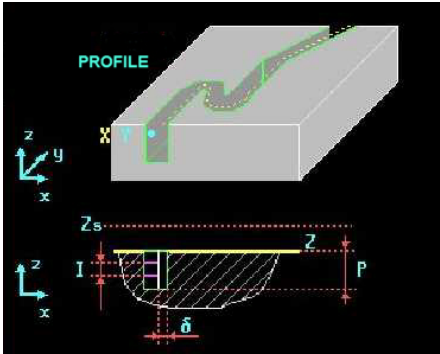


CNC 8055
MC

REF. 0601

3.2.1 PROFILE MILLING

To fill in the PROFILE MILLING cycle, we press [F3]. Then, we press [LEVEL CYCLE] to switch from level 1 to level 2, where we can draw the profile to be contoured using the PROFILE EDITOR.



We select the starting point in XY coordinates, assign a number to the drawing and press [RECALL]:

STARTING POSITION		
X	Starting point on the X axis	130
Y	Starting point on the Y axis	80
	Profile	5

Once on the screen of the PROFILE EDITOR, we draw the desired geometry:

PROFILE PROGRAMMING				
STARTING POINT	X1: 130	Y1: 80		
STRAIGHT	X2: 100	Y2: 80		
CLOCKWISE ARC	X2: 80	Y2: 100	XC: 100	YC: 100
STRAIGHT	X2: 80	Y2: 130		
STRAIGHT	X2: 60	Y2: 130		
STRAIGHT	X2: 60	Y2: 100		
COUNTERCLOCKWISE ARC	X2: 100	Y2: 60	XC: 100	YC: 100
STRAIGHT	X2: 130	Y2: 60		
STRAIGHT	X2: 130	Y2: 40		
STRAIGHT	X2: 100	Y2: 40		
CLOCKWISE ARC	X2: 40	Y2: 100	XC: 100	YC: 100
STRAIGHT	X2: 40	Y2: 130		
STRAIGHT	X2: 20	Y2: 130		
STRAIGHT	X2: 20	Y2: 100		
COUNTERCLOCKWISE ARC	X2: 100	Y2: 20	XC: 100	YC: 100
STRAIGHT	X2: 130	Y2: 20		
STRAIGHT	X2: 130	Y2: 0		
STRAIGHT	X2: 100	Y2: 0		
CLOCKWISE ARC	X2: 0	Y2: 100	XC: 100	YC: 100
STRAIGHT	X2: 0	Y2: 130		
STRAIGHT	X2: 130	Y2: 130		
STRAIGHT	X2: 130	Y2: 80		

3.

COORDINATE TRANSFORMATION CYCLES
EXAMPLE: USING THE THREE MIRROR IMAGE COMMANDS



CNC 8055
MC

Geometry of the example in the PROFILE EDITOR






3.

COORDINATE TRANSFORMATION CYCLES
EXAMPLE: USING THE THREE MIRROR IMAGE COMMANDS

After drawing the contour of the figure, we save the drawing in the PROFILE MILLING cycle and fill in the rest of the data:

GENERAL CONDITIONS		
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	5
I	Value of each pass	2
Fz	In feed	100

ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	8
D	Tool offset	8
	Without tool compensation	

FINISHING		
F	Finishing feedrate	0
S	RPM	0
	Clockwise turning direction	
T	Number of the tool to be used	0
D	Tool offset	0
δ	Finishing stock	0



CNC 8055
MC

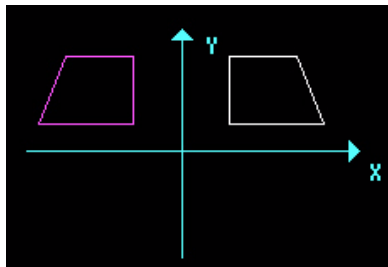
REF. 0601

3.2.2 MIRROR IMAGES

We first do a mirror image of this drawing in X, then in Y and finally in XY to complete all four quadrants of the tile.



We press the [ISO] key and use the [LEVEL CYCLE] key to find the MIRROR IMAGE cycle (Level 5).



Mirror image of the drawing in X.

CONDITIONS		
NEW	Axis upon which mirror image is applied.	X

We press the [PPROG] key and insert that cycle into the program by pressing [ENTER]. We must insert the PROFILE MILLING cycle again under the mirror image cycle so the geometry can be repeated, but now in the corresponding quadrant.

Mirror image of the drawing in Y.

CONDITIONS		
NEW	Axis upon which mirror image is applied.	Y

Press [PPROG] and [ENTER]. Enter the PROFILE MILLING cycle again

Mirror image of the drawing in XY.

CONDITIONS		
NEW	Axis upon which mirror image is applied.	XY

Press [PPROG] and [ENTER]. Enter the PROFILE MILLING cycle again

3.

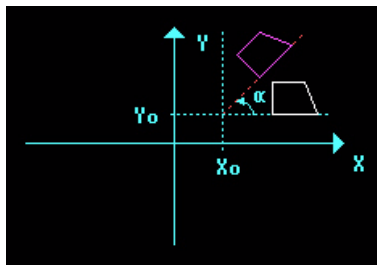
COORDINATE TRANSFORMATION CYCLES
EXAMPLE: USING THE THREE MIRROR IMAGE COMMANDS

3.2.3 PATTERN ROTATION

The next step is to program the center square, but this square is rotated 45 degrees around its geometrical center; therefore, since we will be drawing the square horizontally, we must indicate in the program a previous 45 degree rotation.



We press the [ISO] key and use the [LEVEL CYCLE] key to find the PATTERN ROTATION cycle (Level 7).



CONDITIONS		
New		
τo	Rotation angle	45
Xo	X position of the rotation center	0
Yo	Y position of the rotation center	0

We save the cycle in the program, machine the inside square using the PROFILE MILLING cycle

3.

COORDINATE TRANSFORMATION CYCLES
EXAMPLE: USING THE THREE MIRROR IMAGE COMMANDS

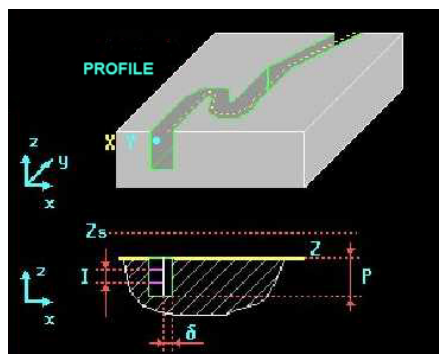


CNC 8055
MC

REF. 0601

3.2.4 PROFILE MILLING

To fill in the PROFILE MILLING cycle, we press [F3]. Then, we press [LEVEL CYCLE] to switch from level 1 to level 2, where we can draw the profile to be contoured using the PROFILE EDITOR.



We select the starting point in XY coordinates, assign a number to the drawing and press [RECALL]:

STARTING POSITION		
X	Starting point on the X axis	20
Y	Starting point on the Y axis	0
	Profile	6

Once inside the PROFILE EDITOR, we draw the desired geometry.

PROFILE PROGRAMMING

STARTING POINT	X1: 20	Y1: 0
STRAIGHT	X2: 20	Y2: 20
STRAIGHT	X2: -20	Y2: 20
STRAIGHT	X2: -20	Y2: -20
STRAIGHT	X2: 20	Y2: -20
STRAIGHT	X2: 20	Y2: 0

Geometry of the example in the PROFILE EDITOR



After drawing the contour of the figure, we save the drawing in the PROFILE MILLING cycle and fill in the rest of the data:

GENERAL CONDITIONS		
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	5
I	Value of each pass	2
Fz	In feed	100

3.

COORDINATE TRANSFORMATION CYCLES
EXAMPLE: USING THE THREE MIRROR IMAGE COMMANDS



FAGOR


CNC 8055
MC

REF. 0601

3.

