EXAMPLES MANUAL

REF. 0402

CNC 8070

FAGOR

FAGOR

CNC 8070

REF. 0402

EXAMPLES MANUAL

INDEX

1. CONCEPTS

2. CONVERSATIONAL PROGRAMMING

3. ISO PROGRAMMING

4. PARAMETRIC PROGRAMMING

CONCEPTS

1

1 TECHNICAL SPECIFICATIONS

TYPE OF MACHINE

- 3-axis machining center.

MACHINING CONDITIONS

- The machining technical data are based on using construction steel of up to 700 N/mm2.
- The feedrate and rpm values obtained will depend on the type of tool used in each example.

2 BASIC CNC OPERATING CONCEPTS

USEFUL KEYS


(a) Key for editing and simulation


(b) Cycle-start button.

(c) Reset button.


(d) Execution button.


(e) Cycle stop.

OPEN A PROGRAM.

1. Press the panel key (a).
2. Press the OPEN PROGRAM softkey (F1).
3. Use the file managing window to enter the name, number or letters of the exercise and press ENTER to confirm it.
4. Once inside the program, start entering the data.

CONFIGURING A FORM

- A form is the screen displayed at the CNC when editing a canned cycle.
- Press the corresponding softkey to access any form, if it is not displayed, use the [+] softkey,
- To access the CYCLE EDITOR, press F2 and select the relevant cycle. Once the form has been filled out, save it using the [INS] key.
- The forms are basically divided into three blocks: Geometry, Roughing and Finishing.

  Geometry. This block indicates the position where the cycle will be executed as well as its dimensions and the work planes.

  Roughing. Machining conditions for roughing (pass, feedrate, rpm, etc.).

  Finishing. Machining conditions for finishing (pass, feedrate, rpm, etc.).
- All the values entered must be confirmed with [ENTER].

PROGRAM DISPLAY

There are five options when simulating a program and they may be selected alternately with the (a) key.

1. Program test without graphic representation, it will only display the data blocks that make up the program.
2. Solid graphics simulation. It simulates the part as a block that is previously defined by the user.
3. Program test without graphic display, but indicating the various functions, cycles and total execution time.
4. Simulation and program test. The screen is split in two with the program on the left and the solid block on the right.
5. Exactly identical to the previous one, but this type of simulation does not allow modifying any program block.

All the previous simulation options allow selecting the graphic display of the program: 3D lines, sections, XY, XZ, YZ, combined and solid 3D.

PROGRAM SIMULATION.

The program must be opened prior to being simulated. Once the chosen program appears on the screen, use the (a) key to select the simulation mode as described in the previous section.

Press the cycle-start button (b) to start the simulation; if an error message comes up, it may be removed using the reset softkey (c) and the screen will show the program test without graphics.

MACHINING A PROGRAM.

Before executing any program, it should be simulated first to check that that it may be machined properly.

Use the (d) key to choose the best screen type and then proceed like when simulating; in other words, press the (b) key to start machining. Use the cycle-stop key (e) to interrupt the execution of the program at any time.

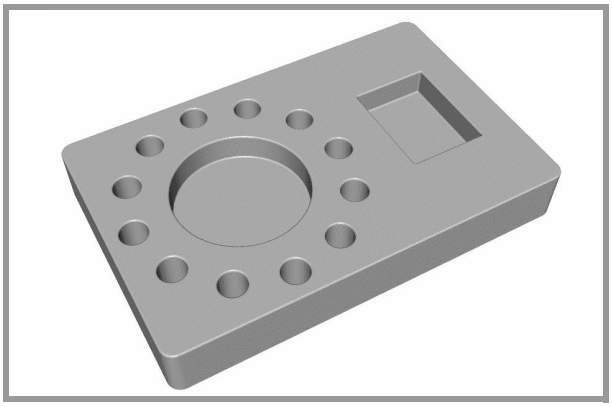
3 GOALS

The goal of the following practical programming examples is to machine stock piece by milling its surface and running a number of cycles using the relevant machining conditions and tools; therefore, we first indicate all the tools, feedrates and rpm for each example.

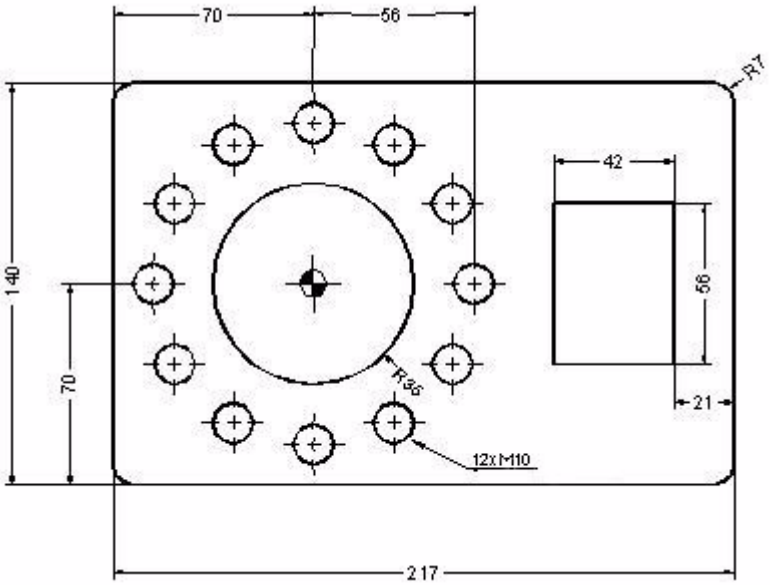
EXERCISE 1. POCKETS

The purpose of the following exercise is to make a cam from a stock whose dimensions are 237 x 160.

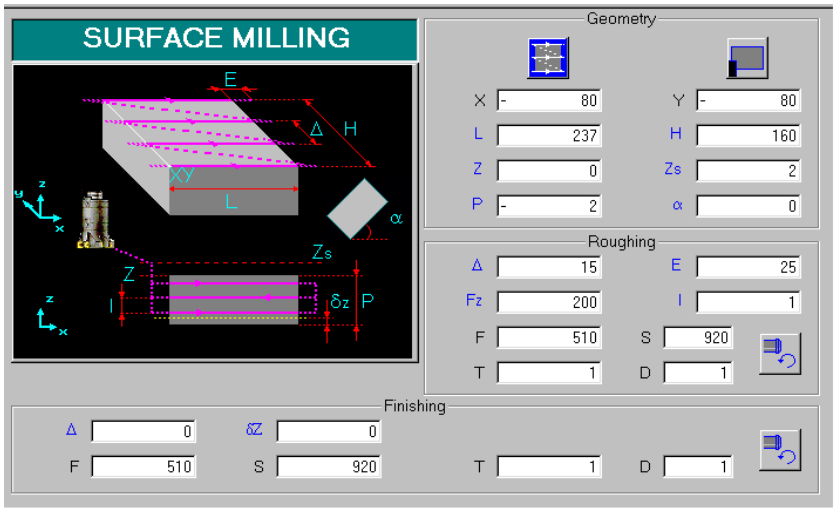
Making this part requires the following steps:



Operations	Tools
Surface milling	Endmill Ø100 T1 D1
Rectangular Boss	Endmill Ø20 T2 D1
Circular pocket	Endmill Ø10 T3 D1
Rectangular pocket	Endmill Ø2 T5 D1
Center punching	Drill bit Ø6 T6 D1
Drill	Drill bit Ø8.5 T7 D1
Tapping	M10 metric tap



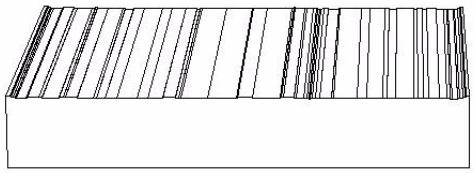
SURFACE MILLING CANNED CYCLE

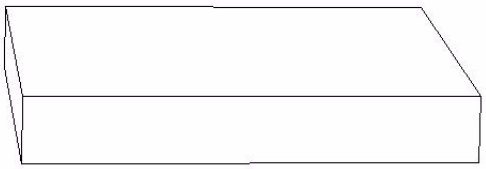


GEOMETRY		
	Type of surface milling. Unidirectional surface milling along X	
	Corner where the surface milling begins. Lower left corner.	
X	X coordinate of the initial corner	-80
Y	Y coordinate of the initial corner	-80
L	Total length in X	237
H	Total length in Y	160
Z	Height coordinate of the surface	0
Zs	Safety Z coordinate	2
P	Total surface milling depth	-2
α	Surface milling angle	0

ROUGHING		
Δ	Roughing pass	15
E	Overshooting for chip relief	25
Fz	Penetrating feedrate at each pass (mm/min.)	200
I	Penetration step in Z	1
F	Machining feedrate in mm/min.	510
S	RPM	920
T	Number of the tool to be used	1
D	Tool offset	1
	Counterclockwise turning direction	

FINISHING		
Δ	Finishing pass	0
δz	Excess material in Z	0
F	Machining feedrate in mm/min.	510
S	RPM	920
T	Tool number	1
D	Tool offset	1
	Counterclockwise turning direction	





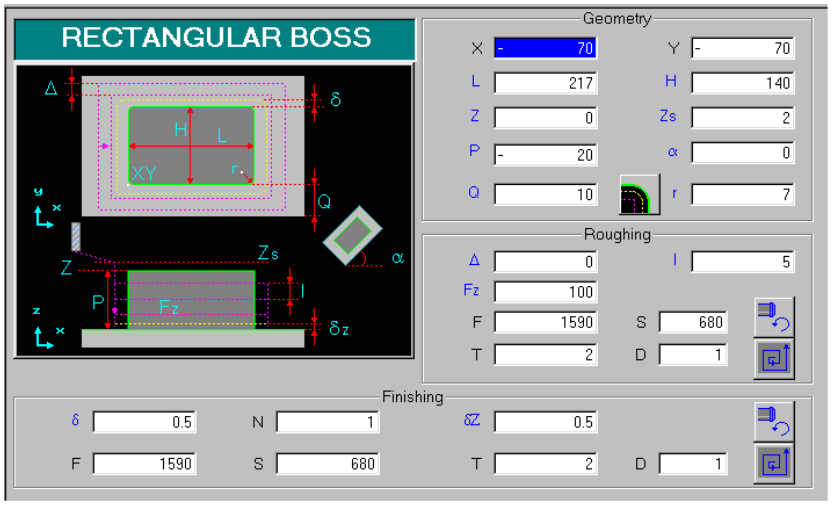
REMARKS

The surface milling may be executed in several ways.

- Unidirectional along X or along Y.
- Bidirectional along X or along Y.

These options are toggled using the (a) softkey.

RECTANGULAR BOSS CANNED CYCLE.



GEOMETRY		
X, Y	Coordinates of the stock's lower left corner	-70, -70
L	Total length in X.	217
H	Total length in Y.	140
Z	Height coordinate of the surface	0
Zs	Safety Z coordinate	2
P	Total depth	-20
a	Boss inclination angle	0
Q	Excess material	10
r	Rounding radius	7
	Corner rounding finish	

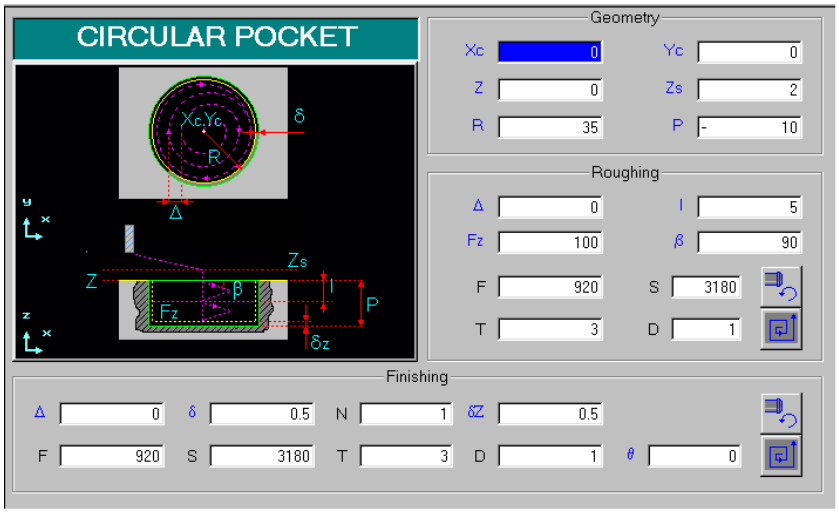
ROUGHING		
Δ	Roughing pass	0
I	Penetration step in Z	5
Fz	Penetrating feedrate at each pass (mm/min.)	100
F	Machining feedrate in mm/min.	1590
S	RPM	680
T	Number of the tool to be used	2
D	Tool offset	1
	Counterclockwise turning direction	
	Counterclockwise machining direction	

FINISHING		
δ	Finishing stock.	0.5
F	Finishing feedrate in mm/min.	1590
N	Number of finishing passes in Z	1
δz	Excess material in Z	0.5
S	RPM	680
T	Finishing tool number	2
D	Tool offset	1
	Counterclockwise turning direction	(b)
	Counterclockwise machining direction	(c)

REMARKS

This machining data is for using a hard metal endmill without covering and two teeth.

CIRCULAR POCKET CANNED CYCLE

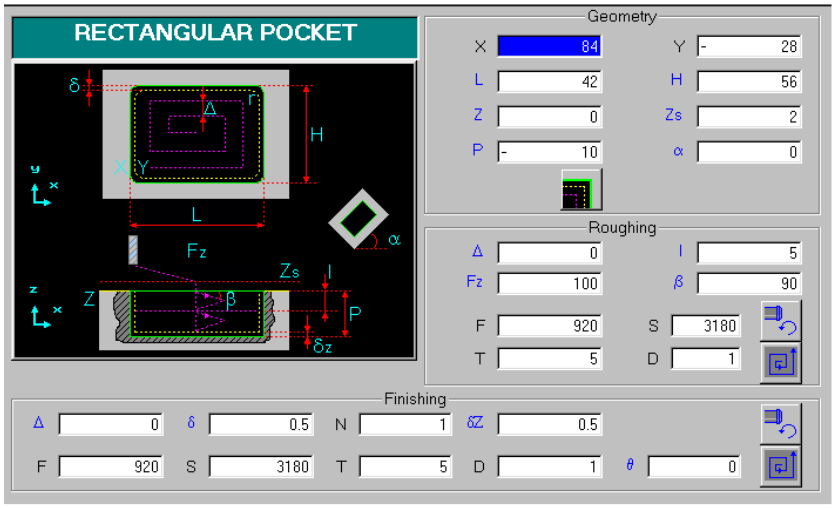


GEOMETRY		
Xc	Pocket center in X	0
Yc	Pocket center in Y	0
Z	Height coordinate of the surface	0
Zs	Safety Z coordinate	2
R	Pocket radius	35
P	Total pocket depth	-10

ROUGHING		
Δ	Roughing pass	0
I	Pass in Z	5
Fz	Penetrating feedrate at each pass in Z mm/min.	100
β	Penetrating angle. It permits several runs on Z	90
F	Roughing feedrate in mm/min.	920
S	RPM	3180
T	Finishing tool number	3
D	Tool offset	1
	Counterclockwise turning direction	
	Counterclockwise machining direction	

FINISHING		
Δ	Finishing pass	0
δ	Finishing stock.	0.5
F	Finishing feedrate in mm/min.	920
N	Number of finishing passes in Z	1
δz	Excess material in Z	0.5
S	RPM	3180
T	Finishing tool number	3
D	Tool offset	1
θ	Tool penetrating angle when finishing	0
	Counterclockwise turning direction	
	Counterclockwise machining direction	

RECTANGULAR POCKET CANNED CYCLE.

