CNC 8055 MC

Examples manual

REF. 1010

FAGOR AUTOMATION

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

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

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

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

The content of this manual and its validity for the product described here has been verified. Even so, involuntary errors are possible, 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. We appreciate your suggestions for improvement.

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

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

INDEX

CHAPTER 1 CONTOURS

1.1 Introduction ... 5
1.2 Creating a part-program ... 5
1.3 Example: profile milling using 12 points ... 6
1.3.1 Positioning 1 ... 7
1.3.2 Profile 1 ... 8
1.3.3 Program simulation ... 9
1.4 Example: profile milling using the profile editor ... 10
1.4.1 Profile milling ... 11
1.4.2 Program simulation ... 12

CHAPTER 2 MACHINING CANNED CYCLES

2.1 Creating programs with machining canned cycles ... 13
2.2 Example: using machining canned cycles ... 14
2.2.1 Profile milling ... 15
2.2.2 Profile milling (milling prior to drilling) ... 17
2.2.3 Simple pocket (top pocket) ... 19
2.2.4 Simple pocket (bottom pocket) ... 20
2.2.5 Circular pocket 1 ... 21
2.2.6 Drilling definition ... 22
2.2.7 Program simulation ... 25

CHAPTER 3 COORDINATE TRANSFORMATION CYCLES

3.1 Example: using the mirror image cycle ... 28
3.1.1 Profile milling ... 29
3.1.2 Rectangular pocket ... 31
3.1.3 Drilling 1 (angular repetition) ... 32
3.1.4 Drilling 1 (linear repetition) ... 34
3.1.5 Drilling 1 (angular repetition) ... 36
3.1.6 Mirror image ... 38
3.1.7 Program simulation ... 39
3.2 Example: using the three mirror image commands ... 40
3.2.1 Profile milling ... 41
3.2.2 Mirror images ... 43
3.2.3 Pattern rotation ... 44
3.2.4 Profile milling ... 45
3.2.5 Program simulation ... 47
3.3 Example: pattern rotation ... 48
3.3.1 Profile milling ... 49
3.3.2 Pattern rotation ... 51
3.3.3 Circular pockets ... 52
3.3.4 Drilling wity linear repetition ... 53
3.3.5 Pattern rotation ... 55
3.3.6 Program simulation ... 56

CHAPTER 4 2D CYCLES

4.1 Example: joint ... 57
4.1.1 2D profile pocket ... 58
4.1.2 Circular pockets ... 60
4.1.3 Program simulation ... 61
4.2 Example: cam ... 62
4.2.1 2D profile pockets ... 63
4.2.2 Drilling ... 68
4.2.3 Circular pocket 1 ... 69
4.2.4 Program simulation ... 70

FAGOR

·MC· Option

REF. 1010

CHAPTER 5 3D CYCLES

5.1 Introduction 71
5.2 Example: toroid 72
5.2.1 3D profile pockets 73
5.2.2 Program simulation 77
5.3 Example: igloo 78
5.3.1 3D profile pockets 79
5.3.2 Program simulation 82

FAGOR

·MC· Option

REF. 1010

CONTOURS

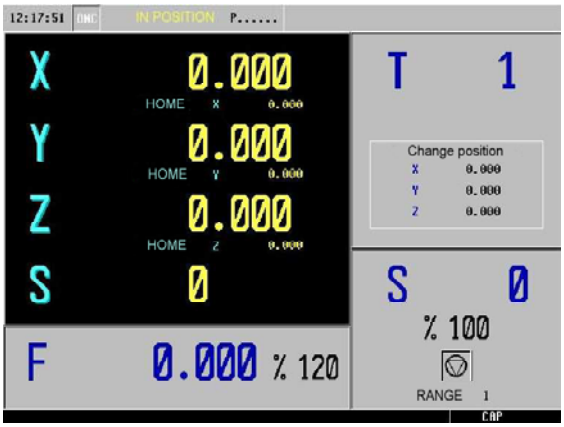
1

1.1 Introduction

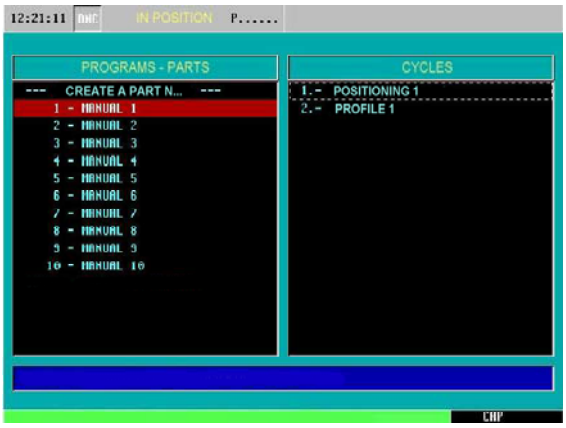
Programming in conversational mode consists in filling in a number of cycles depending the machining operations to perform. These cycles may be stored in a program or executed without storing them. Here are some examples of machining operations normally used with this method.

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.



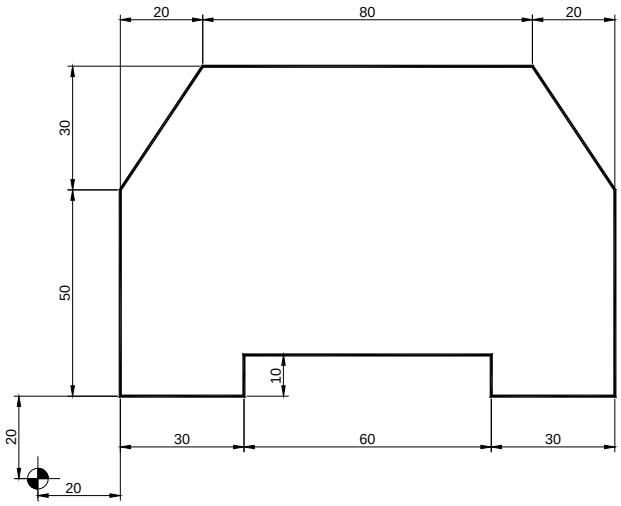
FAGOR

·MC· Option

REF. 1010

1.3 Example: profile milling using 12 points

To machine the contour of this figure:



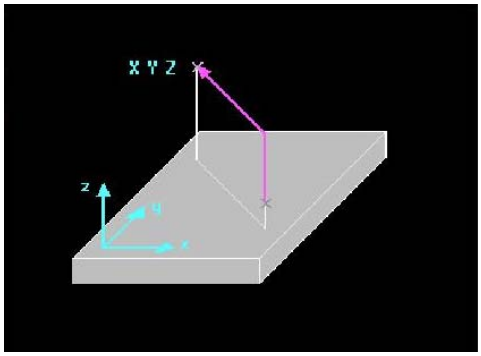
We will use the following tools:

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:

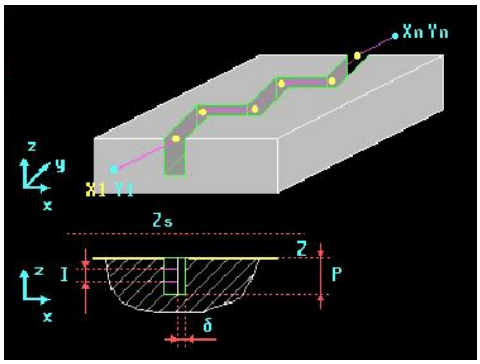
SELECT POSITION TO INSERT POSITIONING 1

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



1.3.2 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:

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.

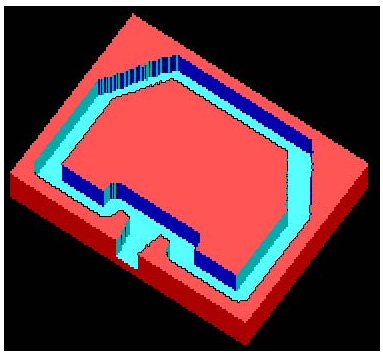
Once the cycle is completed, we proceed the same way as with the positioning cycle; i.e. pressing [PPROG] to insert it in the desired position of the program and pressing [ENTER].