COORDINATE TRANSFORMATION CYCLES EXAMPLE: USING THE THREE MIRROR IMAGE COMMANDS

ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	8
D	Tool offset	8
	Without tool compensation	

FINISHING		
F	Finishing feedrate	0
S	RPM	0
	Clockwise turning direction	
T	Number of the tool to be used	0
D	Tool offset	0
δ	Finishing stock	0



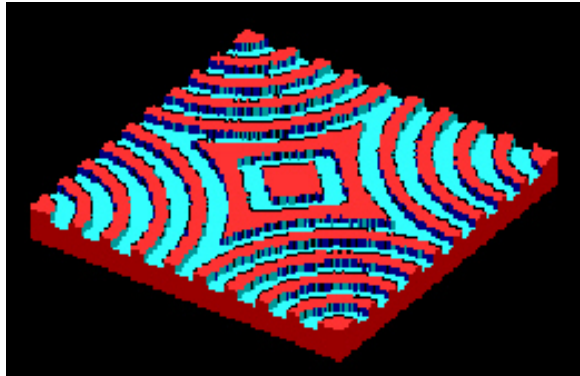
After machining the square, we cancel the pattern rotation. We do this in the turning cycle itself by selecting the CANCEL option.

The program will look like this:

CYCLES
1.- POSITIONING 1
2.- PROFILE MILLING G10G11;} MIRROR IMAGE
3.- PROFILE MILLING G10G12;} MIRROR IMAGE
4.- PROFILE MILLING G10G11G12;} MIRROR IMAGE
5.- PROFILE MILLING TURN, another 45° (0,0)
6.- PROFILE MILLING G73;} PATTERN ROTATION
7.- POSITIONING 1

3.2.5 PROGRAM SIMULATION

1. We position the red cursor on the first cycle of the program, in this case POSITIONING 1, and then press [GRAPHICS].
2. Then, we press [RESET] and then [START].



3.

COORDINATE TRANSFORMATION CYCLES
EXAMPLE: USING THE THREE MIRROR IMAGE COMMANDS

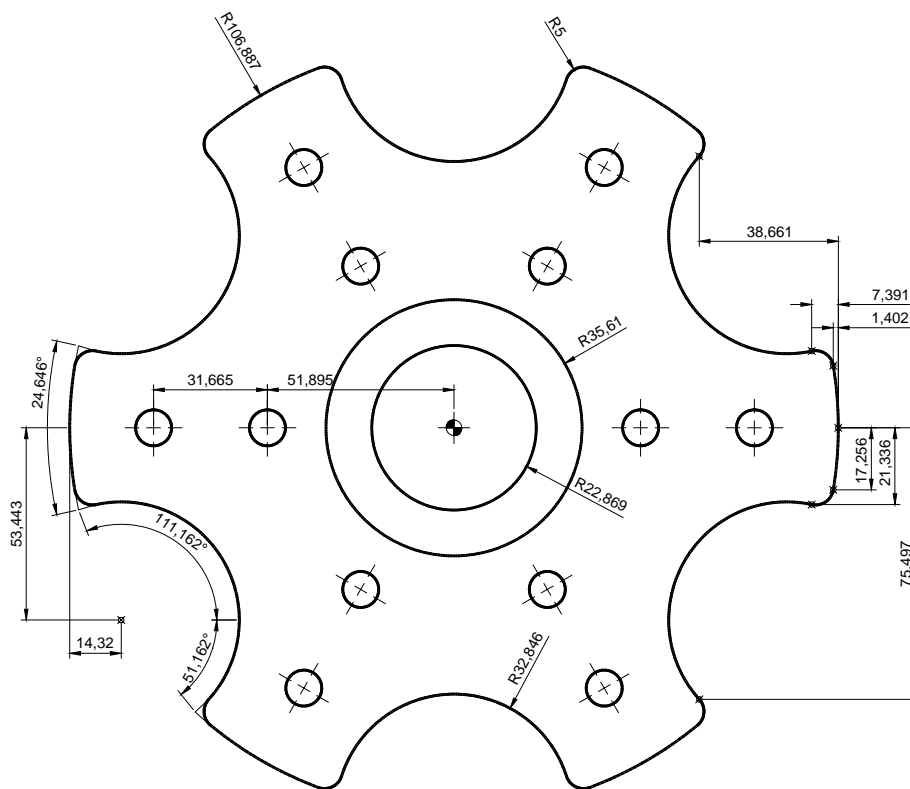
FAGOR 

**CNC 8055
MC**

REF. 0601

3.3 EXAMPLE: PATTERN ROTATION

To make the following part:



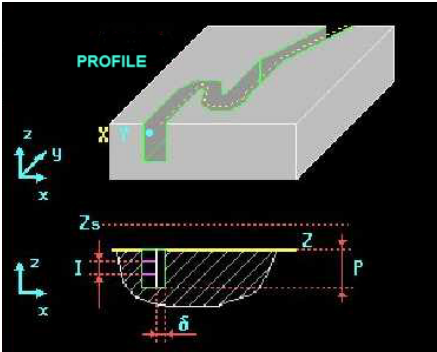
We will use the following tools:

Operations	Tools
Contour machining (roughing)	Flat endmill $\varnothing 6$ T1 D1
Contour machining (finishing)	Flat endmill $\varnothing 4$ T2 D2
Drilling	Drill bit $\varnothing 10$ T4 D4

After creating the part-program and performing the positioning, we make this part by following these steps:

3.3.1 PROFILE MILLING

To fill in the PROFILE MILLING cycle, we press [F3]. Then, we press [LEVEL CYCLE] to switch from level 1 to level 2, where we can draw the profile to be contoured using the PROFILE EDITOR.



We select the starting point in XY coordinates, assign a number to the drawing and press [RECALL]:

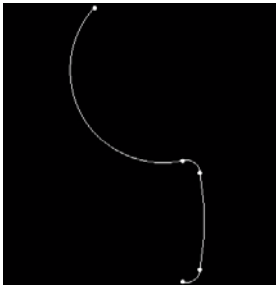
STARTING POSITION		
X	Starting point on the X axis	99.496
Y	Starting point on the Y axis	-21.336
	Profile	7

Once inside the PROFILE EDITOR, we draw the desired geometry.

PROFILE PROGRAMMING

STARTING POINT X1: 99.496 Y1: -21.336
COUNTERCLOCKWISE X2: 105.485 Y2: -17.256 R: 5
ARC
COUNTERCLOCKWISE X2: 105.485 Y2: 17.256 XC: 0 YC: 0
ARC
COUNTERCLOCKWISE X2: 99.496 Y2: 21.336 R: 5
ARC
CLOCKWISE ARC X2: 68.226 Y2: 75.497 R: 32.846

Geometry of the example in the PROFILE EDITOR



3.

COORDINATE TRANSFORMATION CYCLES
EXAMPLE: PATTERN ROTATION



CNC 8055
MC



REF. 0601


After drawing the contour of the figure, we save the drawing in the PROFILE MILLING cycle and fill in the rest of the data:

3.

COORDINATE TRANSFORMATION CYCLES
EXAMPLE: PATTERN ROTATION

GENERAL CONDITIONS		
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	30
I	Value of each pass	2
Fz	In feed	100

ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	1
D	Tool offset	1
	Right hand tool radius compensation	

FINISHING		
F	Finishing feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	2
D	Tool offset	2
δ	Finishing stock	0.2



CNC 8055
MC

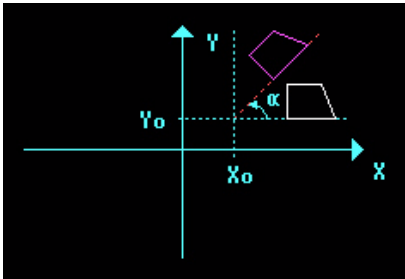
REF. 0601

3.3.2 PATTERN ROTATION

Then, we cancel the pattern rotation:



We press the [ISO] key and use the [LEVEL CYCLE] key to find the PATTERN ROTATION cycle (Level 7).



CONDITIONS		
Additive		
τO	Rotation angle	60
Xo	X position of the rotation center	0
Yo	Y position of the rotation center	0

We must select the additive option to make the angle incremental and to complete the geometry.

We have the cycle in the program and proceed to define labels. These labels serve to specify which portion (from where to where) will be repeated.



We press the [ISO] key and use the [LEVEL CYCLE] key to find the cycle corresponding to the labels (Level 4).