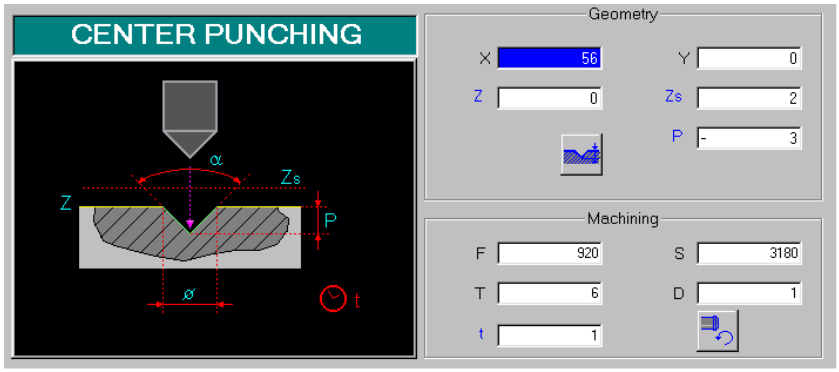


GEOMETRY		
X	Lower left corner in X	84
Y	Lower left corner in Y	-28
L	Length in X	42
H	Length in Y	56
Z	Surface Z coordinate	0
Zs	Safety Z coordinate	2
P	Total pocket depth	-10
a	pocket inclination angle	0
r	Corner blending radius	
	Corner finishing	

ROUGHING		
Δ	Roughing pass	0
I	Pass in Z	5
Fz	Penetrating feedrate at each pass in Z mm/min.	100
β	Penetrating angle.	90
F	Roughing feedrate in mm/min.	920
S	RPM	3180
T	Finishing tool number	5
D	Tool offset	1
	Counterclockwise turning direction	
	Counterclockwise machining direction	

FINISHING		
Δ	Finishing pass	0
δ	Finishing stock.	0.5
F	Finishing feedrate in mm/min.	920
N	Number of finishing passes in Z	1
δz	Excess material in Z	0.5
S	RPM	3180
T	Finishing tool number	5
D	Tool offset	1
θ	Tool penetrating angle when finishing	0
	Counterclockwise turning direction	
	Counterclockwise machining direction	

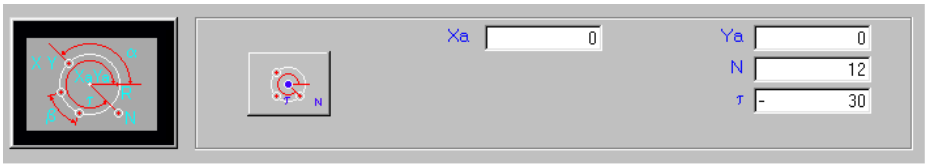
CENTER PUNCHING CANNED CYCLE

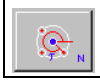


GEOMETRY		
X	X coordinate of the first center punch	56
Y	Y coordinate of the first center punch	0
Z	Surface Z coordinate	0
Zs	Safety Z coordinate	2
P	Total pocket depth	-3
	Center punching by depth	

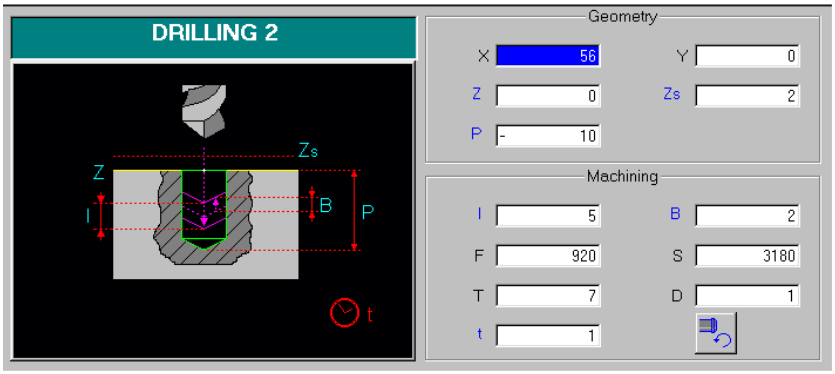
MACHINING		
F	Roughing feedrate in mm/min.	920
S	RPM	3180
T	Finishing tool number	6
D	Tool offset	1
t	Dwell at the bottom (sec.)	1
	Counterclockwise turning direction	

Once the form is filled out, instead of inserting the cycle in the program, you must locate the center punching positions using the Multiple softkey (F7) that shows the various types of positioning. Press the Arc softkey (F2) that shows the following form.



MULTIPLE POSITIONING		
Xa	Arc center in X	0
Ya	Arc center in Y	0
N	Number of positions	12
τ	Arc final angle	-30
	Method. Center coordinates, total number of center punches, angle of the final point	

DRILLING CANNED CYCLE

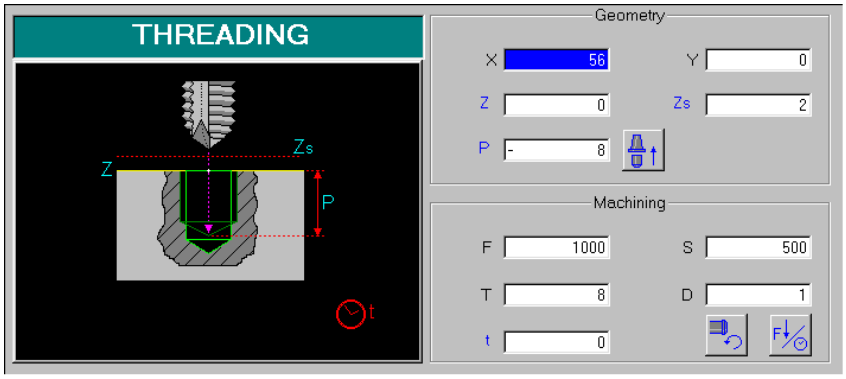


GEOMETRY		
X	X coordinate of the first hole	56
Y	Y coordinate of the first hole	0
Z	Surface Z coordinate	0
Zs	Safety Z coordinate	2
P	Total pocket depth	-10

MACHINING		
I	Penetration step	5
B	Chip relief distance after each pass I.	2
F	Feedrate in mm/min.	920
S	Spindle rpm	3180
T	Tool number	7
D	Tool offset number	1
t	Dwell at the bottom (seconds)	1
	Counterclockwise turning direction	

Once the form has been filled out, instead of inserting the cycle into the program, you must locate the drilling positions using the Multiple softkey (F7) that shows the various types of positioning as described for the previous form and make sure that the points coincide with the ones previously programmed in the center punching cycle.

TAPPING CANNED CYCLE



GEOMETRY		
X	X coordinate of the first hole	56
Y	Y coordinate of the first hole	0
Z	Surface Z coordinate	0
Zs	Safety Z coordinate	2
P	Total pocket depth	-8

MACHINING		
F	Feedrate in mm/min.	1000
S	Spindle rpm	500
T	Tool number	8
D	Tool offset number	1
t	Dwell at the bottom (seconds)	0
	Counterclockwise turning direction	

To enter the arc positioning, proceed as in the previous two cycles, press the Multiple softkey and then the [INS] key to accept the data (which will be the same as the ones used for center punching and drilling) and include them in the program.

The program is now complete and it is a good idea to create an ISO block to withdraw the tool and another one to indicate the "end of the program".

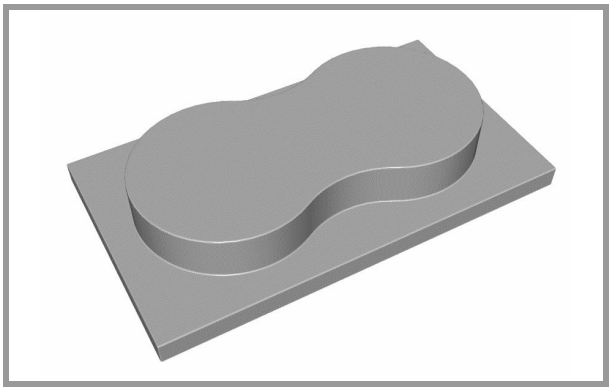
G0 Z200 Tool withdraw in rapid.

M30 End of program.

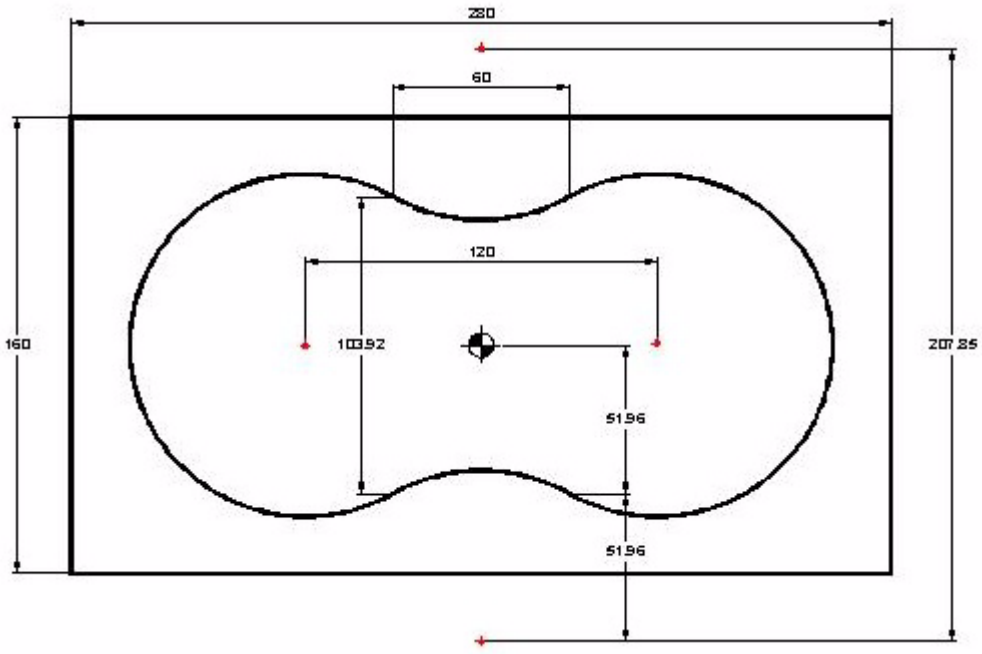
EXERCISE 2. 2D POCKET

The purpose of the following exercise is to make a cam from a stock whose dimensions are 280 x 160.

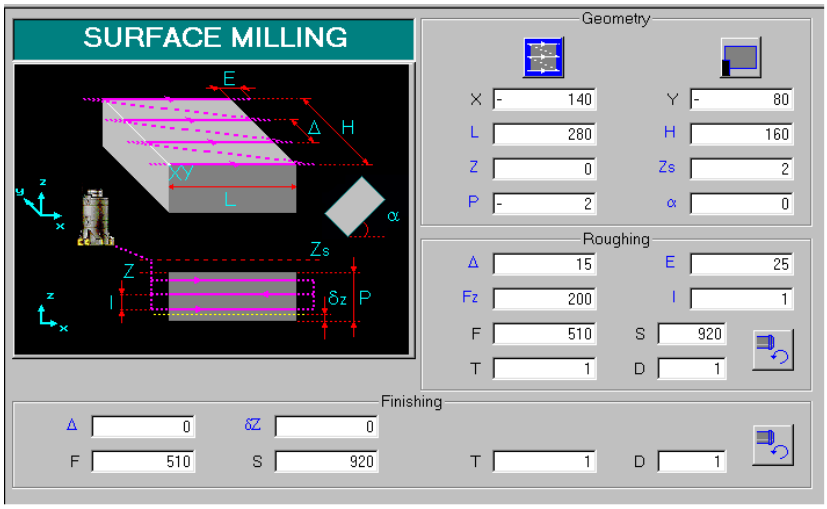
Making this part requires the following steps:



Operations	Tools
Surface milling cycle	Endmill Ø100 T1 D4
2D pocket	End mill Ø20
2D pocket. Profile editor	End mill Ø20



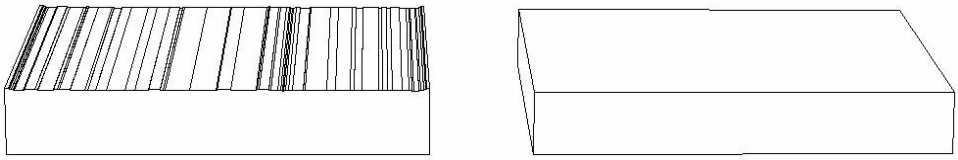
SURFACE MILLING CANNED CYCLE



GEOMETRY		
	Type of surface milling. Unidirectional surface milling along X	
	Corner where the surface milling begins. Lower left corner.	
X	X coordinate of the initial corner	-140
Y	Y coordinate of the initial corner	-80
L	Total length in X	280
H	Total length in Y	160
Z	Height coordinate of the surface	0
Zs	Safety Z coordinate	2
P	Total surface milling depth	-2
α	Surface milling angle	0

ROUGHING		
Δ	Roughing pass	15
E	Overshooting for chip relief	25
Fz	Penetrating feedrate at each pass (mm/min.)	200
I	Penetration step in Z	1
F	Machining feedrate in mm/min.	510
S	RPM	920
T	Number of the tool to be used	1
D	Tool offset	1
	Counterclockwise turning direction	

FINISHING		
Δ	Finishing pass	0
δz	Excess material in Z	0
F	Machining feedrate in mm/min.	510
S	RPM	920
T	Tool number	1
D	Tool offset	1
	Counterclockwise turning direction	



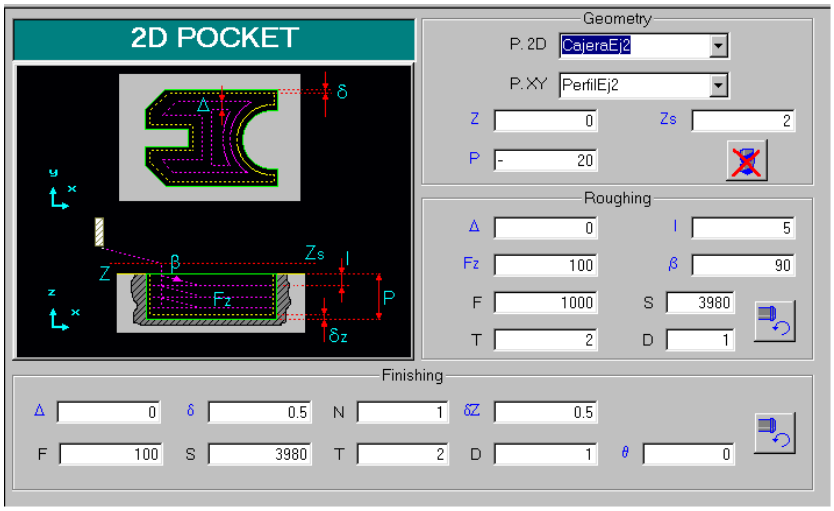
REMARKS

The surface milling may be executed in several ways.