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:

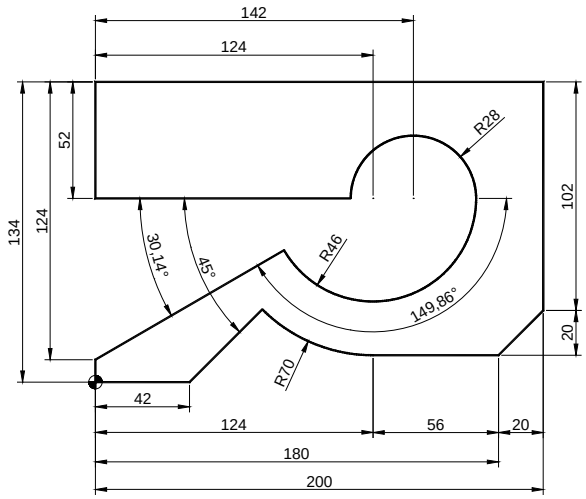


2. We press [RESET] and then [START].



1.4 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:

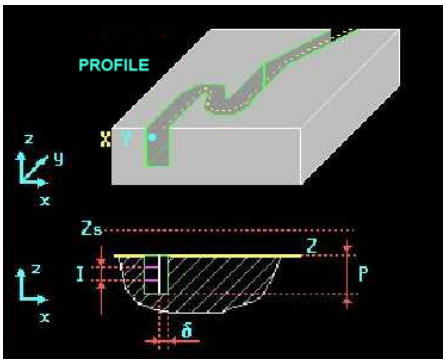
Operations	Tools
Positioning	
Profile milling (roughing)	Flat endmill Ø6 T1 D1
Profile milling (finishing)	Flat endmill Ø4 T2 D2

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:

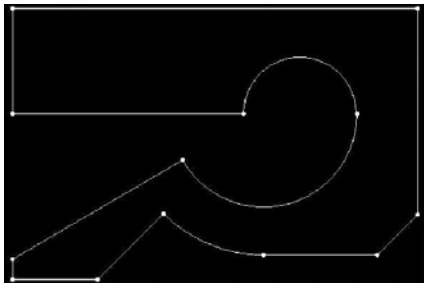
STARTING POSITION		
X	Starting point on the X axis	-25
Y	Starting point on the Y axis	-25
	Profile	1

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



When the drawing is completed, we press [END] and the CNC will request a comment for that drawing. Once we confrimed 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

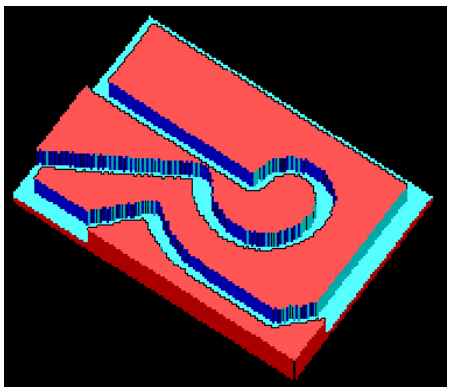
ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	1
D	Tool offset	1
	Right hand tool radius compensation	

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

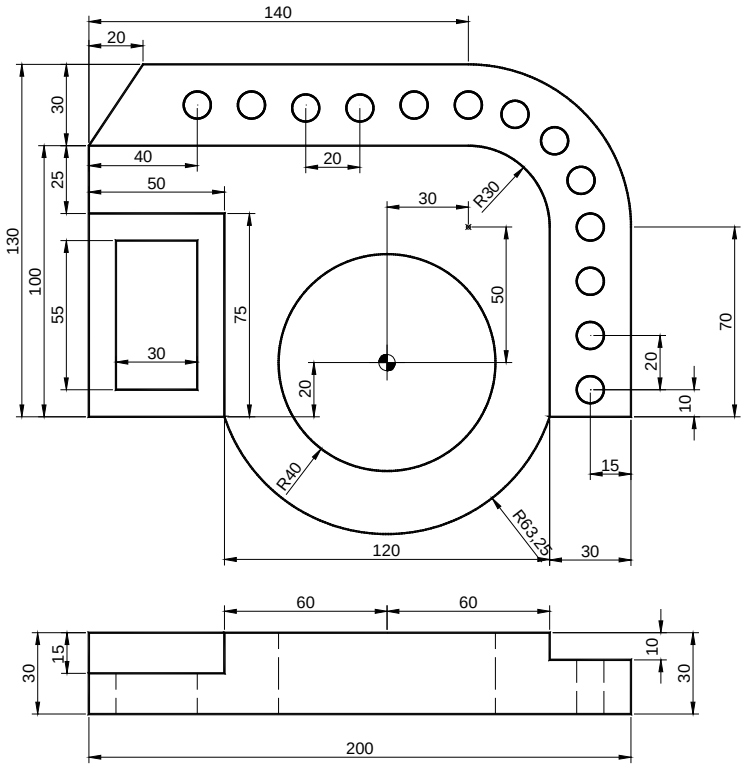
2.1 Creating programs with machining canned cycles

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:



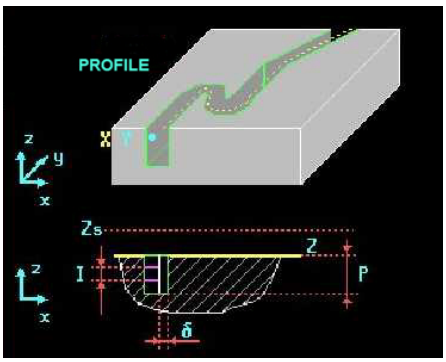
We will use the following tools:

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]:

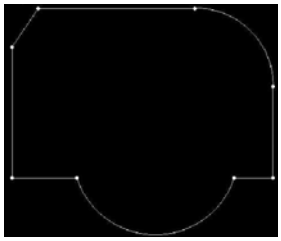
STARTING POSITION		
X	Starting point on the X axis	-140
Y	Starting point on the Y axis	-50
	Profile	2

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 ARC	X2: 60	Y2: -20	XC: 0	YC: 0
STRAIGHT	X2: 90	Y2: -20		
STRAIGHT	X2: 90	Y2: 50		
COUNTERCLOCKWISE ARC	X2: 30	Y2: 110	XC: 30	YC: 50
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

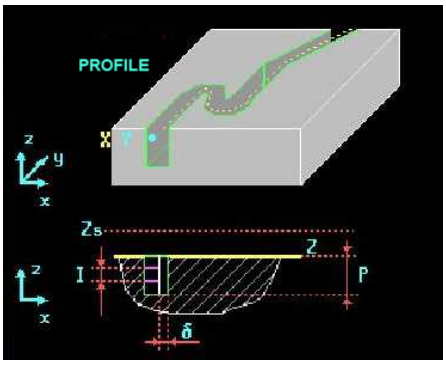
ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	3
D	Tool offset	3
	Right hand tool radius compensation	

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

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]:

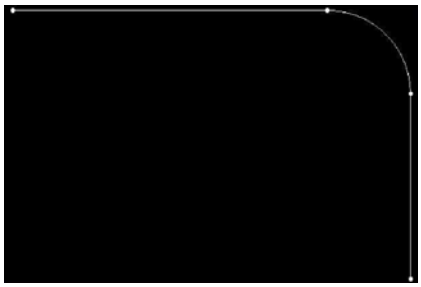
STARTING POSITION		
X	Starting point on the X axis	-140
Y	Starting point on the Y axis	95
	Profile	3

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:

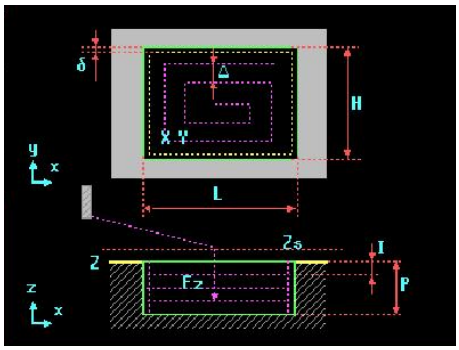
GENERAL CONDITIONS		
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	10
I	Value of each pass	3
Fz	In feed	100

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

2.2.3 Simple pocket (top pocket)

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



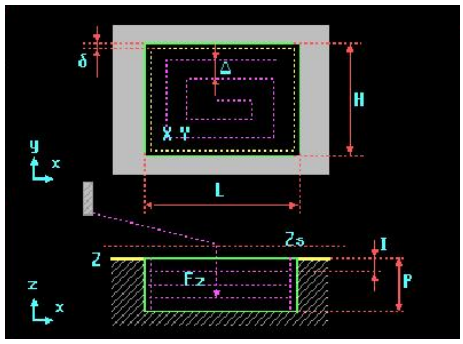
The position of the pocket may be defined in two ways, from the bottom left corner, or from the center; the [TWO-COLOR] key toggles this option.