After entering the label N2, we give the repetition command by accessing the ISO cycle again Then, cancel the rotation and insert it into the program.

3.

COORDINATE TRANSFORMATION CYCLES
EXAMPLE: PATTERN ROTATION

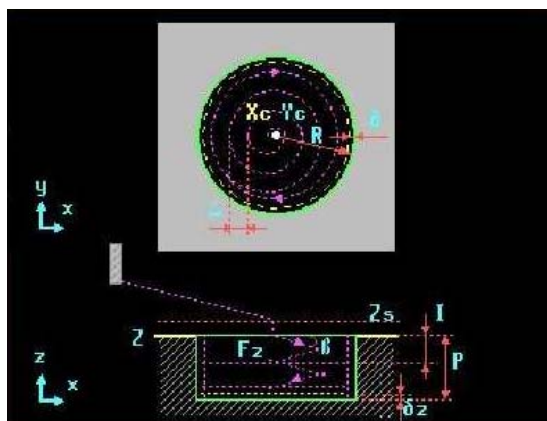


CNC 8055
MC



REF. 0601



3.3.3 CIRCULAR POCKETS

After carrying out the positioning, we program the cycles for the two pockets in the center. These pockets are identical, just their radius and depth are different.



GENERAL CONDITIONS			
		POCKET 1	POCKET 2
Xc	Pocket center in X	0	0
Yc	Pocket center in Y	0	0
R	Pocket radius	35.61	22.869
Zs	Safety Z coordinate	2	2
Z	Surface coordinate	0	0
P	Total depth in absolute coordinates	15	30
I	Value of each pass	3	3
Fz	In feed	100	100

ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	1
D	Tool offset	1
β	Penetrating angle	90
	Clockwise machining	
Δ	Roughing pass	0

FINISHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	2
D	Tool offset	2
θ	Tool penetrating angle when finishing	90
	Clockwise machining	
δ	Finishing stock	0.2
δz	Excess material in Z	0.1
N	Number of finishing passes in Z	1

3.

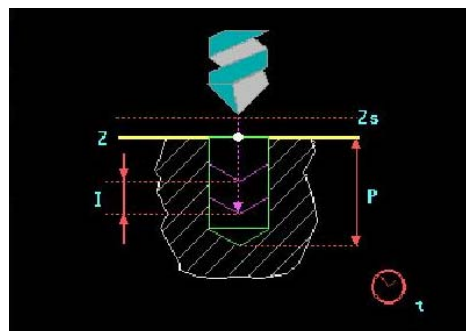
COORDINATE TRANSFORMATION CYCLES
EXAMPLE: PATTERN ROTATION




CNC 8055
MC

REF. 0601

3.3.4 DRILLING WITH LINEAR REPETITION

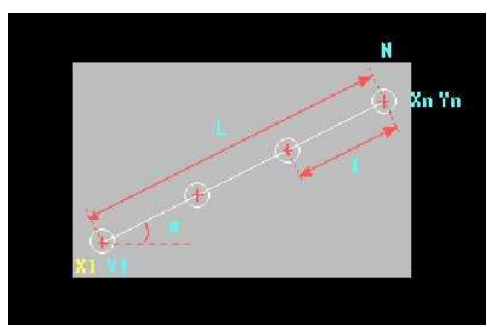



GENERAL CONDITIONS		
X	X coordinate of the first hole	51.895
Y	Y coordinate of the first hole	0
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	30
I	Value of each pass	5
t	Dwell at the bottom	0

PENETRATION		
F	Machining feedrate	600
S	RPM	750
	Clockwise turning direction	
T	Number of the tool to be used	4
D	Tool offset	4



Once we filled in the cycle of the second hole and before saving it in the program, we define a linear repetition by pressing the [LINEAR REPETITION] key.



GENERAL CONDITIONS		
X1	X coordinate of the first hole	51.895
Y1	Y coordinate of the first hole	0
	Linear repetition method 1	
τO	Line rotation angle	0
L	Total line length	31.665
N	Number of positions	2

3.

COORDINATE TRANSFORMATION CYCLES
EXAMPLE: PATTERN ROTATION


FAGOR 

CNC 8055
MC

REF. 0601

3.

COORDINATE TRANSFORMATION CYCLES
EXAMPLE: PATTERN ROTATION

DRILLING		
Zs	Safety Z coordinate	2
Z	Surface Z coordinate	0
P	Total pocket depth	30
t	Dwell at the bottom (seconds)	0
I	Penetration step	5
F	Machining feedrate	600
S	RPM	750
	Clockwise turning direction	
T	Number of the tool to be used	4
D	Tool offset	4



CNC 8055
MC

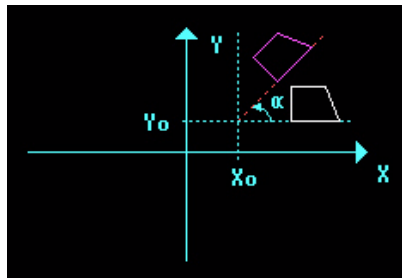
REF. 0601

3.3.5 PATTERN ROTATION

After doing the drilling using linear repetition, we apply a pattern rotation to drill the rest of the holes:



We press the [ISO] key and use the [LEVEL CYCLE] key to find the PATTERN ROTATION cycle (Level 7).



CONDITIONS		
Additive		
τ_0	Rotation angle	60
X_0	X position of the rotation center	0
Y_0	Y position of the rotation center	0

We save the cycle into the program and define the labels, these labels serve to specify which area (from where to where) will be repeated.



We press the [ISO] key and use the [LEVEL CYCLE] key to find the cycle corresponding to the labels (Level 4).

When entering the label N4 in the program, we give the repetition command by accessing the ISO cycle again

The definition of the last two labels must be different from the first two, otherwise the repetition will be wrong. That's why N3 and N4 are defined. There is no need to cancel it because it is the last rotation in the program.

Finally, the program will look like this:

CYCLES
1.- POSITIONING 1 N1;}
2.- PROFILE MILLING G73Q60I0J0;} PATTERN ROTATION N2;} (RPT N1,N2)N5;}
G73;} PATTERN ROTATION
3.- POSITIONING 1
4.- CIRCULAR POCKET 1
5.- CIRCULAR POCKET 1 N3;}
6.- DRILLING 1 + POSIT. LINEAR G73Q60I0J0;} PATTERN ROTATION N4;} (RPT N3,N4)N5;}

3.

COORDINATE TRANSFORMATION CYCLES
EXAMPLE: PATTERN ROTATION

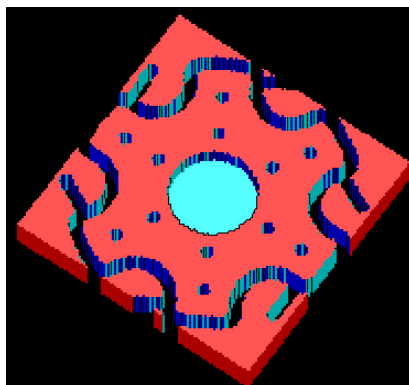


CNC 8055
MC

REF. 0601

3.3.6 PROGRAM SIMULATION

1. We position the red cursor on the first cycle of the program and then press [GRAPHICS].
2. Then, we press [RESET] and then [START].



3.