- Unidirectional along X or along Y.
- Bidirectional along X or along Y.

These options are toggled using the (a) softkey and pressing ESCAPE.

2D POCKET CANNED CYCLE



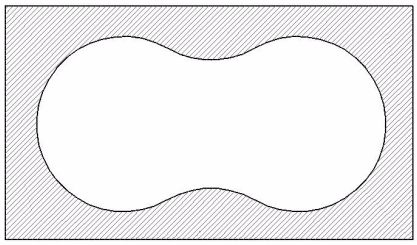
GEOMETRY		
P.2D	Name of the 2D pocket profile.	Pocket-1
P.XY	Name of the depth profile.	Profile-2
Z	Height coordinate of the surface.	0
Zs	Safety Z coordinate.	2
P	Total surface milling depth.	-20
	Without prior drilling.	

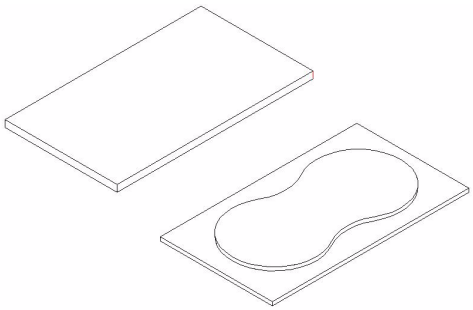
ROUGHING		
Δ	Roughing pass.	0
I	Penetration step in Z.	5
Fz	Penetrating feedrate at each pass (mm/min.)	100
β	Penetrating angle.	90
F	Machining feedrate in mm/min.	1000
S	RPM	3980
T	Number of the tool to be used.	2
D	Tool offset.	1
	Counterclockwise turning direction.	

ROUGHING		
Δ	Finishing pass.	0
δ	Finishing stock.	0.5
F	Finishing feedrate in mm/min.	100
N	Number of finishing passes in Z	1
δz	Excess material in Z	0.5
S	RPM	3980
T	Number of the tool to be used.	2
D	Tool offset.	1
θ	Tool penetrating angle when finishing	0
	Counterclockwise turning direction.	

REMARKS

Once the form has been filled out, before inserting it into the program that you just created with the [INS] key, press the [GENERATE] softkey so the cycle automatically appears in the ISO code.





2D POCKET CANNED CYCLE. PROFILE EDITOR

Once located in the relevant P.XY box to name the profile to be used, press the [RECALL] key to access the profile editor automatically.

Start by making a rectangle delimiting the outside contour of the figure that will request the first point of the geometry.

RECTANGLE

BEGINNING

X = -150	Y = -90	VALIDATE		
Lx = 300	Ly = 180	VALIDATE		

NEW PROFILE

BEGINNING

X = 30	Y = -51.96	VALIDATE		

COUNTERCLOCKWISE ARC

X = 30	Y = 51.96	Xc = 60	Yc = 0	VALIDATE

CLOCKWISE ARC

X = -30	Y = 51.96	Xc = 0	Yc=103.925	TANGENCY = YES

VALIDATE

COUNTERCLOCKWISE ARC

X = -30	Y = -51.96	Xc = -60	Yc = 0	VALIDATE

CLOCKWISE ARC

X = 30	Y = -51.96	Xc = 0	Yc = -103.925	VALIDATE

END

SAVE PROFILE

Once the profile is completed, finish it with the [END] key and then save it displaying a message that indicates that the pocket has been saved and returning to the screen that was displayed before getting into the profile editor.

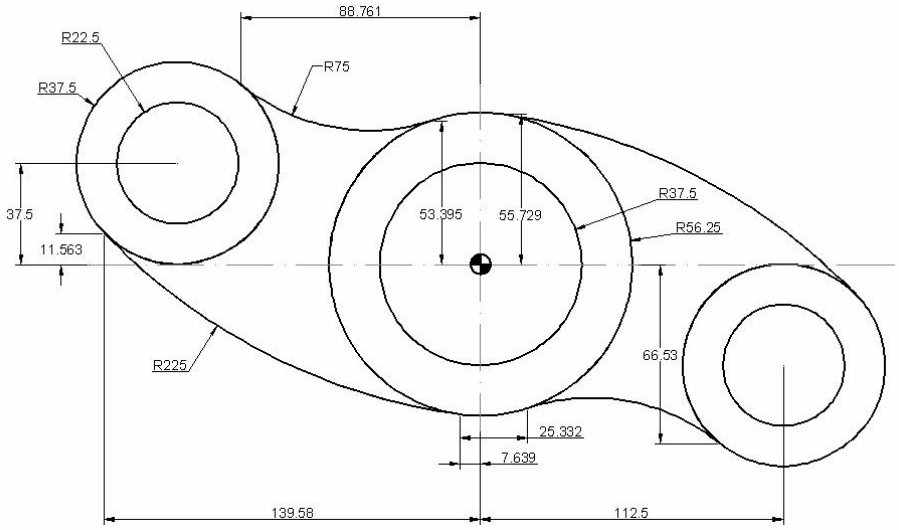
Once the whole 2D-pocket form has been defined, press the [GENERATE] softkey and [ENTER] and then [Insert 2D Pocket]

EXERCISE 3. CAM

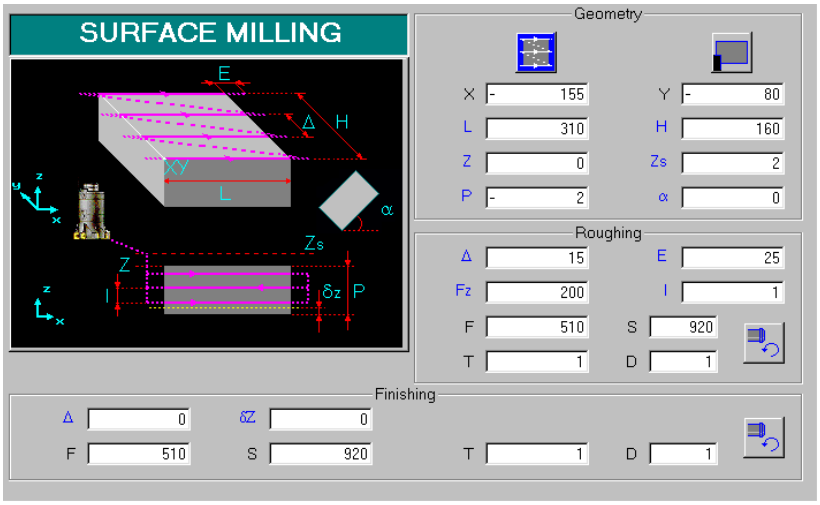
The purpose of the following exercise is to make a cam from a stock whose dimensions are 310 x 160.

Making this part requires the following steps:

Operations	Tools
Surface milling	Endmill Ø100 T1 D1
2D pocket	Endmill Ø20 T2 D1
Circular pocket	Endmill Ø10 T3 D1
2D pocket	Endmill Ø8 T4 D1
ISO coordinate rotation	



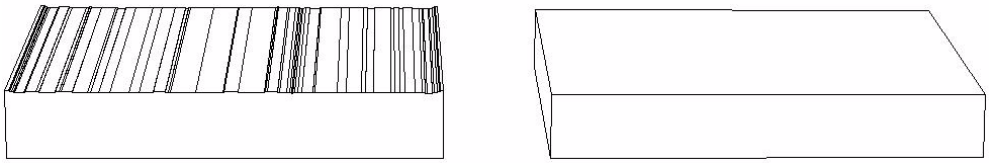
SURFACE MILLING CANNED CYCLE



GEOMETRY		
	Type of surface milling. Unidirectional surface milling along X	
	Corner where the surface milling begins. Lower left corner.	
X	X coordinate of the initial corner	-155
Y	Y coordinate of the initial corner	-80
L	Total length in X	310
H	Total length in Y	160
Z	Height coordinate of the surface	0
Zs	Safety Z coordinate	2
P	Total surface milling depth	-2
α	Surface milling angle	0

ROUGHING		
Δ	Roughing pass	15
E	Overshooting for chip relief	25
Fz	Penetrating feedrate at each pass (mm/min.)	200
I	Penetration step in Z	1
F	Machining feedrate in mm/min.	510
S	RPM	920
T	Number of the tool to be used	1
D	Tool offset	1
	Counterclockwise turning direction	

FINISHING		
Δ	Finishing pass	0
δz	Excess material in Z	0
F	Machining feedrate in mm/min.	510
S	RPM	920
T	Tool number	1
D	Tool offset	1
	Counterclockwise turning direction	



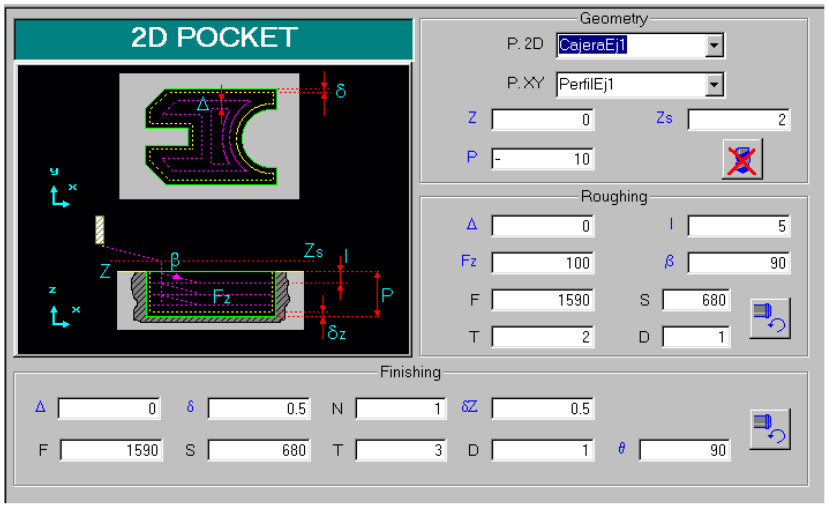
REMARKS

The surface milling may be executed in several ways.

- Unidirectional along X or along Y.
- Bidirectional along X or along Y.

These options are toggled using the (a) softkey.

2D POCKET CANNED CYCLE



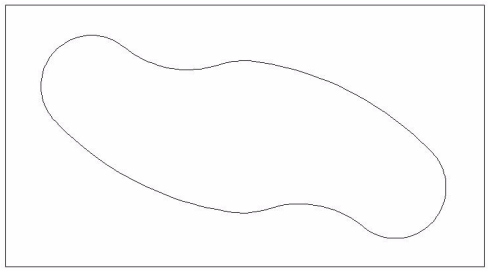
GEOMETRY		
P.2D	Name of the 2D pocket profile	Pocket-1
P.XY	Name of the depth-profile file.	Profile-1
Z	Height coordinate of the surface	0
Zs	Safety Z coordinate	2
P	Total surface milling depth	-10
	Without prior drilling	

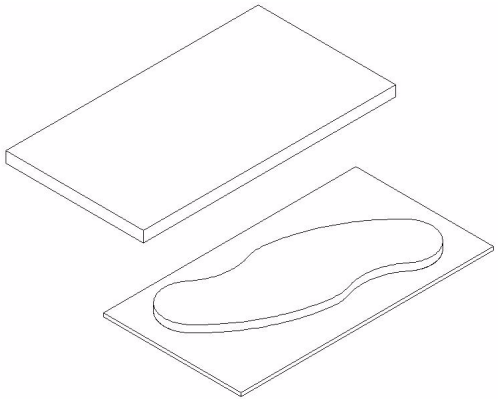
ROUGHING		
Δ	Roughing pass	0
I	Penetration step in Z	5
Fz	Penetrating feedrate at each pass (mm/min.)	100
β	Penetrating angle	90
F	Machining feedrate in mm/min.	1590
S	RPM	680
T	Number of the tool to be used	2
D	Tool offset	1
	Counterclockwise turning direction	

FINISHING		
Δ	Finishing pass	0
δ	Finishing stock.	0.5
F	Finishing feedrate in mm/min.	1590
N	Number of finishing passes in Z	1
δz	Excess material in Z	0.5
S	RPM	680
T	Finishing tool number	3
D	Tool offset	1
θ	Tool penetrating angle when finishing	90
	Counterclockwise turning direction	

REMARKS

This machining data is for using a hard metal endmill without covering and two teeth.





2D POCKET CANNED CYCLE. PROFILE EDITOR

Once located in the relevant P.XY box to name the profile to be used, press the [RECALL] key to access the profile editor automatically.

Start by making a rectangle delimiting the outside contour of the figure that will request the first point of the geometry.