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

2.2.4 Simple pocket (bottom pocket)

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

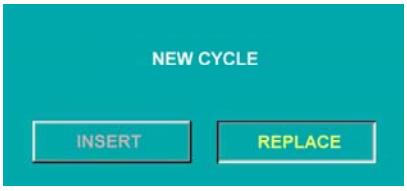


The dimensions of this pocket are different from the previous one, therefore, the previous one may be changed by simply recalling it from the program using the [RECALL] key and changing the starting X and Y, the length L and the width H:

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

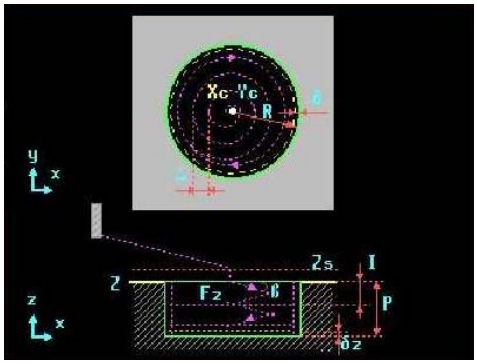
MACHINING CONDITIONS		
		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	1
D	Tool offset	1
	Clockwise machining	
Δ	Finishing pass	0
δ	Finishing stock	0.2

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:



2.2.5 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

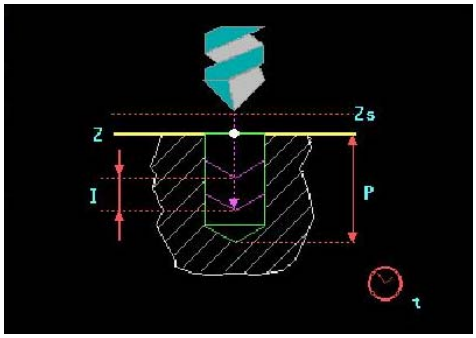
ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	1
D	Tool offset	1
β	Penetrating angle	90
	Clockwise machining	
Δ	Roughing pass	0

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

2.2.6 Drilling definition

To go into the DRILLING 1 cycle, we press:

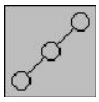




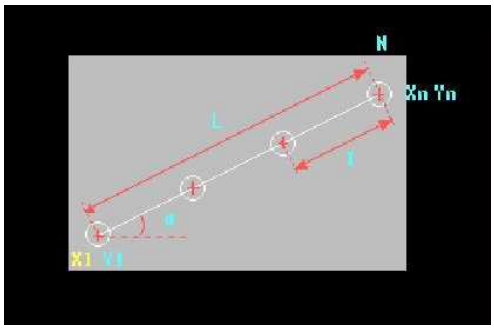
When defining the drilling, we begin from the holes at the bottom right side of the part. The first hole is drilled at X75 Y-10

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



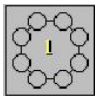
Once the first hole is programmed, we repeat it linearly pressing the [LINEAR REPETITION] key.



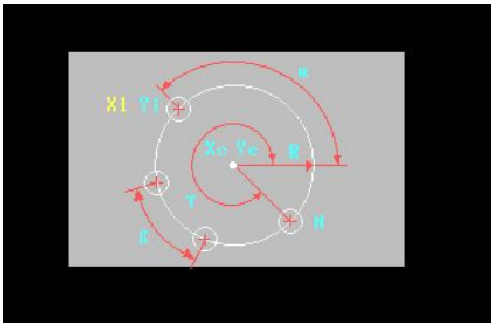
In this cycle, we can enter the necessary data for the linear repetition of different shapes by simply changing the window mode. To toggle this window, we press the [TWO-COLOR] key:

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.

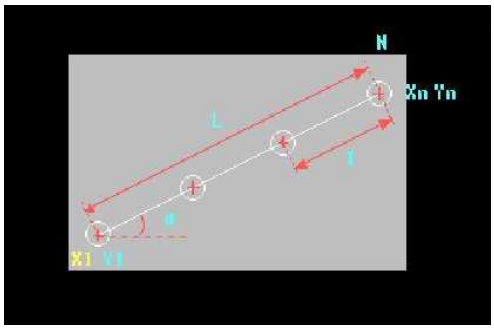


In this cycle, we can also change the data that the CNC needs depending on what we (the programmers) have, using the [TWO-COLOR] key.

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

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.

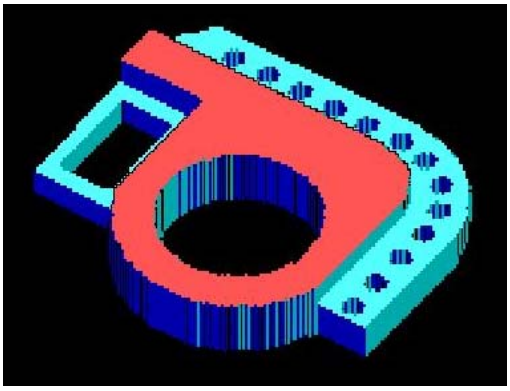


GENERAL CONDITIONS		
X1	X coordinate of the first hole	30
Y1	Y coordinate of the first hole	95
	Linear repetition method 1	
Xn	X coordinate of the last point	-70
Yn	Y coordinate of the last point	95
N	Number of positions	6

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

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

·MC· Option

REF. 1010

COORDINATE TRANSFORMATION CYCLES

3

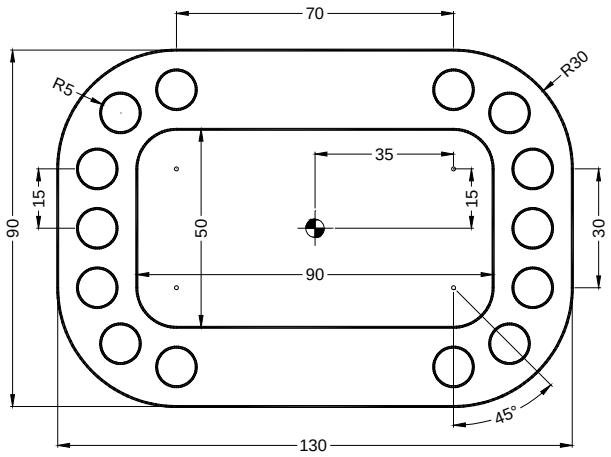
Coordinate transformation cycles are those where, based on an original program, a function is applied to repeat the same geometry differently. There are four types of functions for coordinate transformation:

- 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.



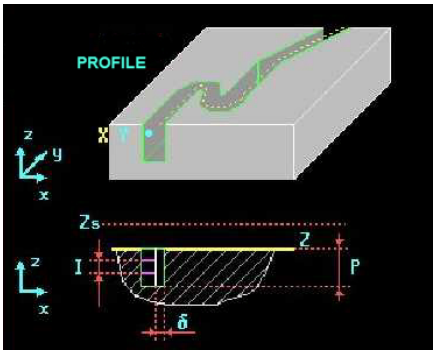
We will use the following tools:

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]:

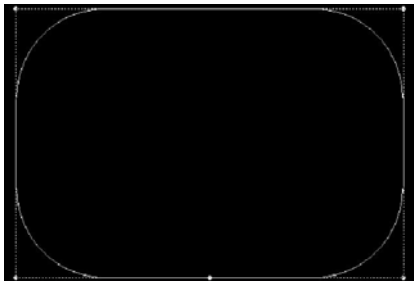
STARTING POSITION		
X	Starting point on the X axis	0
Y	Starting point on the Y axis	-75
	Profile	4

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



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

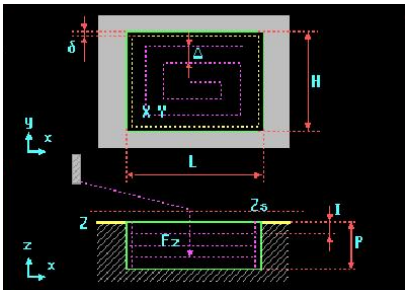
GENERAL CONDITIONS		
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	20
I	Value of each pass	2
Fz	In feed	100

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

3.1.2 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

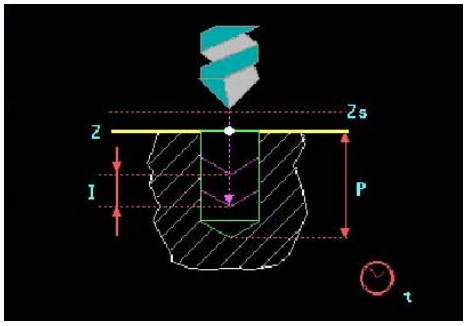
ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	5
D	Tool offset	5
β	Penetrating angle	90
	Clockwise machining	
Δ	Roughing pass	0

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

3.1.3 Drilling 1 (angular repetition)

To go into the DRILLING 1 cycle, we press:

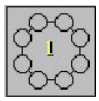




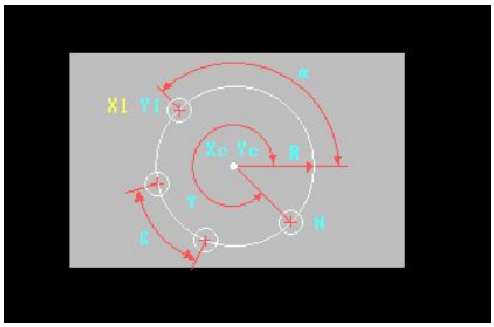
When defining the drilling, we begin from the hole at the bottom right side of the part. This hole will be positioned at X35 Y-35:

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.