COORDINATE TRANSFORMATION CYCLES
EXAMPLE: PATTERN ROTATION



CNC 8055
MC

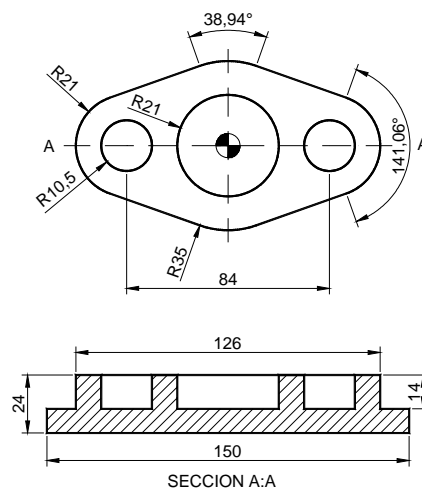
REF. 0601

2D CYCLES

4

4.1 EXAMPLE: JOINT

The following example uses the 2D POCKET cycle. This cycle serves to machine irregular pockets:



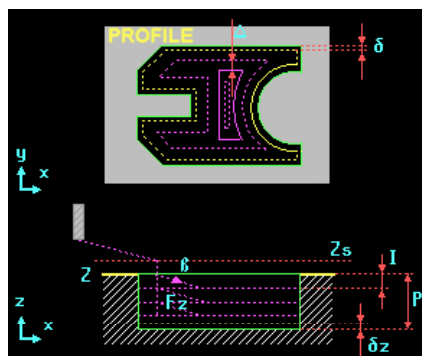
We will use the following tools:

Operations	Tools
Contour machining (roughing)	Flat endmill Ø6 T1 D1
Contour machining (finishing)	Flat endmill Ø4 T2 D2
Pocket machining (roughing)	Flat endmill Ø6 T1 D1
Pocket machining (finishing)	Flat endmill Ø4 T2 D2

After creating the part-program and performing the positioning, we make this part by following these steps:

4.1.1 2D PROFILE POCKET

In this part, we will carry out a roughing operation from the outside contour up to the profile of the joint. Then, we will run a finishing pass. To go into the 2D PROFILE POCKET cycle, press [F5].



We select the starting point in XY coordinates, assign a number to the drawing and press [RECALL]:

STARTING POSITION		
X	Starting point on the X axis	-85
Y	Starting point on the Y axis	-55
	Profile	8

The drawing has two profiles, an outside contour indicating the roughing boundaries and a profile of the figure to be machined in relief. If the pocket must be emptied, we will just have to draw the geometry to be emptied. We first draw the outside contour and then define the second profile by pressing the NEW PROFILE key:

PROFILE PROGRAMMING

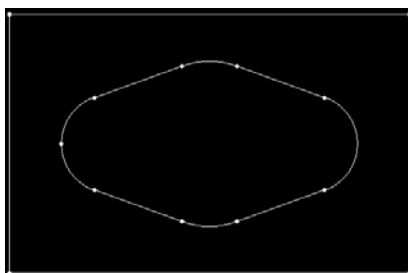
STARTING POINT	X1: -85	Y1: -55				
STRAIGHT	X2: 85	Y2: -55				
STRAIGHT	X2: 85	Y2: 55				
STRAIGHT	X2: -85	Y2: 55				
STRAIGHT	X2: -85	Y2: -55				
STARTING POINT	X1: -63	Y1: 0				
COUNTERCLOCKWISE	XC: -42	YC: 0	R: 21			
ARC						
STRAIGHT	TANG: YES					
COUNTERCLOCKWISE	XC: 0	YC: 0	R: 35	TANG: YES		
ARC						
STRAIGHT	TANG: YES					
COUNTERCLOCKWISE	XC: 42	YC: 0	R: 21	TANG: YES		
ARC						
STRAIGHT	TANG: YES					
COUNTERCLOCKWISE	XC: 0	YC: 0	R: 35	TANG: YES		
ARC						
STRAIGHT	TANG: YES					
COUNTERCLOCKWISE	X2: -63	Y2: 0	XC: -42	YC: 0	R: 21	TANG: YES
ARC						



CNC 8055
MC


REF. 0601


Geometry of the example in the PROFILE EDITOR



After having drawn the geometries, we go back to the 2D PROFILE POCKET cycle to fill in the machining conditions for roughing and finishing:

GENERAL CONDITIONS		
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	14
I	Value of each pass	3
Fz	Penetration feedrate	100

ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	1
D	Tool offset	1
β	Sideways penetration angle	90
Δ	Maximum roughing pass	0

FINISHING		
F	Finishing feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	2
D	Tool offset	2
θ	Sideways penetration angle	90
δ	Finishing stock	0.2
δz	Penetration stock	0.1
N	Number of penetration passes	1

4.

2D CYCLES
EXAMPLE: JOINT

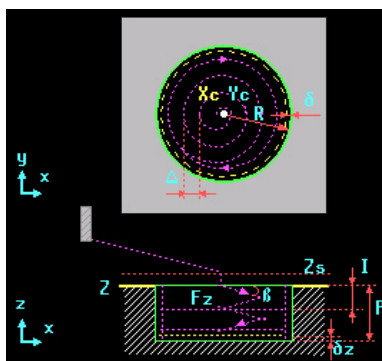
FAGOR 

CNC 8055
MC

REF. 0601

4.1.2 CIRCULAR POCKETS

We then add the three circular pocket cycles to the program. We first program the circular pocket of the center, then the left one and finally the right one.



GENERAL CONDITIONS				
		POCKET 1	POCKET 2	POCKET 3
Xc	Pocket center in X	0	-42	42
Yc	Pocket center in Y	0	0	0
R	Pocket radius	21	10.5	10.5
Zs	Safety Z coordinate	2	2	2
Z	Surface coordinate	0	0	0
P	Total depth in absolute coordinates	14	14	14
I	Value of each pass	3	3	3
Fz	In feed	100	100	100

The machining conditions are the same for the three pockets:

ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	1
D	Tool offset	1
β	Penetrating angle	90
	Clockwise machining	
Δ	Roughing pass	0

FINISHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	2
D	Tool offset	2
θ	Tool penetrating angle when finishing	90
	Clockwise machining	
δ	Finishing stock	0.2
δz	Excess material in Z	0.1
N	Number of finishing passes in Z	1

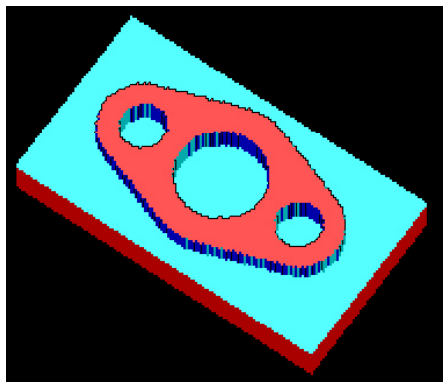
And last, we program a positioning to move the tool to the safety distance in Z.

The structure of the program is:

CYCLES
1.- POSITIONING 1
2 .- 2D PROFILE POCKET
3 .- CIRCULAR POCKET 1
4 .- CIRCULAR POCKET 1
5 .- CIRCULAR POCKET 1
6.- POSITIONING 1

4.1.3 PROGRAM SIMULATION

1. We position the red cursor on the first cycle of the program and then press [GRAPHICS].
2. Then, we press [RESET] and then [START].



4.

2D CYCLES
EXAMPLE: JOINT

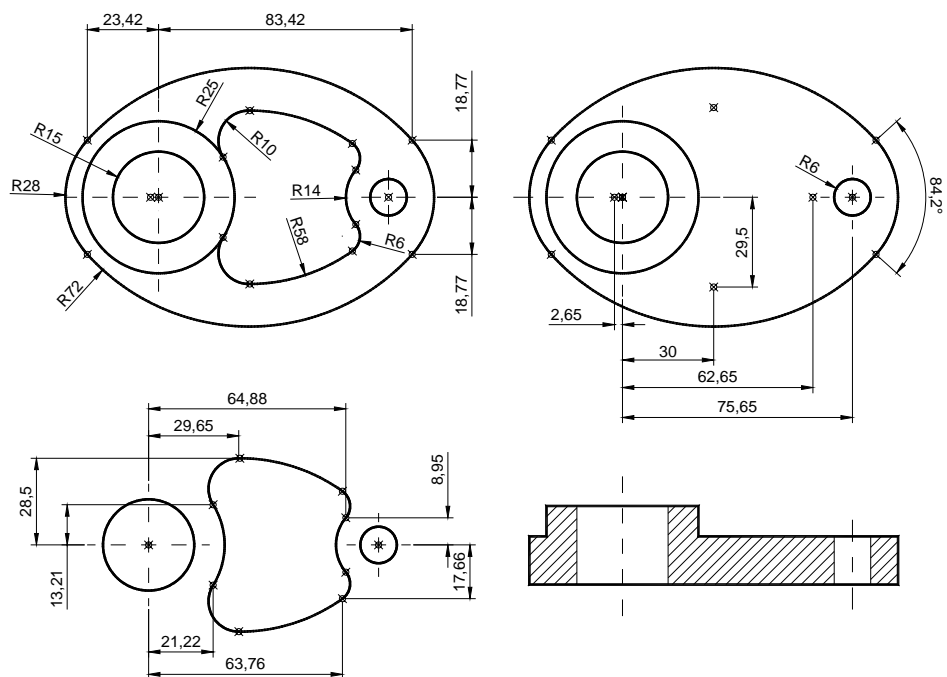
FAGOR 

CNC 8055
MC

REF. 0601

4.2 EXAMPLE: CAM

To make the following part:



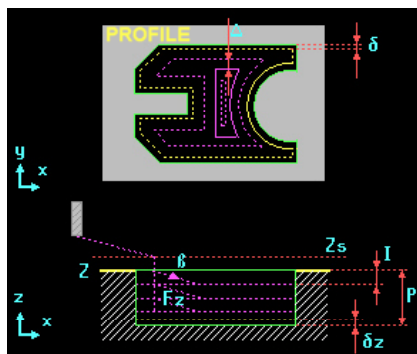
We will use the following tools:

Operations	Tools
2D pockets (roughing)	End mill Ø6 T1 D1
2D pockets (finishing)	End mill Ø4 T2 D2
Drilling	Drill bit Ø12 T9 D9
Circular pocket (roughing)	End mill Ø6 T1 D1
Circular pocket (finishing)	End mill Ø4 T2 D2

After creating the part-program and performing the positioning, we make this part by following these steps:

4.2.1 2D PROFILE POCKETS

To go into the 2D PROFILE POCKET cycle, press [F5].



We select the starting point in XY coordinates, assign a number to the drawing and press [RECALL]:

STARTING POSITION		
X	Starting point on the X axis	-50
Y	Starting point on the Y axis	-70
	Profile	9

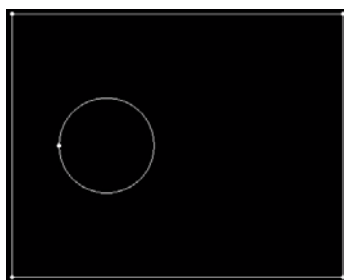
First of all, we machine the raw stock until a circular relief is obtained, to do this, first we have to draw the following geometry inside the first 2D PROFILE POCKET cycle:

PROFILE PROGRAMMING

STARTING POINT X1: -50 Y1: -70
 STRAIGHT X2: 120 Y2: -70
 STRAIGHT X2: 120 Y2: 70
 STRAIGHT X2: -50 Y2: 70
 STRAIGHT X2: -50 Y2: -70

COUNTERCLOCKWISE X1: -25 Y1: 0 XC: 0 YC: 0 R: 25
 CIRCLE

This drawing defines the outside contour that represents the raw stock and the circular relief inside the part:



GENERAL CONDITIONS		
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	10
I	Value of each pass	3
Fz	Penetration feedrate	100

4.

2D CYCLES
EXAMPLE: CAM


FAGOR


CNC 8055
MC

REF. 0601

4.

2D CYCLES
EXAMPLE: CAM

ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	1
D	Tool offset	1
β	Sideways penetration angle	90
Δ	Maximum roughing pass	0

FINISHING		
F	Finishing feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	2
D	Tool offset	2
θ	Sideways penetration angle	90
δ	Finishing stock	0.2
δz	Penetration stock	0.1
N	Number of penetration passes	1

We insert the cycle in the program with the [PPROG] key and call the 2D PROFILE POCKET cycle again to do the next pocket.

STARTING POSITION		
X	Starting point on the X axis	-50
Y	Starting point on the Y axis	-70
	Profile	10

For the next 2D machining operation, define the same outside contour as in the previous cycle and in this case, the island will be the main ellipse of the part.

PROFILE PROGRAMMING

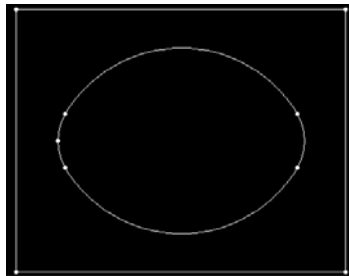
STARTING POINT	X1: -50	Y1: -70		
STRAIGHT	X2: 120	Y2: -70		
STRAIGHT	X2: 120	Y2: 70		
STRAIGHT	X2: -50	Y2: 70		
STRAIGHT	X2: -50	Y2: -70		
STARTING POINT	X1: -28	Y1: 0		
COUNTERCLOCKWISE	XC: 0	YC: 0	R: 28	
ARC				
COUNTERCLOCKWISE	X2: 83.42	Y2: -18.773	R: 72	TANG: YES
ARC				
COUNTERCLOCKWISE	X2: 83.42	Y2: 18.773	YC: 0	R: 28
ARC				
COUNTERCLOCKWISE	X2: -23.42	R: 72	TANG: YES	
ARC				
COUNTERCLOCKWISE	X2: -28	Y2: 0	R: 28	
ARC				

Geometry of the example in the PROFILE EDITOR



CNC 8055
MC


REF. 0601




4.

2D CYCLES
EXAMPLE: CAM

GENERAL CONDITIONS		
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	30
I	Value of each pass	3
Fz	Penetration feedrate	100

ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	1
D	Tool offset	1
β	Sideways penetration angle	90
Δ	Maximum roughing pass	0

FINISHING		
F	Finishing feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	2
D	Tool offset	2
θ	Sideways penetration angle	90
δ	Finishing stock	0.2
δz	Penetration stock	0.1
N	Number of penetration passes	1

We insert the cycle in the program with the [PPROG] key and call the 2D PROFILE POCKET cycle again to do the next pocket.

STARTING POSITION		
X	Starting point on the X axis	25
Y	Starting point on the Y axis	0
	Profile	11

Once on the screen of the PROFILE EDITOR, we draw the desired geometry:

PROFILE PROGRAMMING

STARTING POINT X1: 25 Y1:0
 COUNTERCLOCKWISE ARC X2: 21.22 Y2: 13.21 XC: 0 YC: 0 R: 25
 CLOCKWISE ARC X2: 29.65 Y2: 28.5 R: 10 TANG: YES
 CLOCKWISE ARC X2: 63.76 Y2: 17.66 R: 58
 CLOCKWISE ARC X2: 64.88 Y2: 8.95 R: 6

FAGOR 

CNC 8055
MC

REF. 0601

COUNTERCLOCKWISE ARC	X2: 64.88	Y2: -8.95	XC: 75.65	YC: 0	R: 14
CLOCKWISE ARC	X2: 63.76	Y2: -17.66	R: 6		
CLOCKWISE ARC	X2: 29.65	Y2: -28.5	R: 58		
CLOCKWISE ARC	X2: 21.22	Y2: -13.21	R: 10		
COUNTERCLOCKWISE ARC	X2: 25	Y2: 0	XC: 0	YC: 0	R: 25

4.

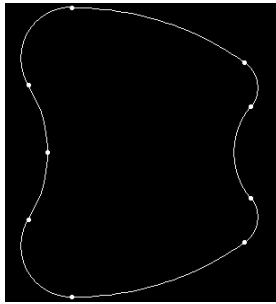
2D CYCLES
EXAMPLE: CAM



CNC 8055
MC

REF. 0601


Geometry of the example in the PROFILE EDITOR




4.