BEGINNING

RECTANGLE

X = -165 Y = -90 VALIDATE

Lx = 330 Ly = 180 VALIDATE

NEW PROFILE

BEGINNING

X = -139.58 Y = 11,563 VALIDATE

CLOCKWISE ARC

X = -88.761 Y = 66.53 Xc = -112.5 Yc = 37.5 R = 37.5 VALIDATE

COUNTERCLOCKWISE ARC

X = -17.693 Y = 53.395 R = 75 TANGENCY = YES VALIDATE

CLOCKWISE ARC

X = 7.639 Y = 55.729 Xc = 0 Yc = 0 R=56.25
TANGENCY = YES VALIDATE

CLOCKWISE ARC

X = 139.58 Y = -11.563 R = 225 TANGENCY=YES VALIDATE

CLOCKWISE ARC

X = 88.761 Y = -66.53 Xc = 112.5 Yc = -37.5 R = 37.5 VALIDATE

COUNTERCLOCKWISE ARC

X = 17.693 Y = -53.395 R = 75 VALIDATE

CLOCKWISE ARC

X = -7.639 Y = -55.729 Xc = 0 Yc = 0 R=56.25
TANGENCY = YES VALIDATE

CLOCKWISE ARC

X = -139.58 Y = 11.563 R = 225 TANGENCY=YES VALIDATE

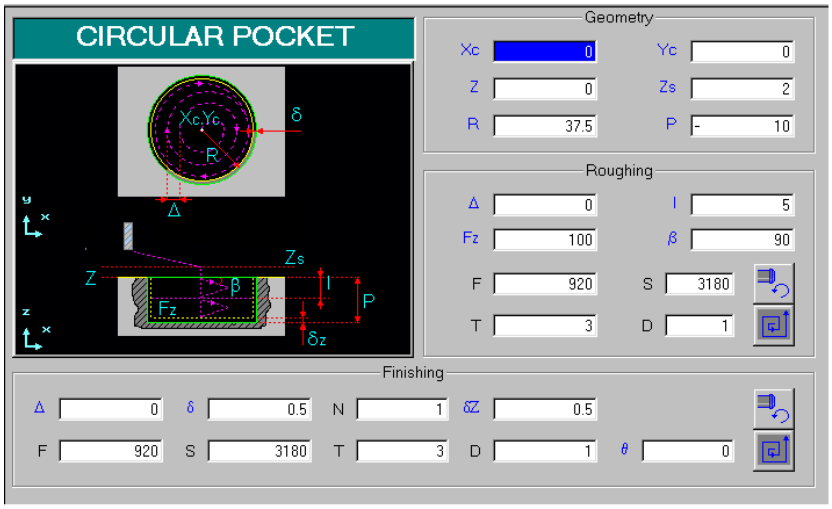
END

SAVE PROFILE

Once the profile is completed, finish it with the [END] key and then save it displaying a message that indicates that the pocket has been saved and returning to the screen that was displayed before getting into the profile editor.

Once the whole 2D-pocket form has been defined, press the [GENERATE] softkey and [ENTER] and then [Insert 2D Pocket]

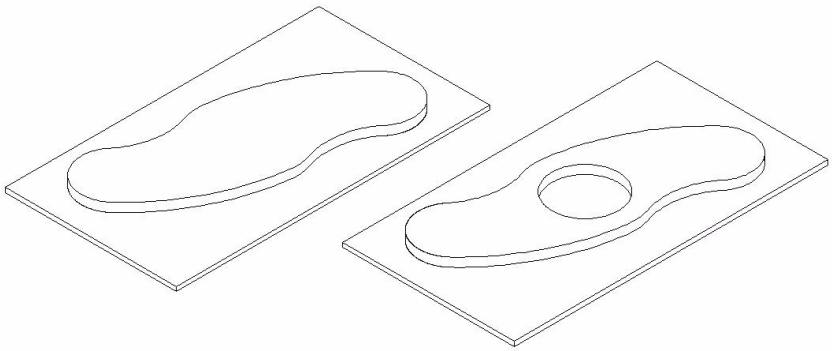
CIRCULAR POCKET CANNED CYCLE



GEOMETRY		
Xc	Pocket center in X	0
Yc	Pocket center in Y	0
Z	Height coordinate of the surface	0
Zs	Safety Z coordinate	2
R	Pocket radius	37.5
P	Total pocket depth	-10

ROUGHING		
Δ	Roughing pass	0
I	Pass in Z	5
Fz	Penetrating feedrate at each pass in Z mm/min.	100
β	Penetrating angle. It permits several runs on Z	90
F	Roughing feedrate in mm/min.	920
S	RPM	3180
T	Finishing tool number	3
D	Tool offset	1
	Counterclockwise turning direction	
	Counterclockwise machining direction	

FINISHING		
Δ	Finishing pass	0
δ	Finishing stock.	0.5
F	Finishing feedrate in mm/min.	920
N	Number of finishing passes in Z	1
δz	Excess material in Z	0.5
S	RPM	3180
T	Finishing tool number	3
D	Tool offset	1
θ	Tool penetrating angle when finishing	0
	Counterclockwise turning direction	
	Counterclockwise machining direction	



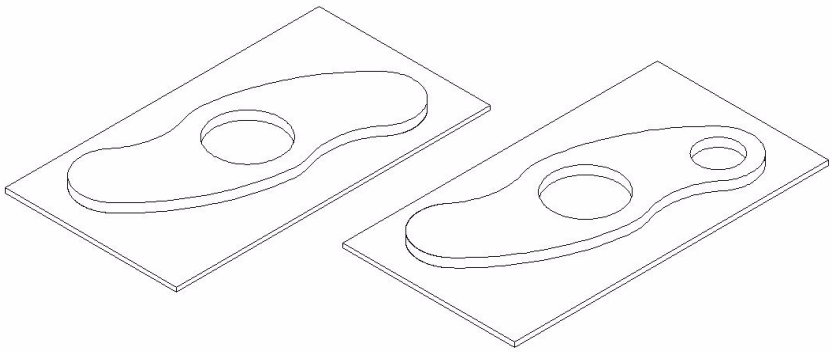
Then, repeat the circular pocket cycle twice to drill the side holes of the cam.

CIRCULAR POCKET CANNED CYCLE (2)

GEOMETRY		
Xc	Pocket center in X	112.5
Yc	Pocket center in Y	-37.5
Z	Height coordinate of the surface	0
Zs	Safety Z coordinate	2
R	Pocket radius	22.5
P	Total pocket depth	-10

ROUGHING		
Δ	Roughing pass	0
I	Pass in Z	5
Fz	Penetrating feedrate at each pass in Z mm/min.	100
β	Penetrating angle. It permits several runs on Z	90
F	Roughing feedrate in mm/min.	1000
S	RPM	3980
T	Finishing tool number	4
D	Tool offset	1
	Counterclockwise turning direction	
	Counterclockwise machining direction	

FINISHING		
Δ	Finishing pass	0
δ	Finishing stock.	0.5
F	Finishing feedrate in mm/min.	1000
N	Number of finishing passes in Z	1
δz	Excess material in Z	0.5
S	RPM	3980
T	Finishing tool number	4
D	Tool offset	1
θ	Tool penetrating angle when finishing	0
	Counterclockwise turning direction	
	Counterclockwise machining direction	



CIRCULAR POCKET CANNED CYCLE (3)

GEOMETRY		
Xc	Pocket center in X	-112.5
Yc	Pocket center in Y	37.5
Z	Height coordinate of the surface	0
Zs	Safety Z coordinate	2
R	Pocket radius	22.5
P	Total pocket depth	-10

ROUGHING		
Δ	Roughing pass	0
I	Pass in Z	5
Fz	Penetrating feedrate at each pass in Z mm/min.	100
β	Penetrating angle. It permits several runs on Z	90
F	Roughing feedrate in mm/min.	1000
S	RPM	3980
T	Finishing tool number	4
D	Tool offset	1
	Counterclockwise turning direction	
	Counterclockwise machining direction	

FINISHING		
Δ	Finishing pass	0
δ	Finishing stock.	0.5
F	Finishing feedrate in mm/min.	1000
N	Number of finishing passes in Z	1
δz	Excess material in Z	0.5
S	RPM	3980
T	Finishing tool number	4
D	Tool offset	1
θ	Tool penetrating angle when finishing	0
	Counterclockwise turning direction	
	Counterclockwise machining direction	

Then, use the same cycle as in section 5.2 to remove the excess material in the middle of the part.

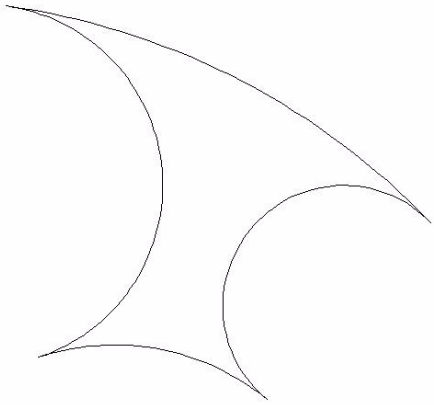
GEOMETRY		
P.2D	Name of the 2D pocket profile	Pocket-2
P.XY	Name of the depth profile	Profile-2
Z	Height coordinate of the surface	0
Zs	Safety Z coordinate	2
P	Total surface milling depth	5
	Without prior drilling	

ROUGHING		
Δ	Roughing pass	0
I	Penetration step in Z	5
Fz	Penetrating feedrate at each pass (mm/min.)	100
β	Penetrating angle	90
F	Machining feedrate in mm/min.	1000
S	RPM	3980
T	Number of the tool to be used	4
D	Tool offset	1
	Counterclockwise turning direction	

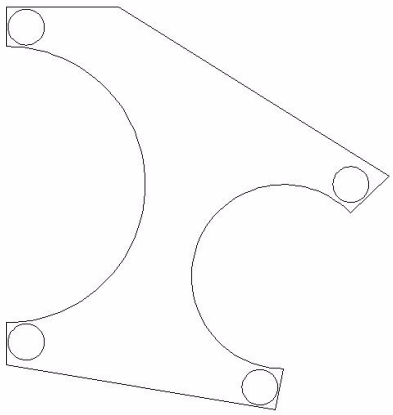
FINISHING		
Δ	Finishing pass	0
δ	Finishing stock.	0.5
F	Finishing feedrate in mm/min.	1000
N	Number of finishing passes in Z	1
δz	Excess material in Z	0.5
S	RPM	3980
T	Finishing tool number	4
D	Tool offset	1
θ	Tool penetrating angle when finishing	90
	Counterclockwise turning direction	

At the profile editor, draw the geometry to be emptied. Before beginning to define it, bear in mind that you're going to machine; therefore, you will have to use a tool with a certain diameter. Pay attention to the geometry of the drawing so the tool can work inside it since it is an emptying operation.

- This is the theoretically drawn geometry; but, as may be seen in the drawing, the tool could never machine the programmed corners because it does not fit. Therefore, the tool must be given some relief in those four corners.



- For example, this would be the best way to program this profile.



NEW PROFILE

BEGINNING

X = 7,639 Y = 65 VALIDATE

STRAIGHT

X = 55 Y = 65 VALIDATE

STRAIGHT

X = 139.58 Y = 0 VALIDATE

STRAIGHT

X = 139.58 Y = -11,563 VALIDATE

COUNTERCLOCKWISE ARC

X = 88.761 Y = -66.53 Xc = 112.5 Yc = -37.5R = 37.5
VALIDATE

STRAIGHT

X = 90 Y = -75 VALIDATE

STRAIGHT

X = 17,693 Y = -65 VALIDATE

STRAIGHT

X = 17,693 Y = -53,395 VALIDATE

COUNTERCLOCKWISE ARC

X = 7.639 Y = 55.729 Xc = 0 Yc = 0 R=56.25 VALIDATE

STRAIGHT

X = 7,639 Y = 65 VALIDATE

END

SAVE PROFILE

ISO COORDINATE ROTATION

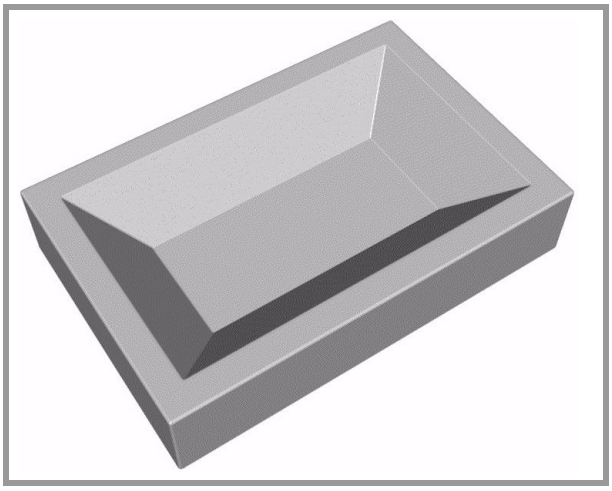
Once the form has been inserted into the program, program an ISO command to rotate the coordinate system in order to repeat the machining operation on the other side. The ISO block will be:

```
N1: (First label)
#POCKET2D BEGIN (1-3C2D)          Canned cycle.
--------------------
--------------------
--------------------
N2: (Second label)
G73 Q180                          Coordinate rotation.
#RPT [N1,N2]                      Block repetition.
G0 Z100
M30
```

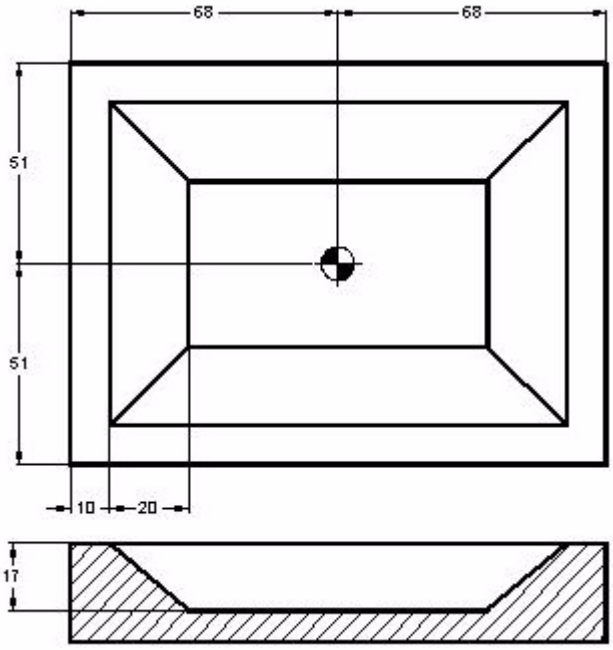
EXERCISE 4. 3D POCKETS

The purpose of the following exercise is to make a cam from a stock whose dimensions are 136 x 102.