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:

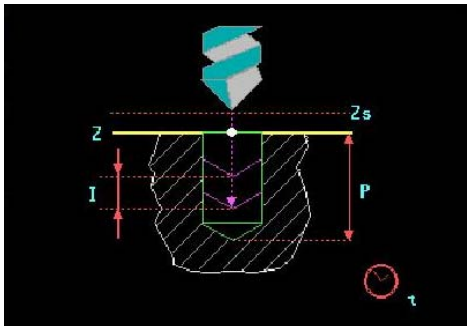
GENERAL CONDITIONS		
X1	X coordinate of the first hole	35
Y1	Y coordinate of the first hole	-35
	Angular repetition method 1	
Xc	X coord. of the arc center	35
Yc	Y coord. of the arc center	-15
N	Number of positions	2
τ	Angular distance of the last cycle, referred to the horizontal axis X	-45

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

3.1.4 Drilling 1 (linear repetition)

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

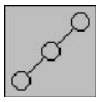




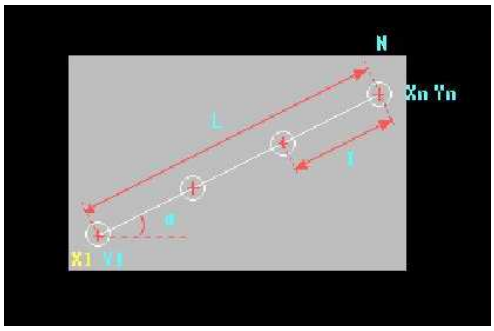
The drill bit will position at the first hole, X55 Y-15, and we will then do a linear repetition.

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.



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:

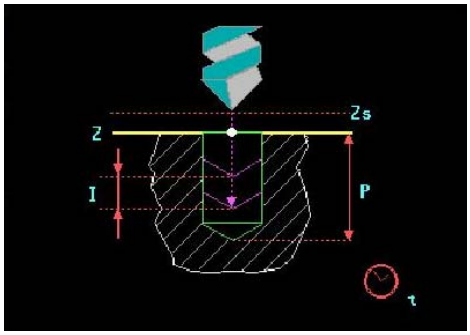
GENERAL CONDITIONS		
X1	X coordinate of the first hole	55
Y1	Y coordinate of the first hole	-15
	Linear repetition method 1	
Xn	X coordinate of the last hole	55
Yn	Y coordinate of the last hole	0
N	Number of positions	2

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

3.1.5 Drilling 1 (angular repetition)

To go into the DRILLING 1 cycle, we press:

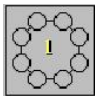




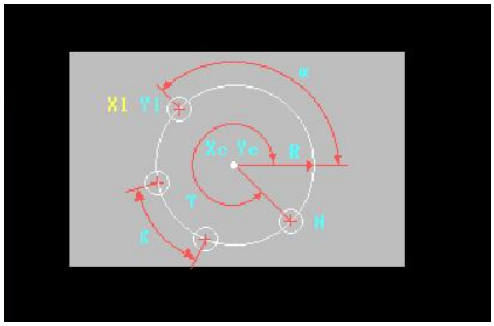
The drill bit will position at the third hole, X55 Y15, and we will then do an angular repetition.

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.



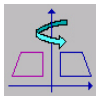
In this cycle, we can enter the necessary data for the angular repetition of different shapes by simply changing the window mode. To toggle this window, we press the [TWO-COLOR] key:

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

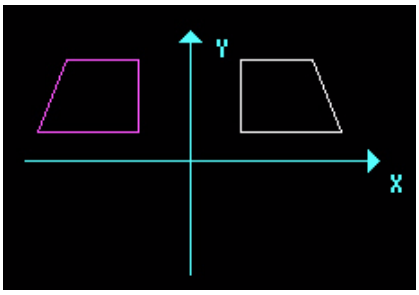
MACHINING		
Zs	Safety Z coordinate	2
Z	Surface Z coordinate	0
P	Total pocket depth	20
t	Dwell at the bottom (seconds)	0
I	Penetration step	5
F	Machining feedrate	600
S	RPM	750
	Clockwise turning direction	
T	Number of the tool to be used	4
D	Tool offset	4

3.1.6 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:



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

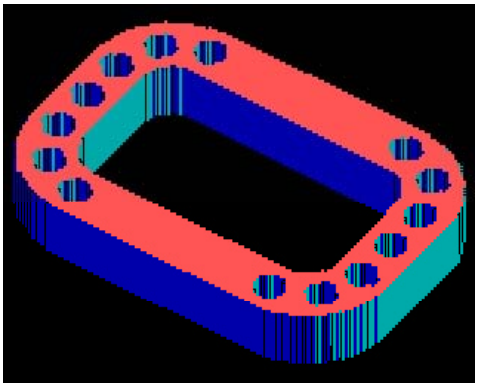
PROGRAM	
N1	Label located above the first machining operation to be repeated
N2	Label located under the last machining operation to be repeated
RTP	Command located under the MIRROR IMAGE cycle

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

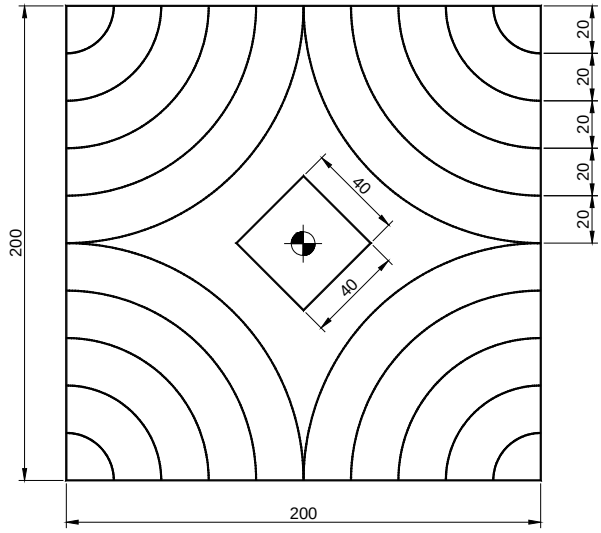
3.1.7 Program simulation

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].



3.2 Example: using the three mirror image commands

To make the following part:



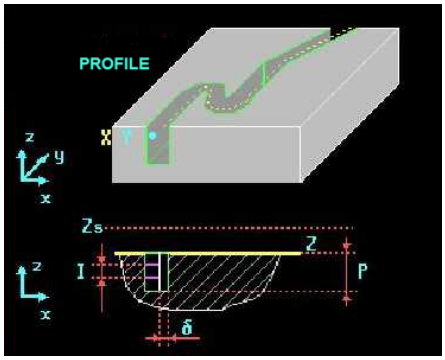
The following example uses the three commands for mirror image and pattern rotation. We will use the following tools:

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:

3.2.1 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		

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:

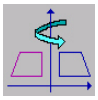
GENERAL CONDITIONS		
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	5
I	Value of each pass	2
Fz	In feed	100

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	

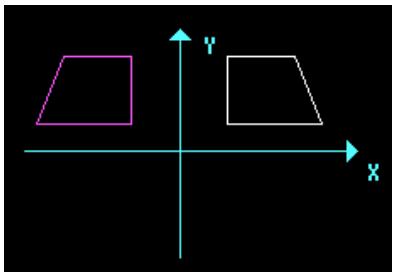
FINISHING		
F	Finishing feedrate	0
S	RPM	0
	Clockwise turning direction	
T	Number of the tool to be used	0
D	Tool offset	0
δ	Finishing stock	0

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

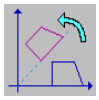
Mirror image of the drawing in XY.