2D CYCLES
EXAMPLE: CAM

GENERAL CONDITIONS		
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	30
I	Value of each pass	3
Fz	Penetration feedrate	100

ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	1
D	Tool offset	1
β	Sideways penetration angle	90
Δ	Maximum roughing pass	0

FINISHING		
F	Finishing feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	2
D	Tool offset	2
θ	Sideways penetration angle	90
δ	Finishing stock	0.2
δz	Penetration stock	0.1
N	Number of penetration passes	1

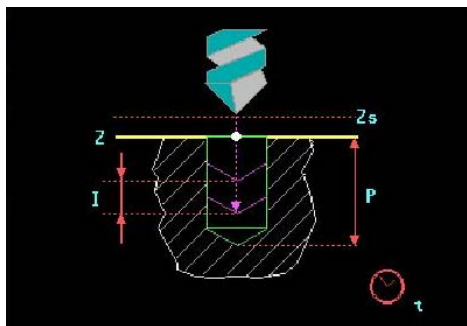
FAGOR 

CNC 8055
MC

REF. 0601


4.2.2 DRILLING

To go into the DRILLING 1 cycle, we press:



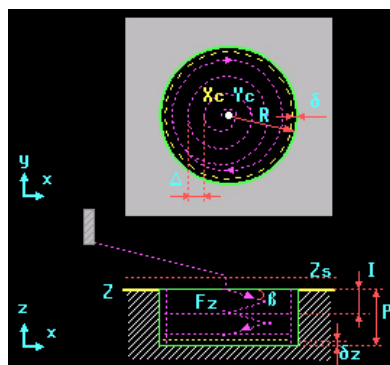
Drilling definition:

GENERAL CONDITIONS		
X	X coordinate of the hole	75.646
Y	Y coordinate of the hole	0
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	30
I	Value of each pass	5
t	Dwell at the bottom	0

PENETRATION		
F	Machining feedrate	600
S	RPM	750
	Clockwise turning direction	
T	Number of the tool to be used	9
D	Tool offset	9



4.2.3 CIRCULAR POCKET 1



To go into the CIRCULAR POCKET 1 cycle, we press [F7]. Then, we press [LEVEL CYCLE] until reaching level 3.



The CIRCULAR POCKET 1 cycle always begins at its center.

GENERAL CONDITIONS		
Xc	Pocket center in X	0
Yc	Pocket center in Y	0
R	Pocket radius	15
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	30
I	Value of each pass	3
Fz	In feed	100

ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	1
D	Tool offset	1
β	Penetrating angle	90
	Clockwise machining	
Δ	Roughing pass	0

FINISHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	2
D	Tool offset	2
θ	Tool penetrating angle when finishing	90
	Clockwise machining	
δ	Finishing stock	0.2
δz	Excess material in Z	0.1
N	Number of finishing passes in Z	1

4.

2D CYCLES
EXAMPLE: CAM

FAGOR 

CNC 8055
MC

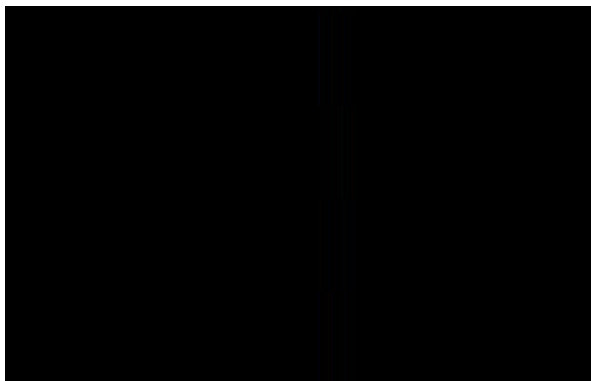
REF. 0601

The structure of the program is:

CYCLES
1.- POSITIONING 1
2 .- 2D PROFILE POCKET
3 .- 2D PROFILE POCKET
4 .- 2D PROFILE POCKET
5 .- DRILLING 1
6 .- CIRCULAR POCKET 1
7.- POSITIONING 1

4.2.4 PROGRAM SIMULATION

1. We position the red cursor on the first cycle of the program and then press [GRAPHICS].
2. Then, we press [RESET] and then [START].



4.

2D CYCLES
EXAMPLE: CAM



CNC 8055
MC

REF. 0601

3D CYCLES

5

5.1 INTRODUCTION

The 3D cycles are programmed the same way as the 2D cycles. The difference is that each profile defined in the main XY plane has its own depth profile XZ or YZ. The cycle used for 3D pockets has several boxes to define the part number and the different depth profiles:

Parameters	Description
POCK. 3D	Name of the Pocket
P. XY	XY plane profile file
P. Z1	Z-profile file for the 1st XY profile
P. Z2	Z-profile file for the 2nd XY profile
P. Z3	Z-profile file for the 3rd XY profile
P. Z4	Z-profile file for the 4th XY profile

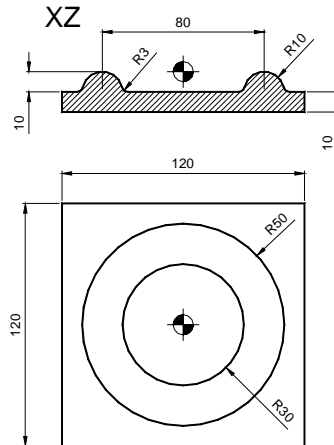
The first two boxes indicate the part number and the number for the XY drawing of the 3D pocket respectively. Boxes PZ1, PZ2, PZ3 and PZ4 are the depth profiles in the programming order of the XY profiles.

The 3D pocket involves the execution of the three machining operations. First, we run a previous roughing operation of the geometry in the plane; we then do a semi-finishing operation that is like a fine roughing and, finally, we carry out the finishing operation. The three operations are optional and may be omitted by programming all the values of the desired operation at zero.

Parameters	Description
I1	Maximum penetration pass when roughing
I2	Maximum Z pass when semi-finishing

5.2 EXAMPLE: TOROID

To make the next part, two 3D cycles will be used, the first one to achieve the outside relief and the second one to empty the inside.



We will use the following tools:

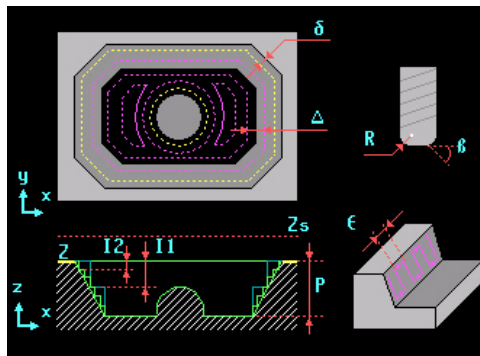
Operations	Tools
Machining of the 3D pocket (roughing)	End mill Ø6 T1 D1
Machining of the 3D pocket (finishing)	End mill Ø4 T2 D7

To make the part, we follow these steps:

5.2.1 3D PROFILE POCKETS

To fill in the 3D POCKET cycle, we press [F5]. Then, we press [LEVEL CYCLE] to go from level 1 to level 2.

In the 3D cycle, we define first the geometry of the XY plane as if it were a 2D machining operation and then, inside the same cycle, we define the depth profiles for each profile defined in XY. These profiles are defined by making the starting points of each geometry coincide.



First, we assign a number to the pocket and then a number to each profile. We do this by following a logical numeric order in order not to mix drawings of different programs. We assign to XY profile the tens of the number chosen for the pocket and, from then on, we assign consecutive numbers to each profile:

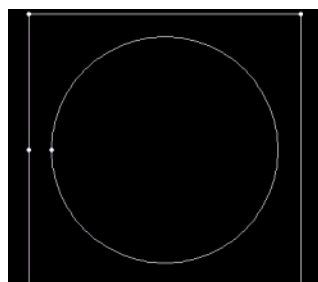
POCKET AND PROFILE NUMBERING		
POCK. 3D	Name of the Pocket	1
P. XY	XY plane profile file	11
P. Z1	Z-profile file for the 1st XY profile	12
P. Z2	Z-profile file for the 2nd XY profile	13
P. Z3	Z-profile file for the 3rd XY profile	0
P. Z4	Z-profile file for the 4th XY profile	0

In the PXY box, we draw the XY profile of the pocket with an outside profile since we are going to machine a relief. First we draw the outside contour and then define the second profile of the island by pressing the NEW PROFILE key:

PROFILE PROGRAMMING

STARTING POINT	X1: -60	Y1: 0			
STRAIGHT	X2: 0	Y2: 60			
STRAIGHT	X2: 60	Y2: 60			
STRAIGHT	X2: 60	Y2: -60			
STRAIGHT	X2: -60	Y2: -60			
STRAIGHT	X2: -60	Y2: 0			
STARTING POINT	X1: -50	Y1: 0			
CLOCKWISE ARC	X2: -50	Y2: 0	XC: 0	YC: 0	R: 50

Geometry of the example in the PROFILE EDITOR



5.