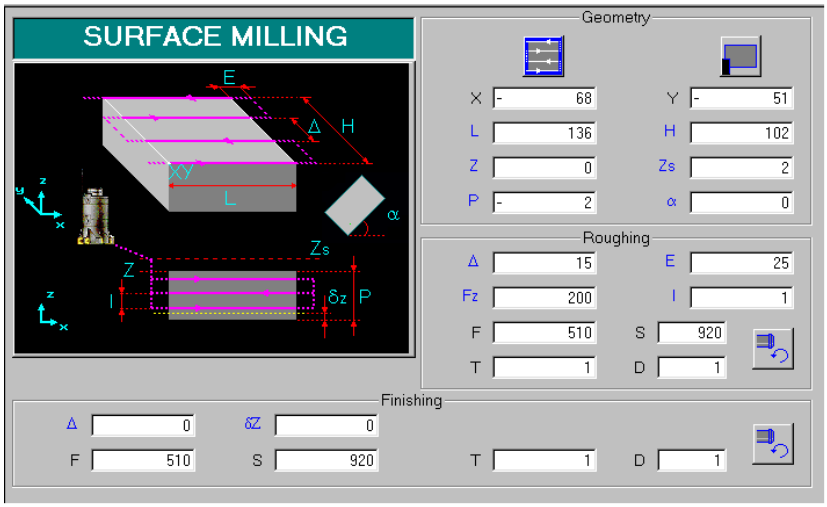
Making this part requires the following steps:



Operations	Tools
Surface milling	Endmill Ø100 T1 D1
3D pocket	Endmill Ø20 T2 D2



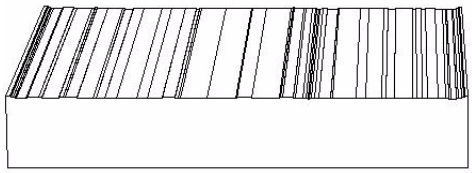
SURFACE MILLING CANNED CYCLE

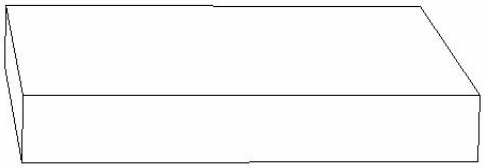


GEOMETRY		
	Type of surface milling. Unidirectional surface milling along X	
	Corner where the surface milling begins. Lower left corner.	
X	X coordinate of the initial corner	-68
Y	Y coordinate of the initial corner	-51
L	Total length in X	136
H	Total length in Y	102
Z	Height coordinate of the surface	0
Zs	Safety Z coordinate	2
P	Total surface milling depth	-2
α	Surface milling angle	0

ROUGHING		
Δ	Roughing pass	15
E	Overshooting for chip relief	25
Fz	Penetrating feedrate at each pass (mm/min.)	200
I	Penetration step in Z	1
F	Machining feedrate in mm/min.	510
S	RPM	920
T	Number of the tool to be used	1
D	Tool offset	1
	Counterclockwise turning direction	

FINISHING		
Δ	Finishing pass	0
δz	Excess material in Z	0
F	Machining feedrate in mm/min.	510
S	RPM	920
T	Tool number	1
D	Tool offset	1
	Counterclockwise turning direction	





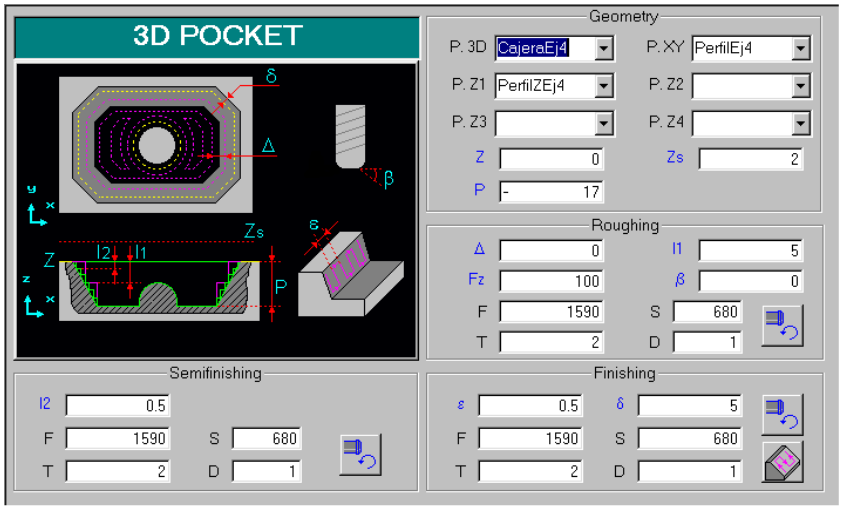
REMARKS

The surface milling may be executed in several ways.

- Unidirectional along X or along Y.
- Bidirectional along X or along Y.

These options are toggled using the (a) softkey and pressing ESCAPE.

3D POCKET CANNED CYCLE.



GEOMETRY		
P.3D	Name of the 3D pocket	Pocket3D-1
P.XY	Name of the profile file. *	Profile-2
P.Z1	Depth profile. **	Profile-3
P.Z2...	Depth profiles.	
Z	Height coordinate of the surface.	0
Zs	Safety Z coordinate	2
P	Profile depth.	-17

ROUGHING		
Δ	Roughing pass	0
I1	Maximum penetration step.	5
Fz	Penetrating feedrate at each pass (mm/min.)	100
β	Side penetrating angle	0
F	Finishing feedrate in mm/min.	1590
S	RPM	680
T	Number of the tool to be used	2
D	Tool offset	1
	Counterclockwise turning direction	

SEMIFINISHING		
I2	Maximum penetration step	0.5
F	Finishing feedrate in mm/min.	1590
S	Spindle rpm for roughing	680
T	Roughing tool number	2
D	Offset for that tool	1
	Counterclockwise turning direction	

FINISHING		
e	Roughing pass	0.5
d	Maximum penetration step.	5
F	Finishing feedrate in mm/min.	1590
S	RPM	680
T	Number of the tool to be used	2
D	Tool offset	1
	Counterclockwise turning direction	
	Zig Zag	

* Pressing the [RECALL] in this box, the CNC shows the Profile Editor screen where the geometry will have to be drawn. Once the profile is completed, the screen will return to the 3D pocket form and the profile must be confirmed by pressing [ENTER]. The following section describes how to use the profile editor.

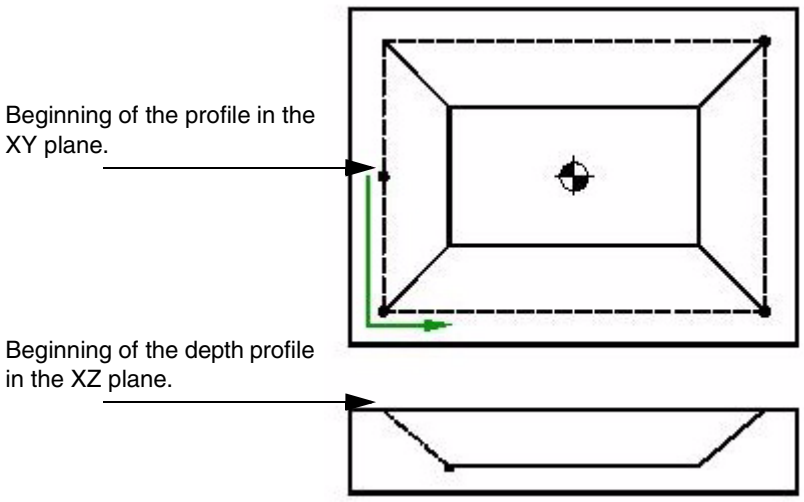
** P.Z2, Z3, Z4. Depth profiles allowed, if there is only one or two, the rest will be left blank.

REMARKS

This machining data is for using a hard metal endmill without covering and two teeth.

3D POCKET CANNED CYCLE. PROFILE EDITOR

First make the rectangle defining the outside contour of the figure to be made. Use the "line" command to set the first coordinate of the profile in the center of the figure.



BEGINNING

X=-58	Y=0	VALIDATE

STRAIGHT

X = -58	Y = -41	VALIDATE
X=58	Y=-41	VALIDATE
X=58	Y=41	VALIDATE
X=-58	Y=41	VALIDATE
X=-58	Y=0	VALIDATE

END

SAVE PROFILE

Once you programmed this profile, the external contour of the geometry would already be defined. Then, use the P.Z1 box to do the pocket's depth profile.

BEGINNING

X=-58	Z=0	VALIDATE

STRAIGHT

X=-38	Z=-17	VALIDATE

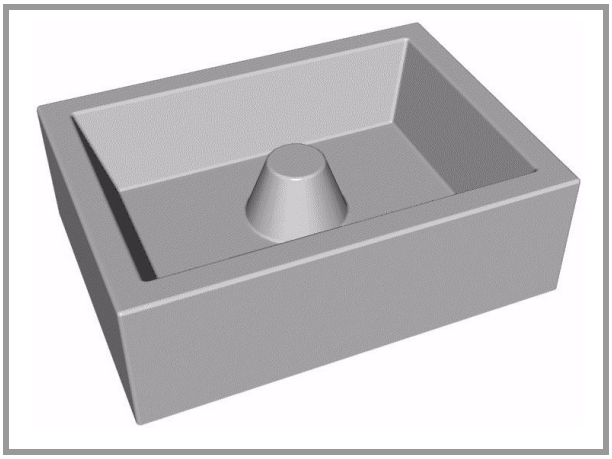
END

SAVE PROFILE

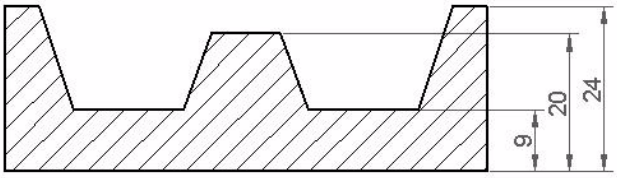
EXERCISE 5. 3D POCKETS WITH ISLANDS

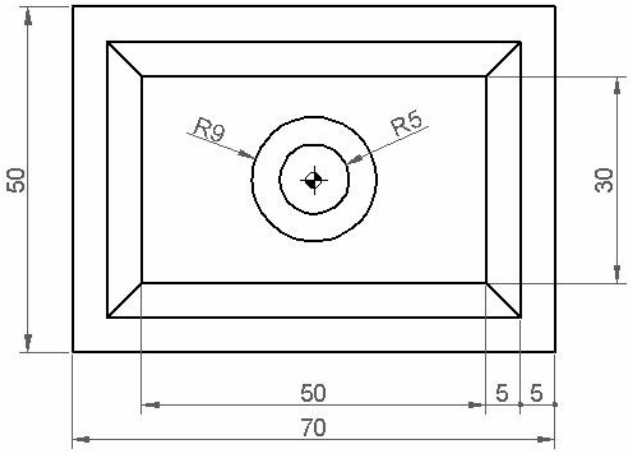
The purpose of the following exercise is to make a cam from a stock whose dimensions are 50 x 70.

Making this part requires the following steps:

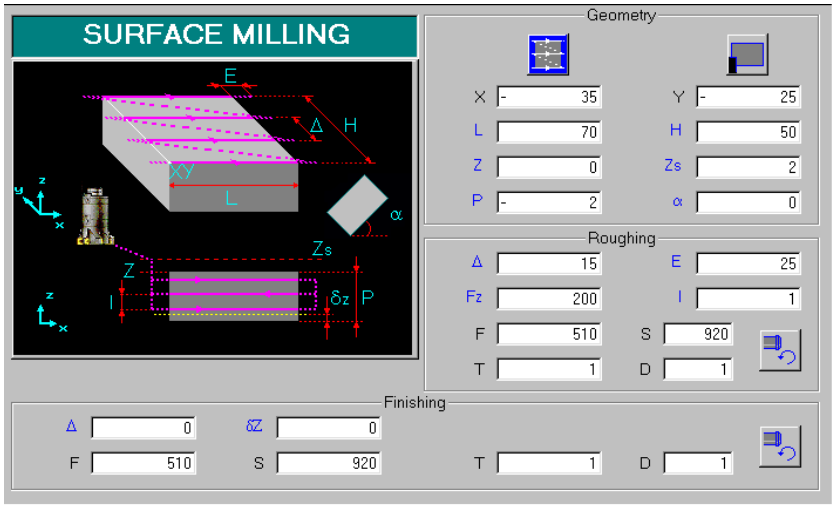


Operations	Tools
Surface milling	Endmill Ø100 T1 D1
3D pocket	Endmill Ø5 T9 D1





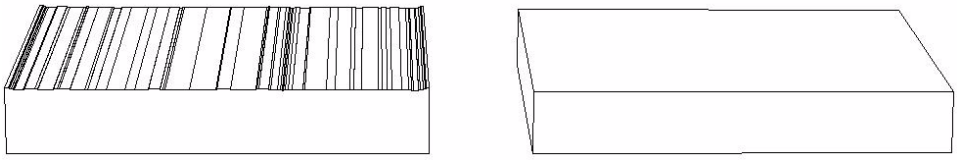
SURFACE MILLING CANNED CYCLE



GEOMETRY		
	Type of surface milling. Unidirectional surface milling along X	
	Corner where the surface milling begins. Lower left corner.	
X	X coordinate of the initial corner	-35
Y	Y coordinate of the initial corner	-25
L	Total length in X	70
H	Total length in Y	50
Z	Height coordinate of the surface	0
Zs	Safety Z coordinate	2
P	Total surface milling depth	-2
α	Surface milling angle	0

ROUGHING		
Δ	Roughing pass	15
E	Overshooting for chip relief	25
Fz	Penetrating feedrate at each pass (mm/min.)	200
I	Penetration step in Z	1
F	Machining feedrate in mm/min.	510
S	RPM	920
T	Number of the tool to be used	1
D	Tool offset	1
	Counterclockwise turning direction	

FINISHING		
Δ	Finishing pass	0
δz	Excess material in Z	0
F	Machining feedrate in mm/min.	510
S	RPM	920
T	Tool number	1
D	Tool offset	1
	Counterclockwise turning direction	



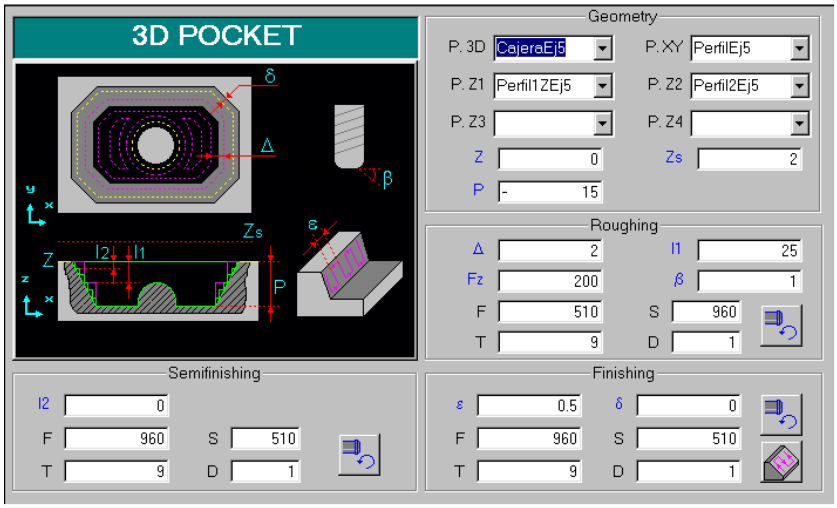
REMARKS

The surface milling may be executed in several ways.