CONDITIONS		
NEW	Axis upon which mirror image is applied.	XY

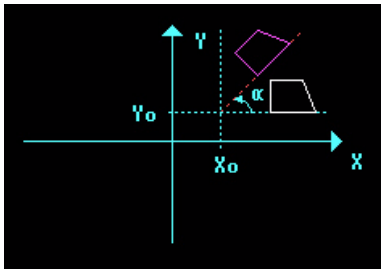
Press [PPROG] and [ENTER]. Enter the PROFILE MILLING cycle again

3.2.3 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).

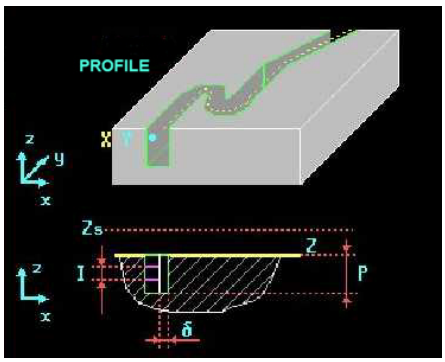


CONDITIONS		
New		
τo	Rotation angle	45
Xo	X position of the rotation center	0
Yo	Y position of the rotation center	0

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

3.2.4 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]:

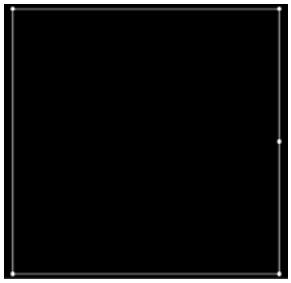
STARTING POSITION		
X	Starting point on the X axis	20
Y	Starting point on the Y axis	0
	Profile	6

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

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



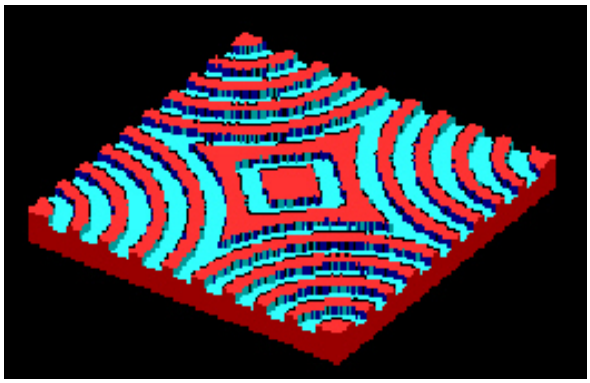
After machining the square, we cancel the pattern rotation. We do this in the turning cycle itself by selecting the CANCEL option.

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 45º (0,0) 6 .- PROFILE MILLING G73;} PATTERN ROTATION 7.- POSITIONING 1

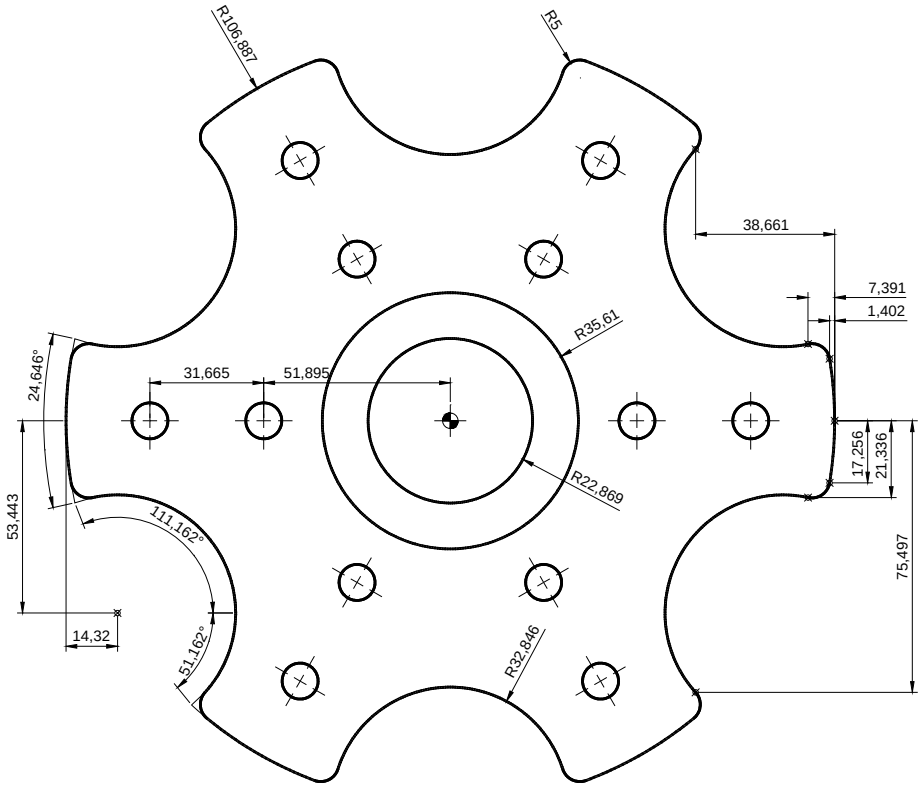
3.2.5 Program simulation

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].



3.3 Example: pattern rotation

To make the following part:



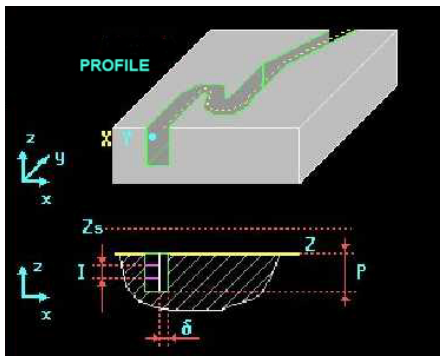
We will use the following tools:

Operations	Tools
Contour machining (roughing)	Flat endmill Ø6 T1 D1
Contour machining (finishing)	Flat endmill Ø4 T2 D2
Drilling	Drill bit Ø10 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]:

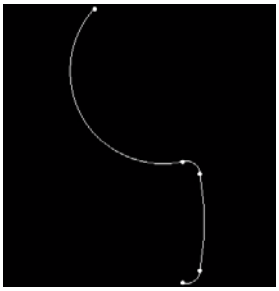
STARTING POSITION		
X	Starting point on the X axis	99.496
Y	Starting point on the Y axis	-21.336
	Profile	7

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

PROFILE PROGRAMMING

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

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	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	

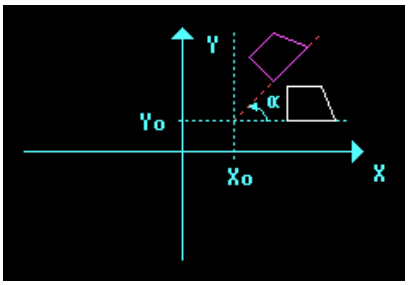
FINISHING		
F	Finishing feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	2
D	Tool offset	2
δ	Finishing stock	0.2

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.

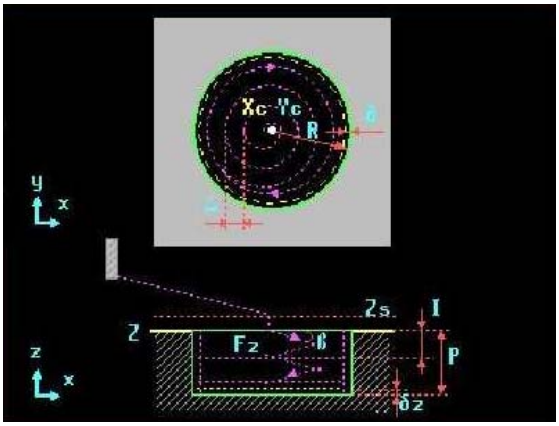


We press the [ISO] key and use the [LEVEL CYCLE] key to find the cycle corresponding to the labels (Level 4).

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.

3.3.3 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.