3D CYCLES
EXAMPLE: TOROID

FAGOR

CNC 8055
MC

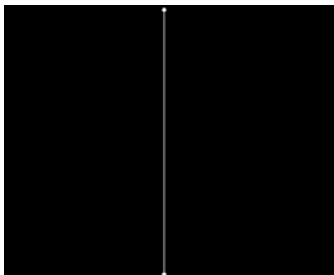
REF. 0601

In the box for PZ1, we draw the depth profile for the outside contour. This depth profile is drawn from up down because it is for emptying:

PROFILE PROGRAMMING

STARTING POINT X1: -60 Z1: -10
STRAIGHT X2: -60 Z2: -20

Geometry of the example in the PROFILE EDITOR

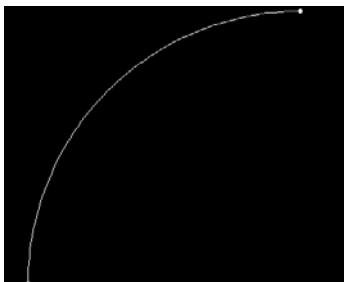


In PZ2, we draw the depth profile for the island. This time, we define it from down up:

PROFILE PROGRAMMING


STARTING POINT X1: -50 Z1: -10
CLOCKWISE ARC X2: -40 Z2: 0 XC: -40 ZC: -10 R: 10



Geometry of the example in the PROFILE EDITOR



After programming the profiles, we fill in the section that indicates the machining conditions:

GENERAL CONDITIONS		
X	Starting point on the X axis	-60
Y	Starting point on the Y axis	0
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	10
I1	Maximum penetration pass when roughing	3
I2	Maximum Z pass when semi-finishing	1
Fz	Penetration feedrate	100

ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	1
D	Tool offset	1
β	Sideways penetration angle	90
Δ	Maximum roughing pass	0

FINISHING		
F	Finishing feedrate	500
S	RPM	2000
	Clockwise turning direction	
T	Number of the tool to be used	7
D	Tool offset	7
R	Tool radius	3
δ	Finishing stock	0.1
ε	Finishing pass	1
	Zigzag finishing	

To machine the inside of the toroid, we have to define another 3D pocket; therefore, we access the 3D PROFILE POCKET cycle again and we make the XY profile to empty it.

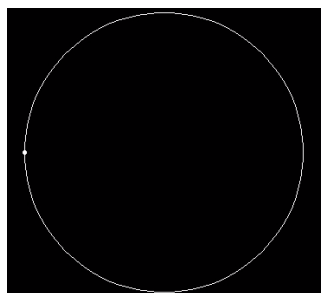
POCKET AND PROFILE NUMBERING		
POCK. 3D	Name of the Pocket	2
P. XY	XY plane profile file	21
P. Z1	Z-profile file for the 1st XY profile	22
P. Z2	Z-profile file for the 2nd XY profile	23
P. Z3	Z-profile file for the 3rd XY profile	0
P. Z4	Z-profile file for the 4th XY profile	0

Since it is an emptying operation, we only have to define the inside geometry and later apply its corresponding depth profile to it. This circle corresponds to the intermediate radius between the outside and the inside circle.

PROFILE PROGRAMMING

COUNTERCLOCKWISE X1: -40 Y1: 0 XC: 0 YC: 0 R: 40
CIRCLE

Geometry of the example in the PROFILE EDITOR



Being an emptying operation, this depth profile is defined from up down and the starting point of the X coordinate must coincide with the X of the starting point of the previous XY profile; this way, this depth profile will be used for the whole XY process.

PROFILE PROGRAMMING

STARTING POINT X1: -40 Z1: 0
CLOCKWISE ARC X2: -30 Z2: -10 XC: -40 ZC: -10 R: 10

5.

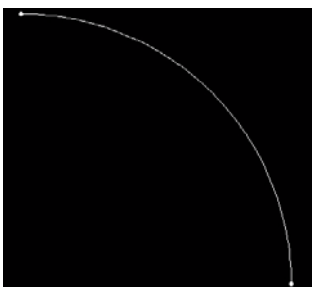
3D CYCLES
EXAMPLE: TOROID

FAGOR 

CNC 8055
MC

REF. 0601

Geometry of the example in the PROFILE EDITOR






5.

3D CYCLES
EXAMPLE: TOROID

After programming the profiles, we fill in the section that indicates the machining conditions:

GENERAL CONDITIONS		
X	Starting point on the X axis	-60
Y	Starting point on the Y axis	0
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	10
I1	Maximum penetration pass when roughing	3
I2	Maximum Z pass when semi-finishing	1
Fz	Penetration feedrate	100

ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	1
D	Tool offset	1
β	Sideways penetration angle	90
Δ	Maximum roughing pass	0

FINISHING		
F	Finishing feedrate	500
S	RPM	2000
	Clockwise turning direction	
T	Number of the tool to be used	7
D	Tool offset	7
R	Tool radius	3
δ	Finishing stock	0.1
ε	Finishing pass	1
	Zigzag finishing	

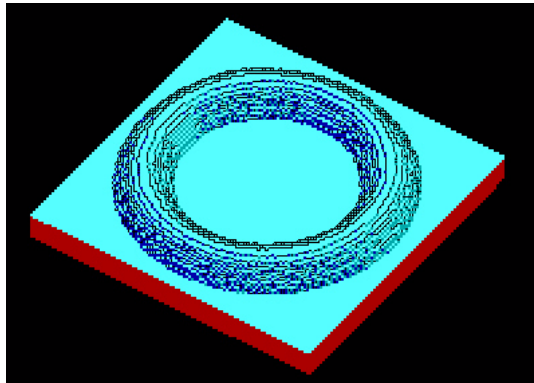


CNC 8055
MC

REF. 0601

5.2.2 PROGRAM SIMULATION

1. We position the red cursor on the first cycle of the program and then press [GRAPHICS].
2. Then, we press [RESET] and then [START].



5.

3D CYCLES
EXAMPLE: TOROID

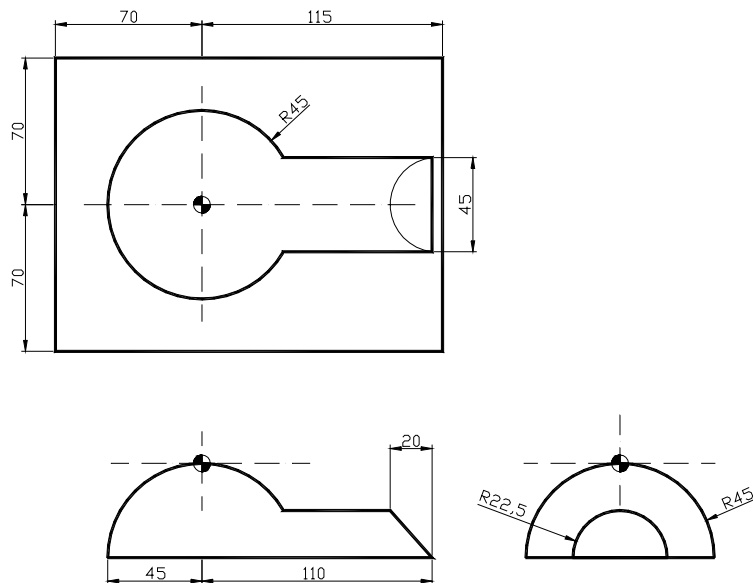
FAGOR 

CNC 8055
MC

REF. 0601

5.3 EXAMPLE: IGLOO

Programming an IGLOO requires intersecting profiles because the geometry has three different depth profiles, the one for the dome, the one for the corridor and the one for the entrance. This intersection of profiles is drawn in the box for XY profiles and, to do that, we use the BOOLEAN PROFILE INTERSECTION system. The CNC will adapt each XY profile to its corresponding depth profile.