- Unidirectional along X or along Y.
- Bidirectional along X or along Y.

These options are toggled using the (a) softkey and pressing ESCAPE.

3D POCKET CANNED CYCLE.



GEOMETRY		
P.3D	Name of the 3D pocket.	Pocket3D.1
P.XY	Name of the profile file. *	p-1
P.Z1	Depth profile **	p-2
P.Z2	Depth profile ***	p-3
Z	Height coordinate of the surface	0
Zs	Z depth coordinate	2
P	Profile depth	-15

ROUGHING		
Δ	Roughing pass	2
I1	Maximum penetration step	25
Fz	Penetrating feedrate at each pass (mm/min.)	200
β	Side penetrating angle	1
F	Machining feedrate in mm/min.	510
S	RPM	960
T	Number of the tool to be used	9
D	Tool offset	1

SEMIFINISHING		
I2	Maximum penetration step	0
F	Finishing feedrate	960
S	RPM	510
T	Finishing tool number	9
D	Offset for that tool	1

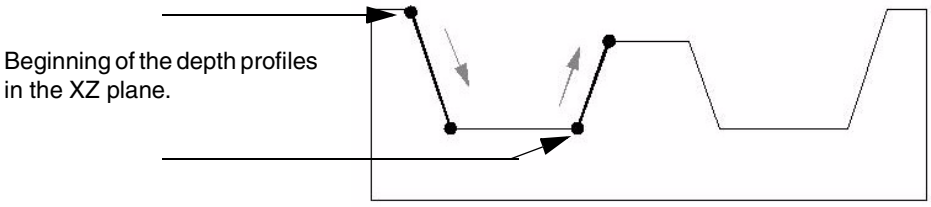
FINISHING		
ε	Finishing pass	0.5
δ	Finishing stock	0
F	Finishing feedrate	960
S	RPM	510
T	Finishing tool number	9
D	Offset for that tool	1

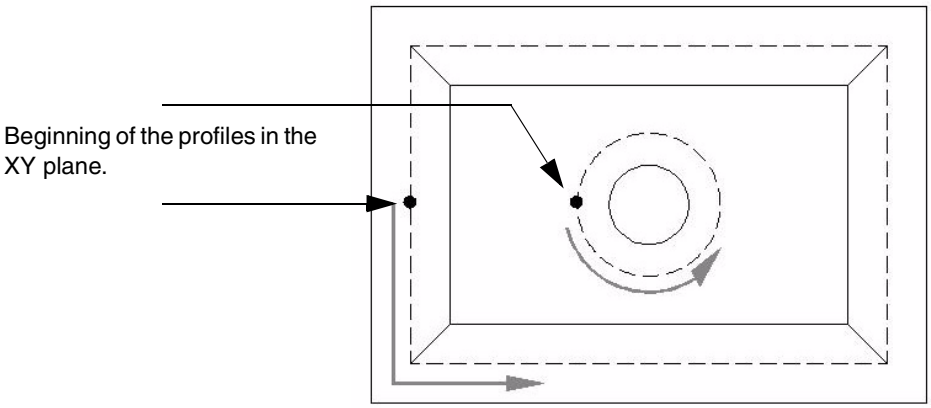
* Pressing the [RECALL] in this box, the CNC shows the Profile Editor screen where the geometry will have to be drawn. Once the profile is completed, the screen will return to the 3D pocket form and the profile must be confirmed by pressing [ENTER]. The following section describes how to use the profile editor

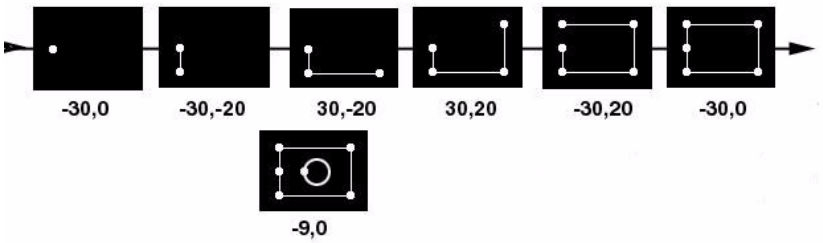
** P.Z2,Z3,Z4. Depth profiles allowed, if there is only one or two, the rest will be left blank.

3D POCKET CANNED CYCLE. PROFILE EDITOR

Start by making the rectangle delimiting the external contour of the figure. Due to the importance of the profile initial point, use the LINE command instead of the rectangle to do the figure.







BEGINNING

X=-30	Y=0	VALIDATE

STRAIGHT

X = -30	Y = -20	VALIDATE
X=30	Y=-20	VALIDATE
X=30	Y=20	VALIDATE
X=-30	Y=20	VALIDATE
X=-30	Y=0	VALIDATE

NEW PROFILE

CIRCLE

X=-9	Y=0	Xc=0	Yc=0	R=9	VALIDATE

END

SAVE PROFILE

With what has been previously programmed, the outside contour of the geometry and the island would already be defined.

Then, use the P.Z1 and P.Z2 boxes to do the pocket's depth profile.

P.Z1

BEGINNING

X=-30 Z=0 VALIDATE

STRAIGHT

X=-25 Z=-15 VALIDATE

END

SAVE PROFILE

P.Z2

BEGINNING

X=-9 Z=-15 VALIDATE

STRAIGHT

X=-5 Z=-4 VALIDATE

END

SAVE PROFILE

ISO PROGRAMMING

3

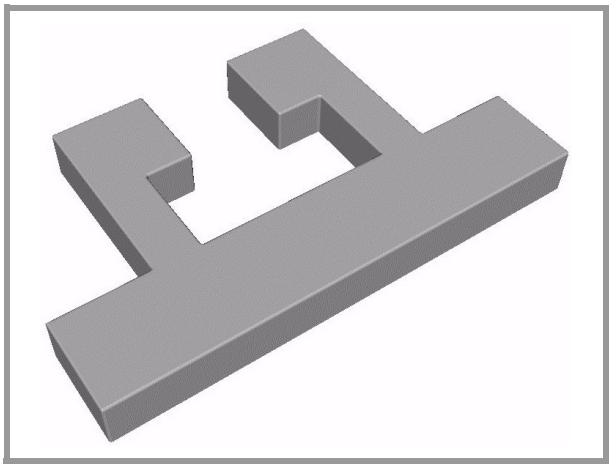
In general, ISO programming consists in entering a number of blocks that properly ordered make up a machining program. Basically, the programs are divided into three parts:

1. Header.
2. Geometry.
3. End.

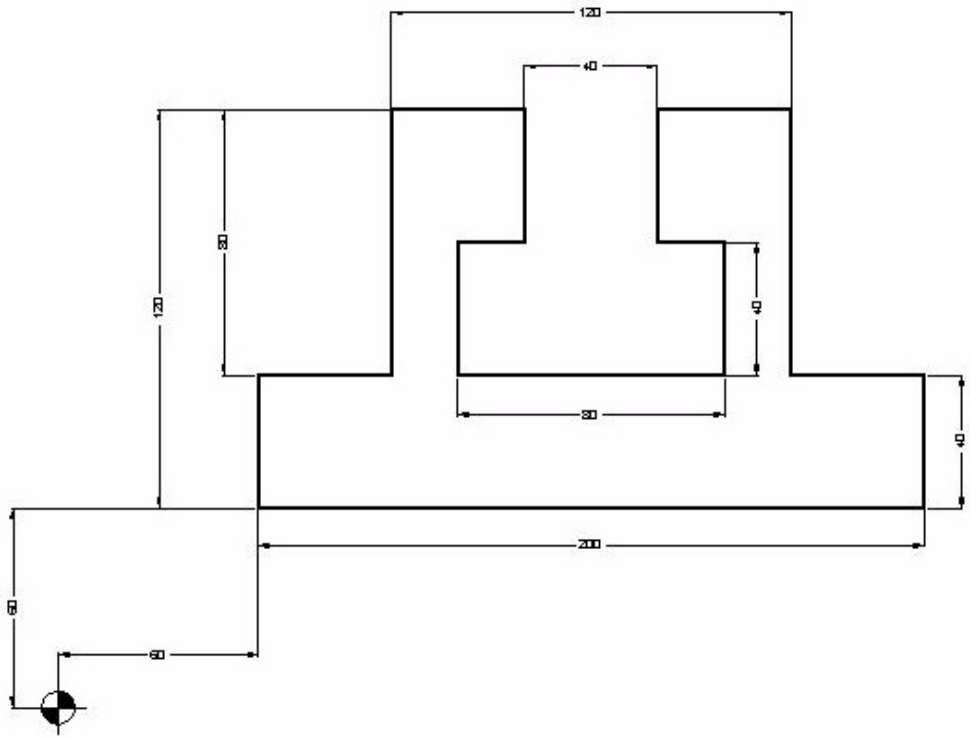
The machining conditions (feedrate and rpm), the tool and the material are the same for all the following examples.

EXERCISE 1

External contouring (climb cutting) with tangential input and a total depth of 20 mm with 5mm passes.



Operations	Tools
Contouring	Endmill Ø15 T10 D1



HEADER

G0 Z100	Safety positioning.
T10D1	Calling a tool and a tool offset.
	Tool change
S1000 M3	
X30 Y30	
Z0	
N1:	Positioning of label Nr 1
G91 G1 Z-5 F100	First pass in Z.
G90 G42 X60 Y60 F1000	Tangential entry with tool compensation.
G37 I10	

GEOMETRY

X260
Y100
X220
Y180
X180
Y140
X200
Y100
X120
Y140
X140
Y180
X100
Y100
X60
Y60
G38 I10
G40 X30 Y30

END

N2	Positioning of label Nr 2.
#RPT[N1,N2,3]	Repetitions.
G0 Z100	
M30	Return to the safety position and end of program.

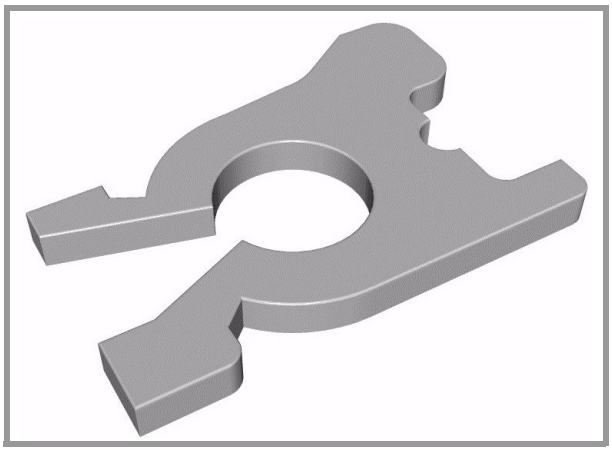
EXERCISE 2

In this exercise, you'll do a contour by entering polar coordinates because the data specifying the necessary points in X and Y is missing.

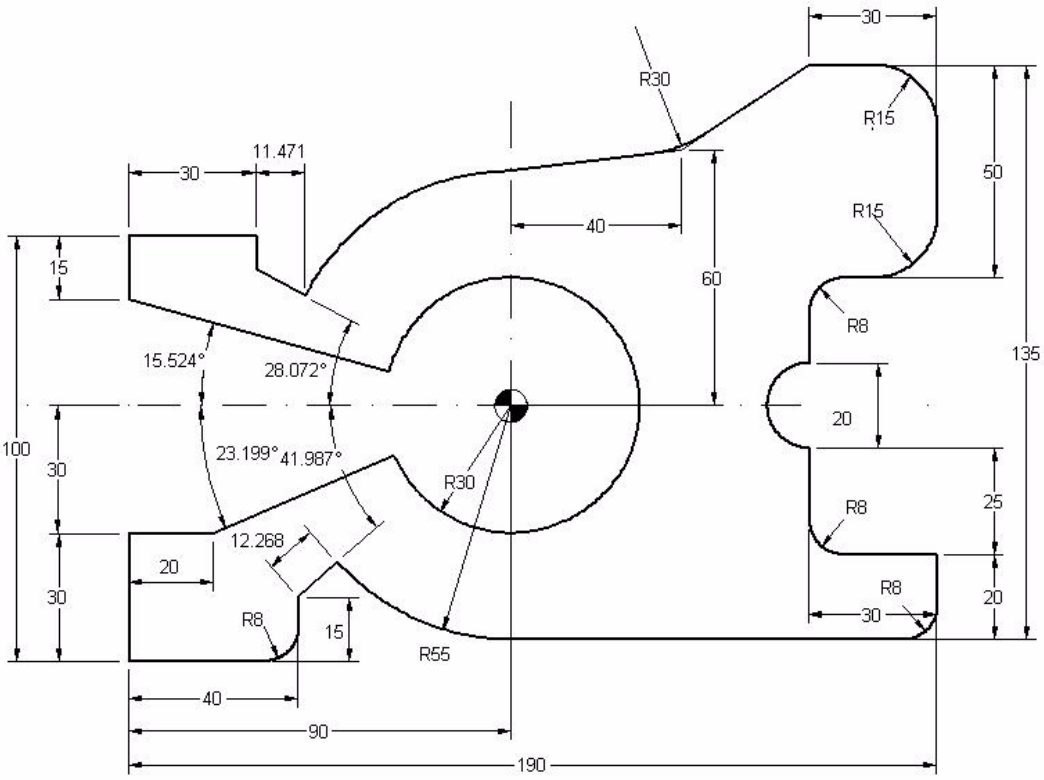
Programming a polar coordinate requires defining a center, a radius and angle (to do a straight line) or by just having an angle (arcs). This center is called Polar Center and is defined using function G30.

In this exercise, you will do an external contour of the geometry obtaining a total depth of 12 mm. Bear in mind that this geometry contains inside rounding with a radius of 8 mm and a tool with a larger diameter cannot be used.