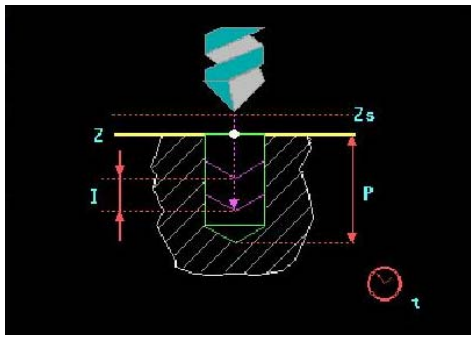


GENERAL CONDITIONS			
		POCKET 1	POCKET 2
Xc	Pocket center in X	0	0
Yc	Pocket center in Y	0	0
R	Pocket radius	35.61	22.869
Zs	Safety Z coordinate	2	2
Z	Surface coordinate	0	0
P	Total depth in absolute coordinates	15	30
I	Value of each pass	3	3
Fz	In feed	100	100

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

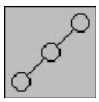
FINISHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	2
D	Tool offset	2
θ	Tool penetrating angle when finishing	90
	Clockwise machining	
δ	Finishing stock	0.2
δz	Excess material in Z	0.1
N	Number of finishing passes in Z	1

3.3.4 Drilling wity linear repetition

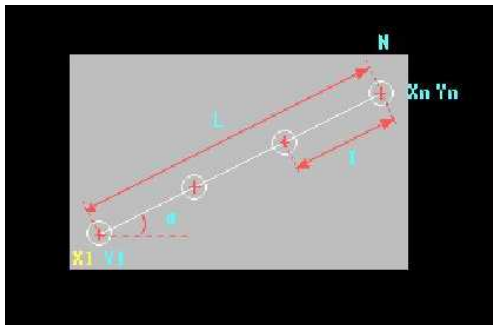


GENERAL CONDITIONS		
X	X coordinate of the first hole	51.895
Y	Y coordinate of the first hole	0
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	30
I	Value of each pass	5
t	Dwell at the bottom	0

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

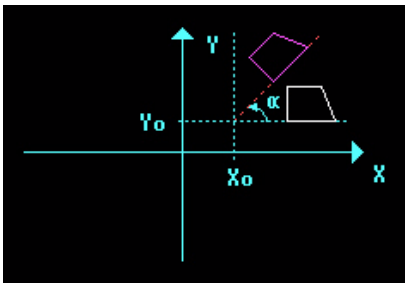
DRILLING		
Zs	Safety Z coordinate	2
Z	Surface Z coordinate	0
P	Total pocket depth	30
t	Dwell at the bottom (seconds)	0
I	Penetration step	5
F	Machining feedrate	600
S	RPM	750
	Clockwise turning direction	
T	Number of the tool to be used	4
D	Tool offset	4

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		
τo	Rotation angle	60
Xo	X position of the rotation center	0
Yo	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

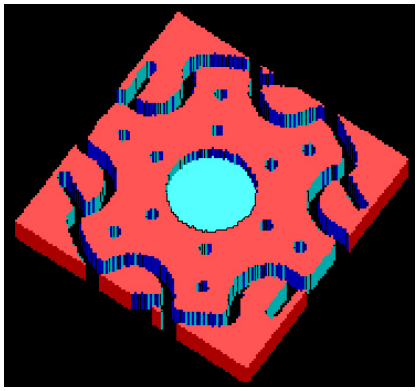
The definition of the last two labels must be different from the first two, otherwise the repetition will be wrong. That's why N3 and N4 are defined. There is no need to cancel it because it is the last rotation in the program.

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;}

3.3.6 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].

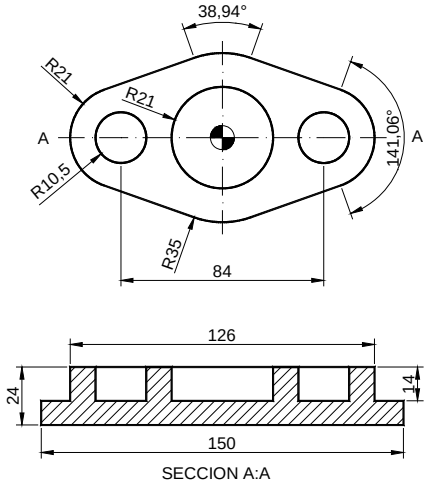


2D CYCLES

4

4.1 Example: joint

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



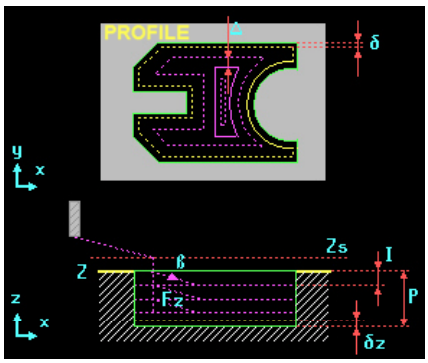
We will use the following tools:

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]:

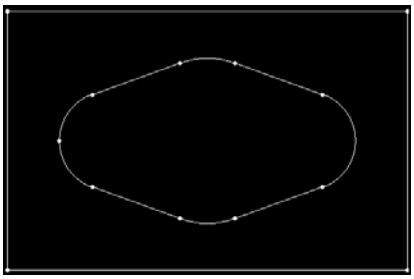
STARTING POSITION		
X	Starting point on the X axis	-85
Y	Starting point on the Y axis	-55
	Profile	8

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 ARC	XC: -42	YC: 0	R: 21		
STRAIGHT	TANG: YES				
COUNTERCLOCKWISE ARC	XC: 0	YC: 0	R: 35	TANG: YES	
STRAIGHT	TANG: YES				
COUNTERCLOCKWISE ARC	XC: 42	YC: 0	R: 21	TANG: YES	
STRAIGHT	TANG: YES				
COUNTERCLOCKWISE ARC	XC: 0	YC: 0	R: 35	TANG: YES	
STRAIGHT	TANG: YES				
COUNTERCLOCKWISE ARC	X2: -63	Y2: 0	XC: -42	YC: 0	R: 21 TANG: YES

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:

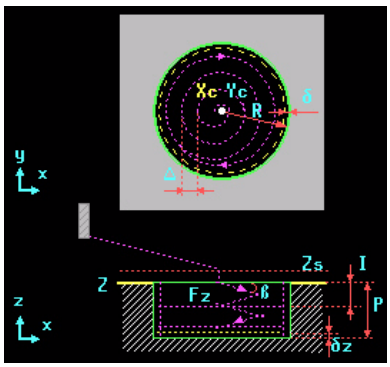
GENERAL CONDITIONS		
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	14
I	Value of each pass	3
Fz	Penetration feedrate	100

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

4.1.2 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

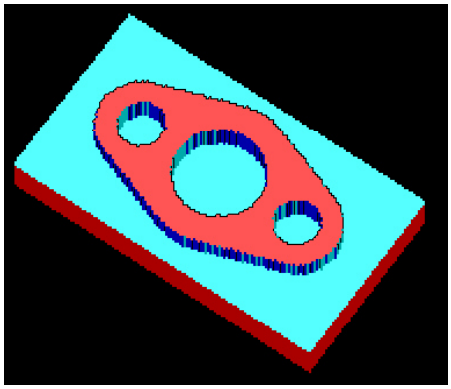
And last, we program a positioning to move the tool to the safety distance in Z.

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

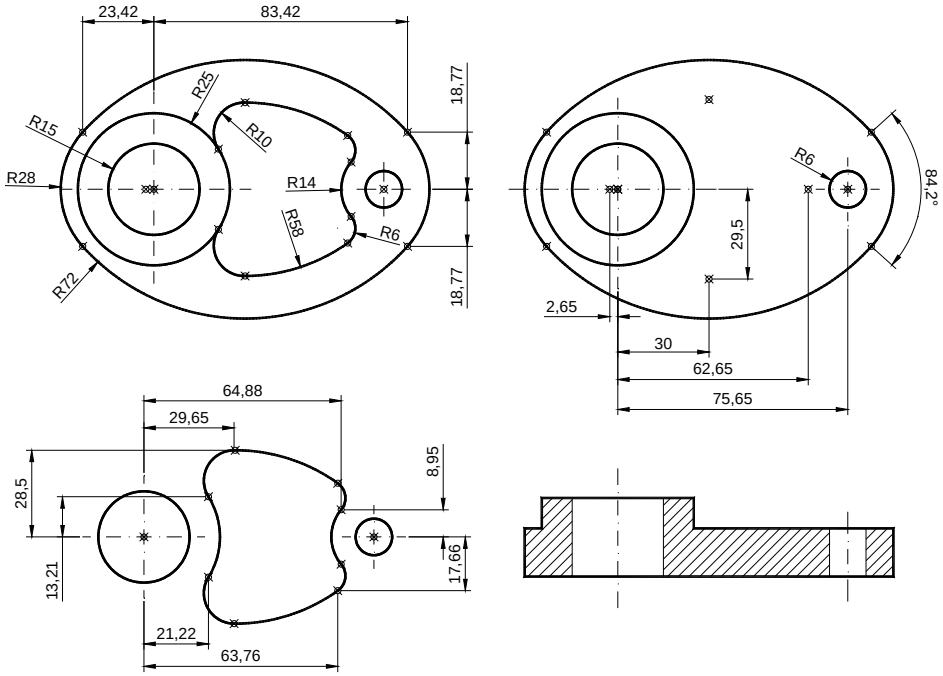
4.1.3 Program simulation

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].