We will use the following tools:

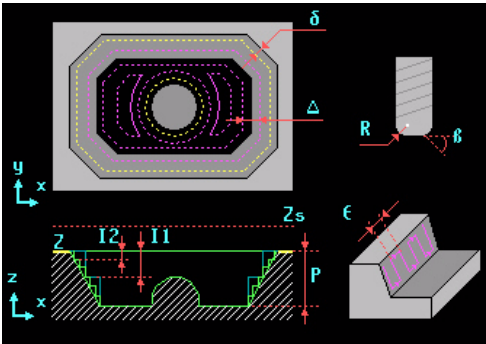
Operations	Tools
Machining of the 3D pocket (roughing)	End mill Ø6 T1 D1
Machining of the 3D pocket (finishing)	End mill Ø4 T2 D7

To make the part, we follow these steps:

5.3.1 3D PROFILE POCKETS

To fill in the 3D POCKET cycle, we press [F5]. Then, we press [LEVEL CYCLE] to go from level 1 to level 2.

In this example, we will fill in the box for the XY PROFILE and the four depth profiles. The depth profile of the corridor of the igloo must be defined in the work plane YZ because its starting point in XY is on top of it. The machining conditions are programmed generically because they depend on the material used in each case.



5.

3D CYCLES

EXAMPLE: IGLOO

First, we assign a number to the whole cycle and then a number to each profile. We do this by following a logical numeric order in order not to mix drawings of different programs. We then assign the tens of the selected number and, from then on, we assign consecutive numbers to each profile.

POCKET AND PROFILE NUMBERING		
POCK. 3D	Name of the Pocket	3
P. XY	XY plane profile file	30
P. Z1	Z-profile file for the 1st XY profile	31
P. Z2	Z-profile file for the 2nd XY profile	32
P. Z3	Z-profile file for the 3rd XY profile	33
P. Z4	Z-profile file for the 4th XY profile	34

XY PROFILE:

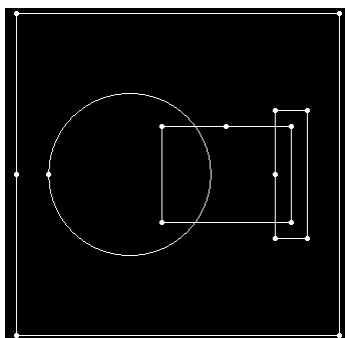
PROFILE PROGRAMMING					
STARTING POINT	X1: -70	Y1: 0			
STRAIGHT	X2: -70	Y2: 70			
STRAIGHT	X2: 115	Y2: 70			
STRAIGHT	X2: 115	Y2: -70			
STRAIGHT	X2: -70	Y2: -70			
STRAIGHT	X2: -70	Y2: 0			
STARTING POINT	X1: -45	Y1: 0			
CLOCKWISE ARC	X2: -45	Y2: 0	XC: 0	YC: 0	R: 45
STARTING POINT	X1: 50	Y1: 22.5			
STRAIGHT	X2: 100	Y2: 22.5			
STRAIGHT	X2: 100	Y2: -22.5			
STRAIGHT	X2: 0	Y2: -22.5			
STRAIGHT	X2: 0	Y2: 22.5			
STRAIGHT	X2: 50	Y2: 22.5			
STARTING POINT	X1: 90	Y1: 0			
STRAIGHT	X2: 90	Y2: 25			
STRAIGHT	X2: 110	Y2: 25			
STRAIGHT	X2: 110	Y2: -25			
STRAIGHT	X2: 90	Y2: -25			
STRAIGHT	X2: 90	Y2: 0			

FAGOR

CNC 8055
MC

REF. 0601

Geometry of the example in the PROFILE EDITOR



5.

3D CYCLES
EXAMPLE: IGLOO

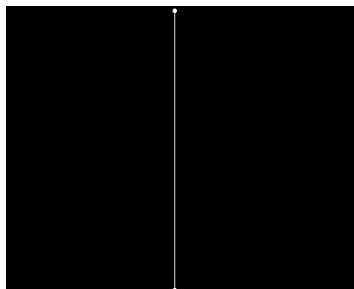
The order these XY geometries are programmed is important because different depth profiles will be programmed respectively and making the starting points in X and in Y coincide.

XZ1 PROFILE:

PROFILE PROGRAMMING

STARTING POINT	X1: -70	Z1: 0
STRAIGHT	X2: -70	Z2: -45

This profile is completely vertical. It defines the outside pocket and it is programmed from up down:

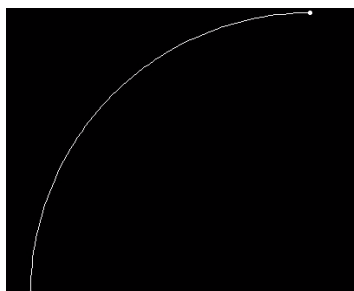


XZ2 PROFILE:

PROFILE PROGRAMMING

STARTING POINT	X1: -45	Z1: -45			
CLOCKWISE ARC	X2: 0	Z2: 0	XC: 0	ZC: -45	R: 45

It defines the dome of the igloo. Since it is an island, it is defined in XZ from down up:



XZ3 PROFILE:

This is the depth profile for the corridor of the igloo. This depth profile is programmed in YZ.

PROFILE PROGRAMMING

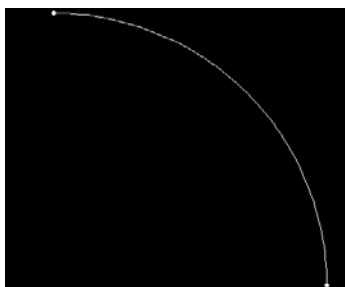
STARTING POINT	Y1: 22.5	Z1: -45			
COUNTERCLOCKWISE ARC	Y2: 0	Z2: -22.5	YC: 0	ZC: -45	R: 22.5
ARC					



CNC 8055
MC

REF. 0601

Geometry of the example in the PROFILE EDITOR

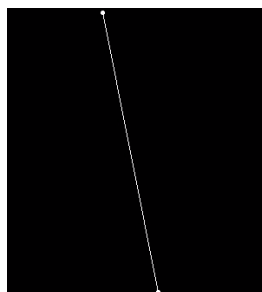


XZ4 PROFILE


PROFILE PROGRAMMING

STARTING POINT X1: 90 Z1: -22.5
STRAIGHT X2: 110 Y2: -45

Finally, we will draw the depth profile for the front of the igloo. This depth profile is drawn from up down because it acts as a cutoff of the previous profile:



GENERAL CONDITIONS		
X	Starting point on the X axis	0
Y	Starting point on the Y axis	0
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	30
I1	Maximum penetration pass when roughing	3
I2	Maximum Z pass when semi-finishing	1
Fz	Penetration feedrate	100

ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	1
D	Tool offset	1
β	Sideways penetration angle	90
Δ	Maximum roughing pass	0

5.

3D CYCLES
EXAMPLE: IGLOO



FAGOR 

CNC 8055
MC

REF. 0601

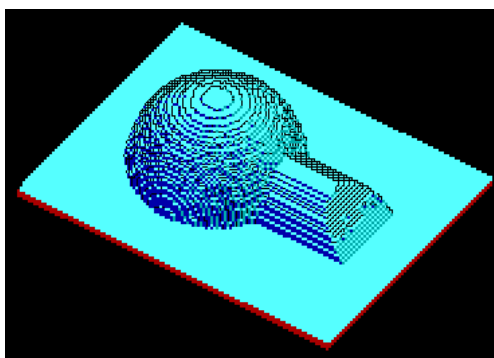
5.

3D CYCLES
EXAMPLE: IGLOO

FINISHING		
F	Finishing feedrate	500
S	RPM	2000
	Clockwise turning direction	
T	Number of the tool to be used	7
D	Tool offset	7
R	Tool radius	3
δ	Finishing stock	0.1
ε	Finishing pass	1
	Zigzag finishing	

5.3.2 PROGRAM SIMULATION

1. We position the red cursor on the first cycle of the program and then press [GRAPHICS].
2. Then, we press [RESET] and then [START].





5.



CNC 8055
MC

REF. 0601

CNC 8055
MC

80