Making this part requires the following steps:



Operations	Tools
Contouring	Endmill Ø8 T4 D1



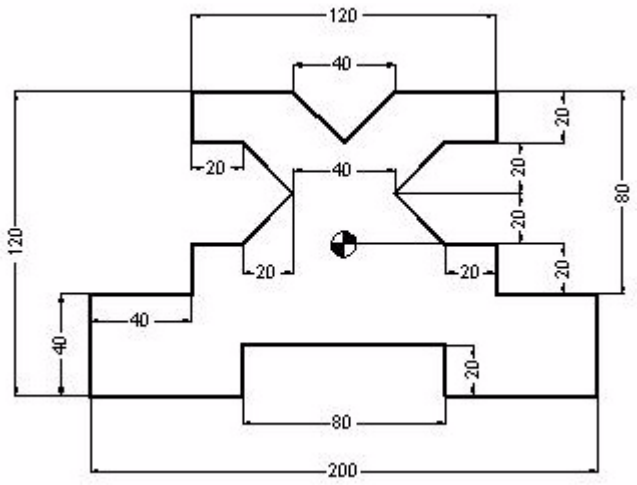
DOING THE EXERCISE

```
G0 Z100                    Safety positioning.
T4D1
S1000 M3
X-30 Y-30
Z0
N1:                        Positioning of label Nr 1.
G91 G1 Z-2 F100
G90 G42 X0 Y0 F1000
G37 I10
X40
G36 I8
G1 Y15
G30 I90 J60
G1 R55 Q221.987 F1000
G3 Q270
G1 X190
G36 I8
G91 Y20
X-30
G36 I8
Y25
G2 X0 Y20 R10
G90 G1 Y90
G36 I8
X190
G36 I15
Y140
G36 I15
G91 X-30
G90 G1 X130 Y120
G36 I30
X90 Y115
G3 Q151.958
G1 R67.268 Q151.928
Y100
X0
Y85
G1 R30 Q164.476
G2 Q203.199
G1 X20 Y30
X0
Y0
G38 I10
X-30 Y-30
N2:                        Positioning of label Nr 2.
#RPT [N1,N2,5]             Repetitions.
M30
```

Remarks

Exercise done in ISO code, using Polar coordinates for linear moves (G30 I J, G1 R Q) as well as for arcs (G30 I J, G2/3 Q).

EXERCISE 3



HEADER

G0 Z100
T4D1
M6
S1000 M3
X-130 Y-90
Z0
N1:
G1 G91 Z-5 F120
G90 G42 X-100 Y-60 F1000

GEOMETRY

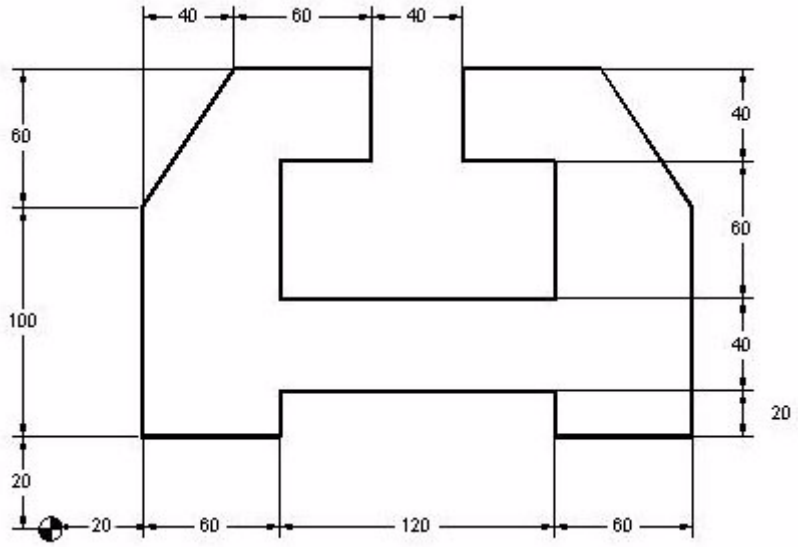
G37 I10
X-40
Y-40
X40
Y-60
X100
Y-20
X60
Y0
X40
X20 Y20
X40 Y40
X60
Y60
X20
X0 Y40
X-20 Y60
X-60
Y40
X-40

X-20 Y20
X-40 Y0
X-60
Y-20
X-100
Y-60

END

G38 I10
G40 X-130 Y-90
N2:
#RPT [N1,N2,4]
G0 Z100
M30

EXERCISE 4



HEADER

G0 Z100
T4D1
M6
S1000 M3
X-10 Y-10
Z0
N1:
G1 G91 Z-5 F150
G90 G42 X20 Y20 F1000
G37 I10

GEOMETRY

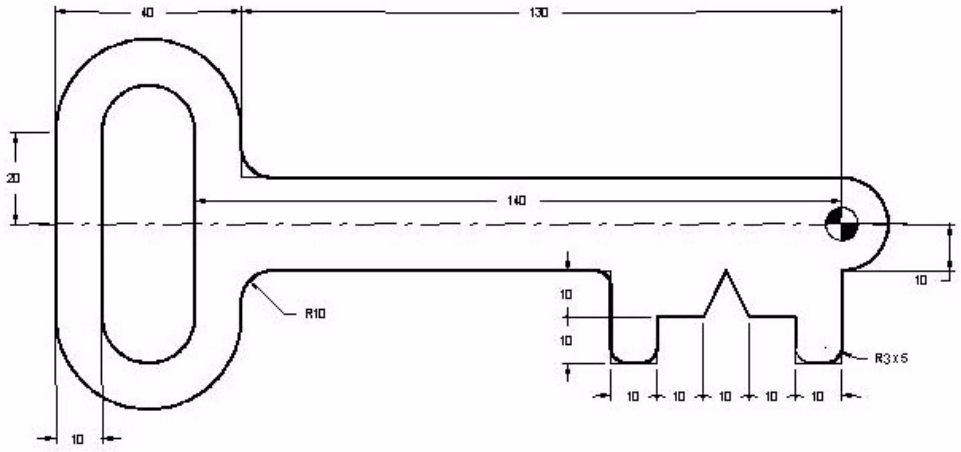
X80
Y40
X200
Y20
X260
Y120
X220 Y180
X160
Y140
X200
Y80
X80
Y140
X120
Y180
X60
X20 Y120

```
Y20
G38 I10
G40 X-10 Y-10
```

END

```
N2:
#RPT[N1,N2,4]
G0 Z100
M30
```

EXERCISE 5



HEADER

G0 Z100
T4D1
M6
S1000 M3
X-90 Y-40
Z0
N1:
G1 G91 Z-5 F160
G90 G42 Y-10
G37 I10

GEOMETRY

X-50
G36 I3
Y-30
G36 I3
X-40
G36 I3
Y-20
X-30
X-25 Y-10
X-20 Y-20
X-10
Y-30
G36 I3
X0
G36 I3
Y-10
G3 X0 Y10 R10
G1 X-130
G36 I10
Y20

```
G3 X-170 Y20 R20
G1 Y-20
G3 X-130 Y-20 R20
G1 Y-10
G36 I10
X-90
G38 R10
G40 Y-40
```

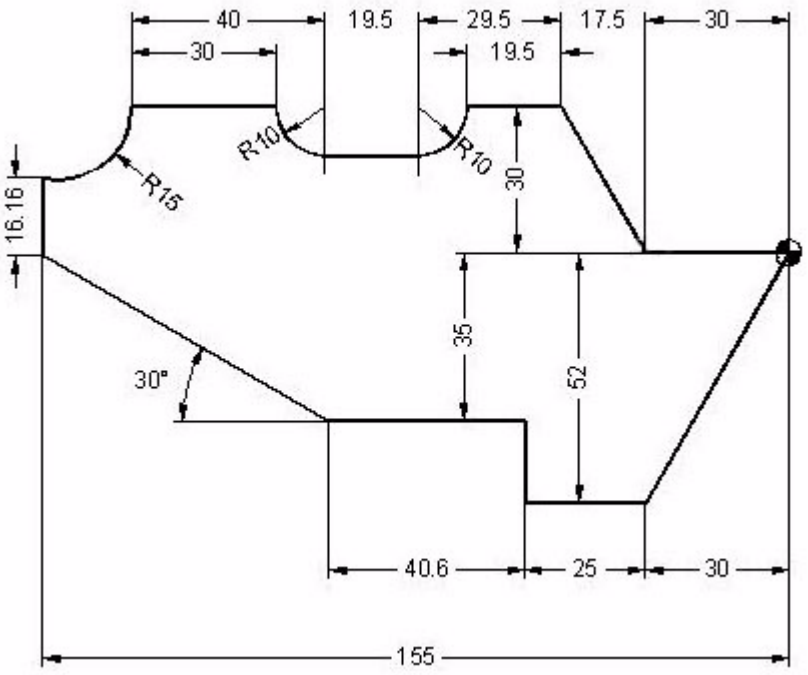
END

```
N2:
#RPT [N1,N2,3]
G0 Z100
```

Replace the current tool with another one whose Ø20 to do the slot.

```
T2D1
M6
X-150 Y20
Z2
G1 Z0 F100
G91 Z-5
Y-40
Z-5
Y40
G0 G90 Z100
M30
```

EXERCISE 6



HEADER

G0 Z100
T4D1
M6
S1000 M3
X25 Y25
Z0
N1:
G1 G91 Z-5 F100
G90 G41 X0 Y0 F1000
G37 I10

GEOMETRY

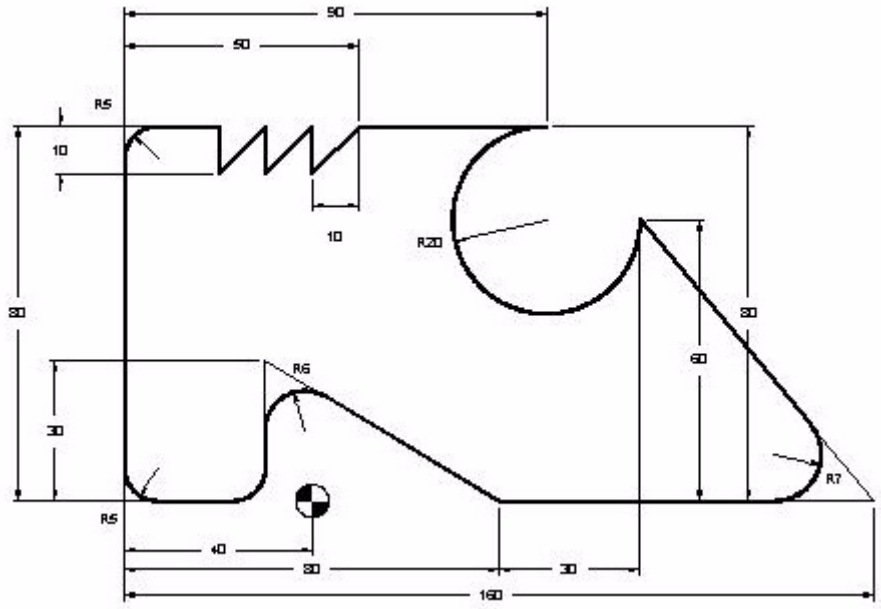
X-30 Y-52
X-55
Y-35
X-95.6
X-155 Y0
G91 Y16.16
G90 G3 X-136.5 Y30 R15
G91 G1 X30
G3 X10 Y-10 R10
G1 X19.5
G3 X10 Y10 R10
G1 X19.5
G90 X-30 Y0
X0 Y0

```
G38 I10
G40 X25 Y25
```

END

```
N2:
#RPT [N1,N2,4]
G0 Z100
M30
```

EXERCISE 7



HEADER

G0 Z100
T5D1
M6
S1000 M3
X20 Y-30
Z0
N1:
G1 G91 Z-5 F100
G90 G42 X40 Y0 F1000
G37 I10

GEOMETRY

X120
G36 I7
X70 Y60
G2 X50 Y80 R-20 The arc radius has a negative sign because it exceeds 180º.
G1 X10
X0 Y70
Y80
X-10 Y70
Y80
X-20 Y70
Y80
X-40
G36 I5
Y0
G36 I5

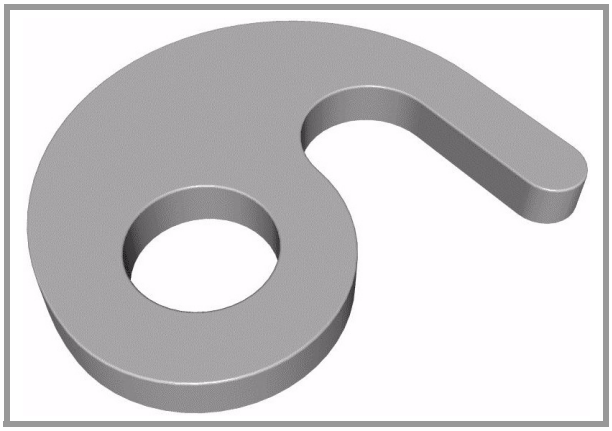
```
X-10
G36 I5
Y30
G36 I6
X40 Y0
G38 I10
G40 X20 Y-30
```

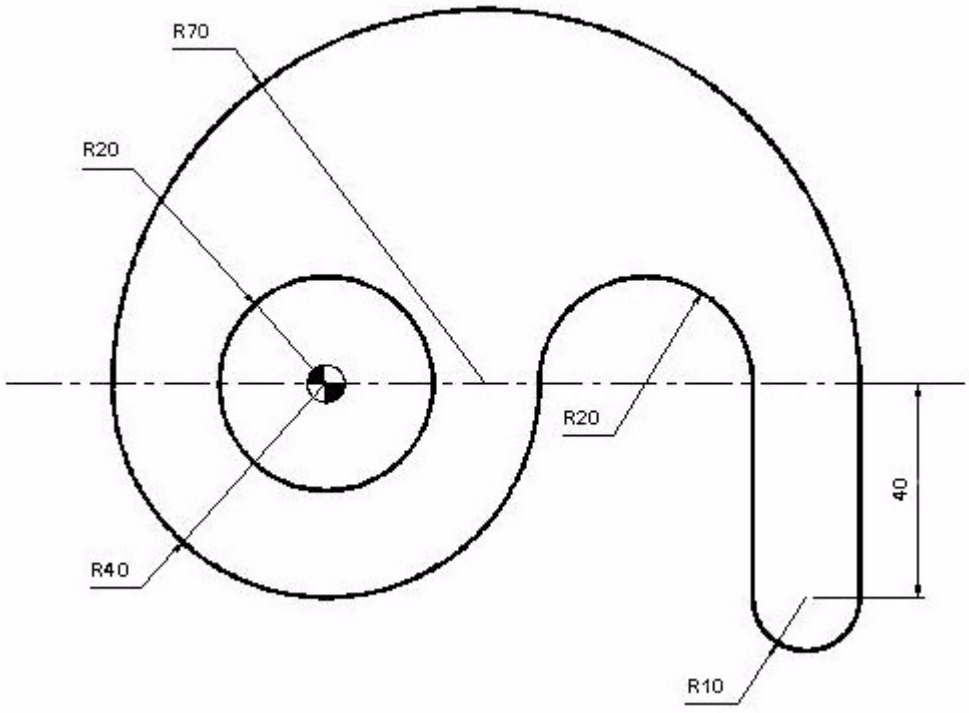
END

```
N2:
#RPT[N1,N2,4]
G0 Z100
M30
```

EXERCISE 8

All circular interpolation exercises are based on the following figure.