4.2 Example: cam

To make the following part:



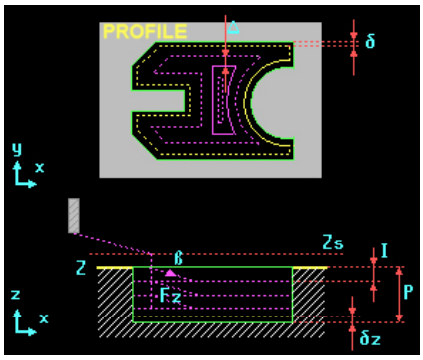
We will use the following tools:

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]:

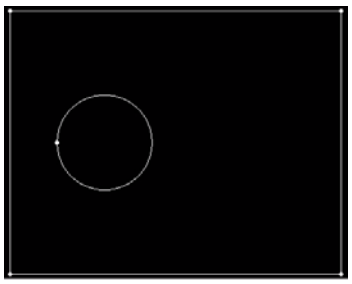
STARTING POSITION		
X	Starting point on the X axis	-50
Y	Starting point on the Y axis	-70
	Profile	9

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

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

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.

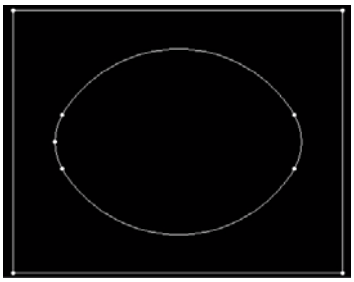
STARTING POSITION		
X	Starting point on the X axis	-50
Y	Starting point on the Y axis	-70
	Profile	10

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 ARC	XC: 0	YC: 0	R: 28	
COUNTERCLOCKWISE ARC	X2: 83.42	Y2: -18.773	R: 72	TANG: YES
COUNTERCLOCKWISE ARC	X2: 83.42	Y2: 18.773	YC: 0	R: 28
COUNTERCLOCKWISE ARC	X2: -23.42	R: 72	TANG: YES	
COUNTERCLOCKWISE ARC	X2: -28	Y2: 0	R: 28	

Geometry of the example in the PROFILE EDITOR



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.

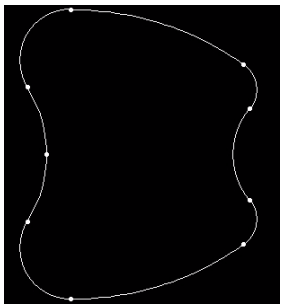
STARTING POSITION		
X	Starting point on the X axis	25
Y	Starting point on the Y axis	0
	Profile	11

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		
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

Geometry of the example in the PROFILE EDITOR



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

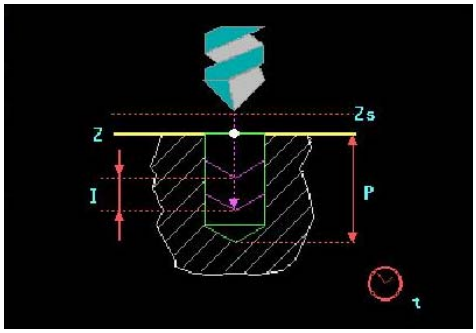
ROUGHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	1
D	Tool offset	1
β	Sideways penetration angle	90
Δ	Maximum roughing pass	0

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

4.2.2 Drilling

To go into the DRILLING 1 cycle, we press:





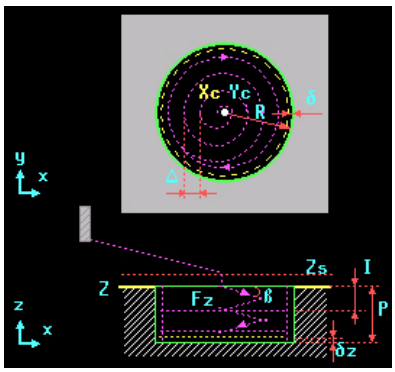
Drilling definition:

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

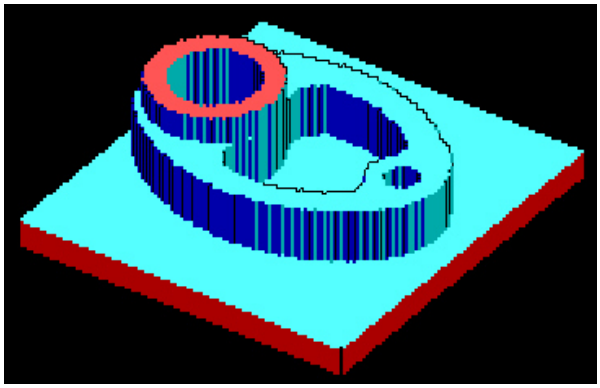
FINISHING		
F	Machining feedrate	1000
S	RPM	1000
	Clockwise turning direction	
T	Number of the tool to be used	2
D	Tool offset	2
θ	Tool penetrating angle when finishing	90
	Clockwise machining	
δ	Finishing stock	0.2
δz	Excess material in Z	0.1
N	Number of finishing passes in Z	1

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].



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

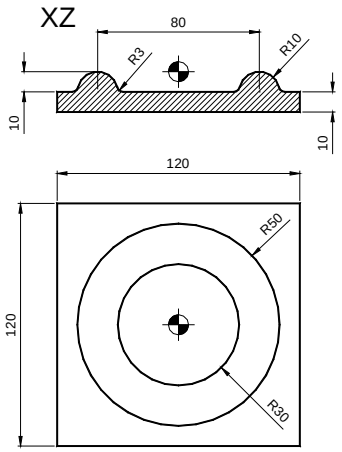
The first two boxes indicate the part number and the number for the XY drawing of the 3D pocket respectively. Boxes PZ1, PZ2, PZ3 and PZ4 are the depth profiles in the programming order of the XY profiles.

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:

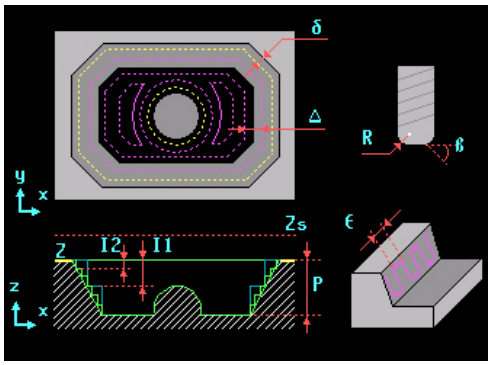
Operations	Tools
Machining of the 3D pocket (roughing)	End mill Ø6 T1 D1
Machining of the 3D pocket (finishing)	End mill Ø4 T2 D7

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:

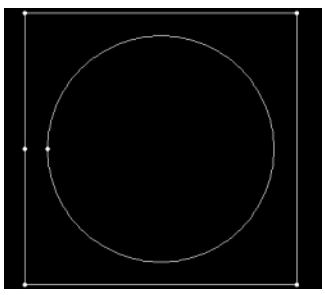
POCKET AND PROFILE NUMBERING		
POCK. 3D	Name of the Pocket	1
P. XY	XY plane profile file	11
P. Z1	Z-profile file for the 1st XY profile	12
P. Z2	Z-profile file for the 2nd XY profile	13
P. Z3	Z-profile file for the 3rd XY profile	0
P. Z4	Z-profile file for the 4th XY profile	0

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



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

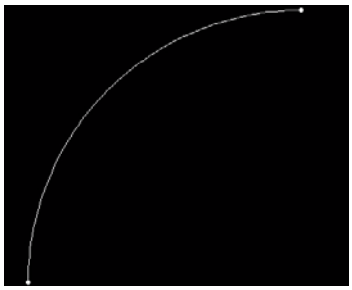


In PZ2, we draw the depth profile for the island. This time, we define it from down up:

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.

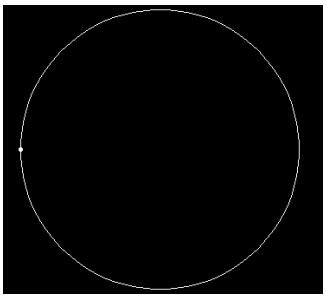
POCKET AND PROFILE NUMBERING		
POCK. 3D	Name of the Pocket	2
P. XY	XY plane profile file	21
P. Z1	Z-profile file for the 1st XY profile	22
P. Z2	Z-profile file for the 2nd XY profile	23
P. Z3	Z-profile file for the 3rd XY profile	0
P. Z4	Z-profile file for the 4th XY profile	0

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

Geometry of the example in the PROFILE EDITOR

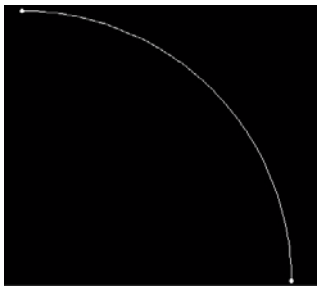


Being an emptying operation, this depth profile is defined from up down and the starting point of the X coordinate must coincide with the X of the starting point of the previous XY profile; this way, this depth profile will be used for the whole XY process.

PROFILE PROGRAMMING

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

Geometry of the example in the PROFILE EDITOR



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

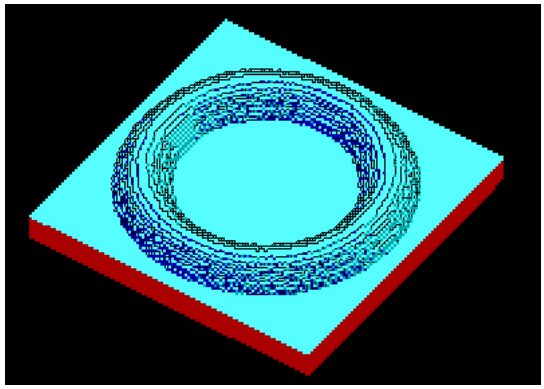
GENERAL CONDITIONS		
X	Starting point on the X axis	-60
Y	Starting point on the Y axis	0
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	10
I1	Maximum penetration pass when roughing	3
I2	Maximum Z pass when semi-finishing	1
Fz	Penetration feedrate	100

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	

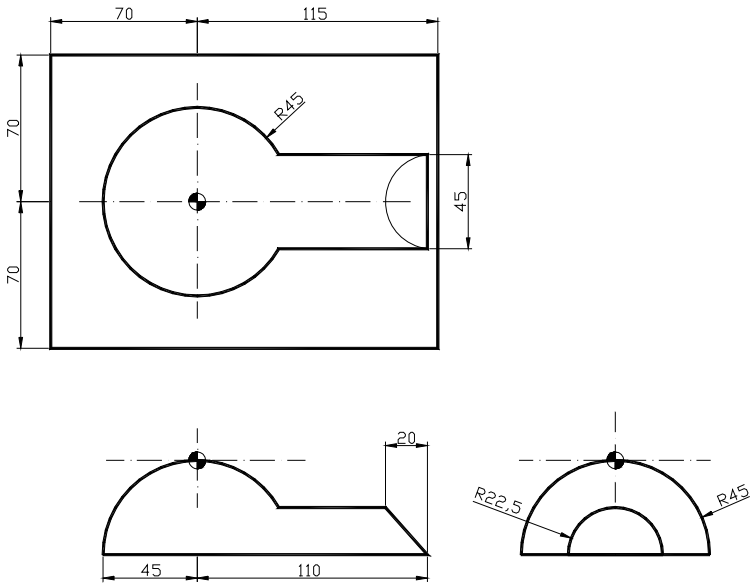
5.2.2 Program simulation

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].



5.3 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:

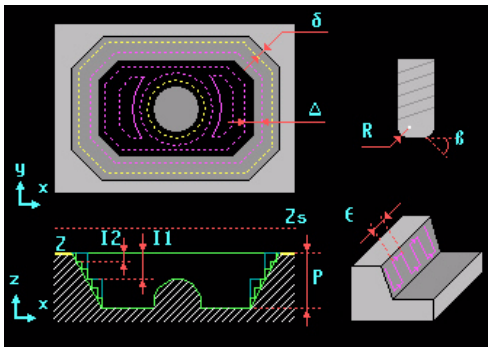
Operations	Tools
Machining of the 3D pocket (roughing)	End mill Ø6 T1 D1
Machining of the 3D pocket (finishing)	End mill Ø4 T2 D7

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.



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.

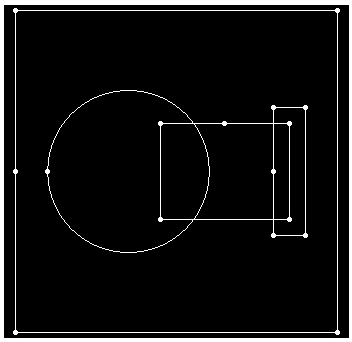
POCKET AND PROFILE NUMBERING		
POCK. 3D	Name of the Pocket	3
P. XY	XY plane profile file	30
P. Z1	Z-profile file for the 1st XY profile	31
P. Z2	Z-profile file for the 2nd XY profile	32
P. Z3	Z-profile file for the 3rd XY profile	33
P. Z4	Z-profile file for the 4th XY profile	34

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			

Geometry of the example in the PROFILE EDITOR



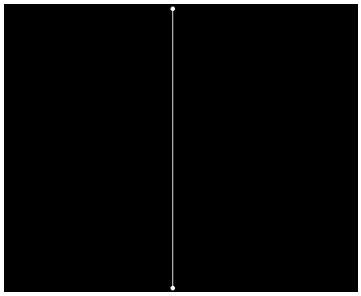
The order these XY geometries are programmed is important because different depth profiles will be programmed respectively and making the starting points in X and in Y coincide.

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:

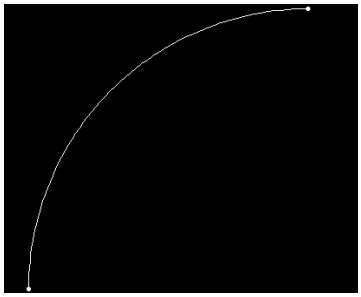


XZ2 PROFILE:

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:



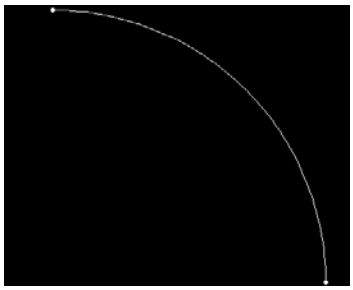
XZ3 PROFILE:

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

Geometry of the example in the PROFILE EDITOR

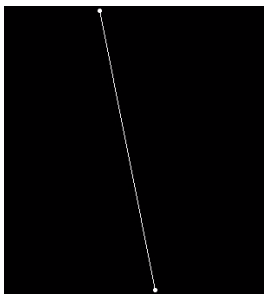


XZ4 PROFILE

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:



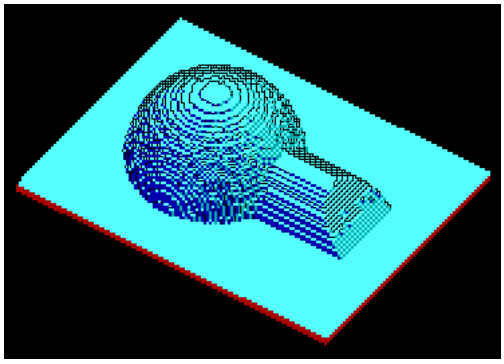
GENERAL CONDITIONS		
X	Starting point on the X axis	0
Y	Starting point on the Y axis	0
Zs	Safety Z coordinate	2
Z	Surface coordinate	0
P	Total depth in absolute coordinates	30
I1	Maximum penetration pass when roughing	3
I2	Maximum Z pass when semi-finishing	1
Fz	Penetration feedrate	100

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

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].





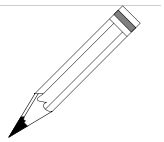
5.

FAGOR

·MC· Option

REF. 1010

5.



FAGOR

·MC· Option

REF. 1010