CIRCULAR INTERPOLATION. G2/3 XY R

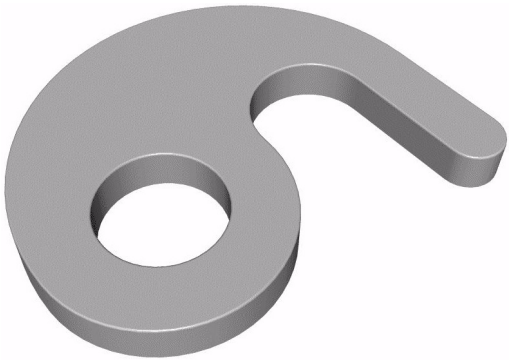
Exercise done using the format:

G2/3 X_ Y_ R_

XY End point.

R Radius of the arc.

```
G0 Z100
T4D1
M6
S1000 M3
X-70 Y0
Z0
N1:
G1 G91 Z-5 F100
G90 G42 X-40 Y0 F1000
G37 I10
G3 X40 Y0 R40
G2 X80 Y0 R20
G1 Y-40
G3 X100 Y-40 R10
G1 Y0
G3 X-40 Y0 R70
G1 Z20
G1 X-20 Y0
G1 Z-20
G3 X-20 Y0 I20 J0
G1 Z20
G38 I10
G1 G40 X-70 Y0
G1 Z-20
N2:
#RPT [N1,N2,3]
G0 Z100
M30
```



CIRCULAR INTERPOLATION. G2/3 XY IJ

Exercise done using the format:

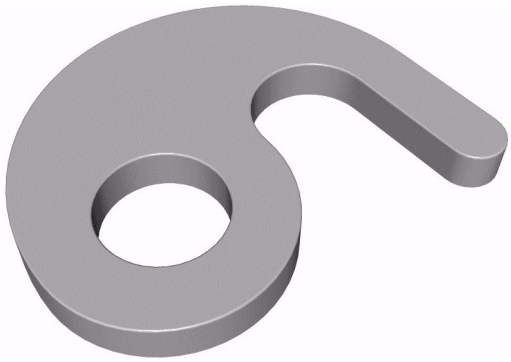
G2/3 X_ Y_ I_ J_

XY End point.

IJ They define the arc center in incremental coordinates referred to the arc's starting point.

The arc center has been defined by incremental auxiliary coordinates.

```
G0 Z100
T4D1
M6
S1000 M3
X-70 Y0
Z0
N1:
G1 G91 Z-5 F100
G90 G42 X-40 Y0 F1000
G37 I10
G3 X40 Y0 I40 J0
G2 X80 Y0 I20 J0
G1 Y-40
G3 X100 Y-40 I10 J0
G1 Y0
G3 X-40 Y0 I-70 J0
G1 Z20
G1 X-20 Y0
G1 Z-20
G3 X-20 Y0 I20 J0
G1 Z20
G38 I10
G1 G40 X-70 Y0
G1 Z-20
N2:
#RPT [N1,N2,3]
G0 Z100
M30
```



CIRCULAR INTERPOLATION. G6 G2/3 XY IJ

Exercise done using the format:

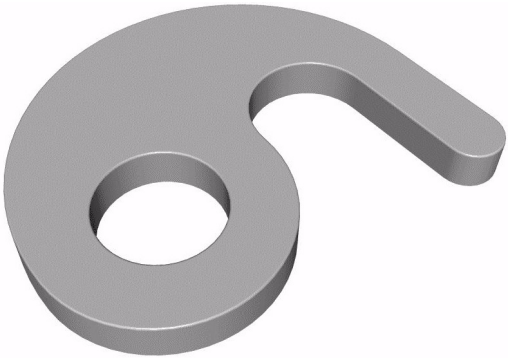
G6 G2/3 X_ Y_ I_ J_

XY End point.

IJ Arc center referred to part zero, only if G6 is at the beginning of the block.

The arc center has been defined by absolute auxiliary coordinates.

```
G0 Z100
T4D1
M6
S1000 M3
X-70 Y0
Z0
N1:
G1 G91 Z-5 F100
G90 G42 X-40 Y0 F1000
G37 I10
G6 G3 X40 Y0 I0 J0
G6 G2 X80 Y0 I60 J0
G1 Y-40
G6 G3 X100 Y-40 I90 J-40
G1 Y0
G6 G3 X-40 Y0 I30 J0
G1 Z20
G1 X-20 Y0
G1 Z-20
G3 X-20 Y0 I20 J0
G1 Z20
G38 I10
G1 G40 X-70 Y0
G1 Z-20
N2:
#RPT [N1,N2,3]
G0 Z100
M30
```



CIRCULAR INTERPOLATION. G2/3 Q IJ

Exercise done using the format:

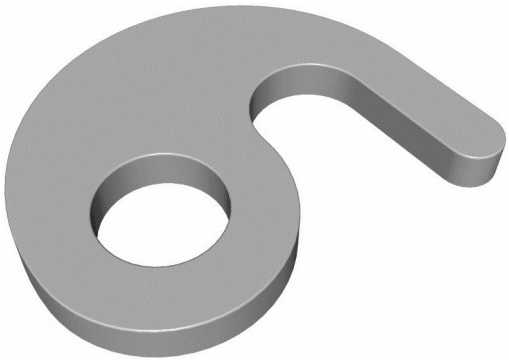
G2/3 Q_ I_ J_

Q Angle.

IJ Incremental distance from the arc's starting point to the arc center.

Using the Polar format with the center in incremental coordinates.

```
G0 Z100
T4D1
M6
S1000 M3
X-70 Y0
Z0
N1:
G1 G91 Z-5 F100
G90 G42 X-40 Y0 F1000
G37 I10
G3 Q0 I40 J0
G2 Q0 I20 J0
G1 Y-40
G3 Q0 I10 J0
G1 Y0
G3 Q180 I-70 J0
G1 Z20
G1 X-20 Y0
G1 Z-20
G3 X-20 Y0 I20 J0
G1 Z20
G38 I10
G1 G40 X-70 Y0
G1 Z-20
N2:
#RPT [N1,N2,3]
G0 Z100
M30
```



CIRCULAR INTERPOLATION. G6 G2/3 Q IJ

Exercise done using the format:

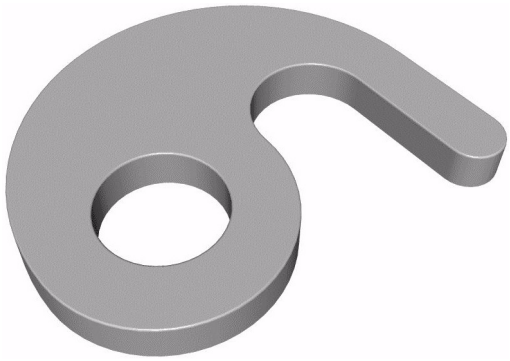
G6 G2/3 Q_ I_ J_

Q Angle.

IJ Arc center referred to part zero, only if G6 is at the beginning of the block.

Using Polar format and center definition in absolute coordinates.

```
G0 Z100
T4D1
M6
S1000 M3
X-70 Y0
Z0
N1:
G1 G91 Z-5 F100
G90 G42 X-40 Y0 F1000
G37 I10
G6 G3 Q0 I0 J0
G6 G2 Q0 I60 J0
G1 Y-40
G6 G3 Q0 I90 J-40
G1 Y0
G6 G3 Q180 I30 J0
G1 Z20
G1 X-20 Y0
G1 Z-20
G3 X-20 Y0 I20 J0
G1 Z20
G38 I10
G1 G40 X-70 Y0
G1 Z-20
N2:
#RPT [N1,N2,3]
G0 Z100
M30
```



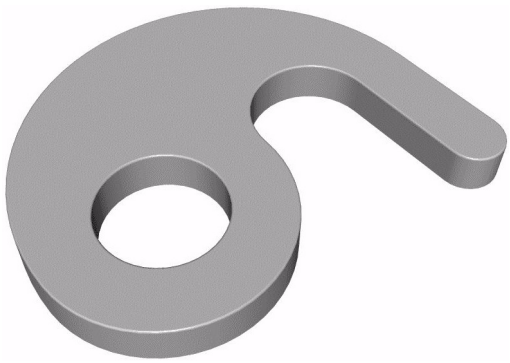
CIRCULAR INTERPOLATION. G2/3 Q

Exercise defining the Polar center (G30) and then the movement to carry out.

G30 I J Definition of the Polar center.
G2/3 Q Interpolation with an angle.

IJ Absolute arc center coordinates referred to part zero. The Polar center is not affected by the incremental coordinates because the format itself is already absolute.

```
G0 Z100
T4D1
M6
S1000 M3
X-70 Y0
Z0
N1:
G1 G91 Z-5 F100
G90 G42 X-40 Y0 F1000
G37 I10
G30 I0 J0
G6 G3 Q0
G30 I60 J0
G6 G2 Q0
G1 Y-40
G30 I90 J-40
G6 G3 Q0
G1 Y0
G30 I30 J0
G6 G3 Q180
G1 Z20
G1 X-20 Y0
G1 Z-20
G3 X-20 Y0 I20 J0
G1 Z20
G38 I10
G1 G40 X-70 Y0
G1 Z-20
N2:
#RPT [N1,N2,3]
G0 Z100
M30
```



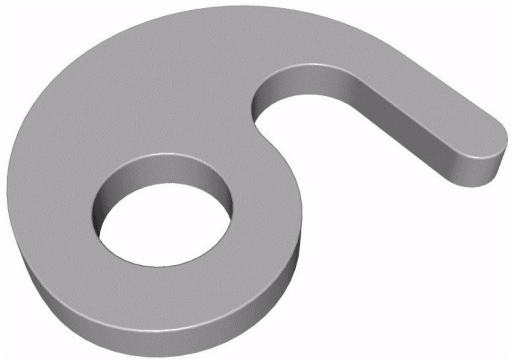
CIRCULAR INTERPOLATION. G8 XY

Exercise done using the format:

G8 X_ Y_

XY End point.

Function for an arc tangent to previous arc.



```
G0 Z100
T4D1
M6
S1000 M3
X-70 Y0
Z0
N1:
G1 G91 Z-5 F100
G90 G42 X-40 Y0 F1000
G37 I10
G3 X40 Y0 R40
G8 X80 Y0
G1 Y-40
G8 X100 Y-40
G1 Y0
G8 X-40 Y0
G1 Z20
G1 X-20 Y0
G1 Z-20
G3 X-20 Y0 I20 J0
G1 Z20
G38 I10
G1 G40 X-70 Y0
G1 Z-20
N2:
#RPT [N1,N2,3]
G0 Z100
M30
```

CIRCULAR INTERPOLATION. G9 XY IJ

Exercise done using the format:

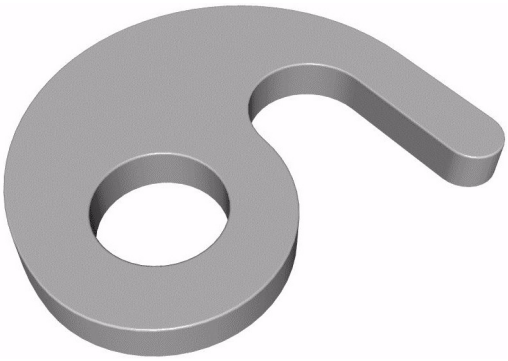
G8 X_ Y_ I_ J_

XY End point.

IJ It defines any point of the arc.

Using the function for an arc defined by three points.

```
G0 Z100
T4D1
M6
S1000 M3
X-70 Y0
Z0
N1:
G1 G91 Z-5 F100
G90 G42 X-40 Y0 F1000
G37 I10
G9 X40 Y0 I0 J-40
G9 X80 Y0 I60 J20
G1 Y-40
G9 X100 Y-40 I90 J-50
G1 Y0
G9 X-40 Y0 I30 J70
G1 Z20
G1 X-20 Y0
G1 Z-20
G3 X-20 Y0 I20 J0
G1 Z20
G38 I10
G1 G40 X-70 Y0
G1 Z-20
N2:
#RPT [N1,N2,3]
G0 Z100
M30
```



CIRCULAR INTERPOLATION. G9 RQ IJ

Using the function for an arc defined by three points, in Polar.

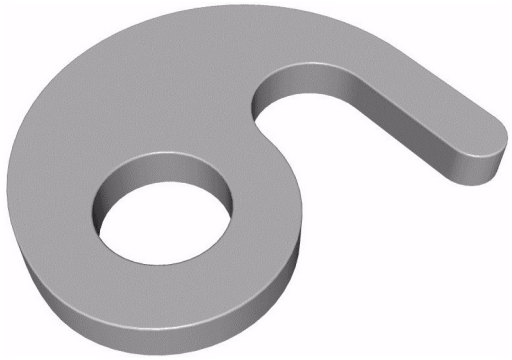
G30 I J Definition of the Polar center. Using absolute auxiliary coordinates.

G9 R_ Q_ I_ J_

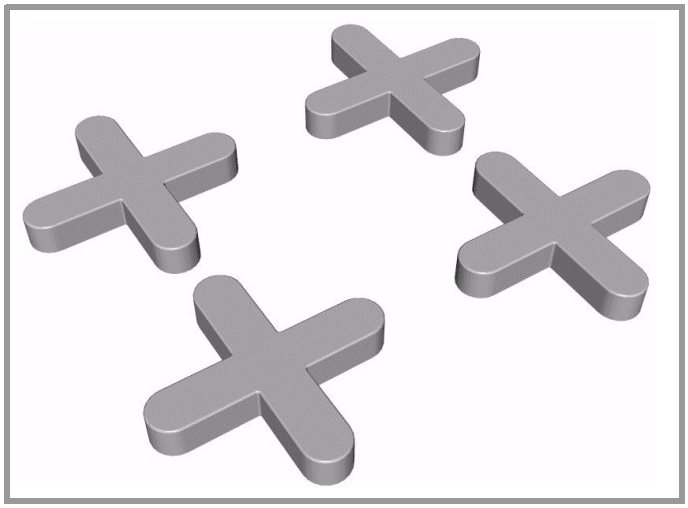
RQ Arc radius and angle referred to the Polar center.

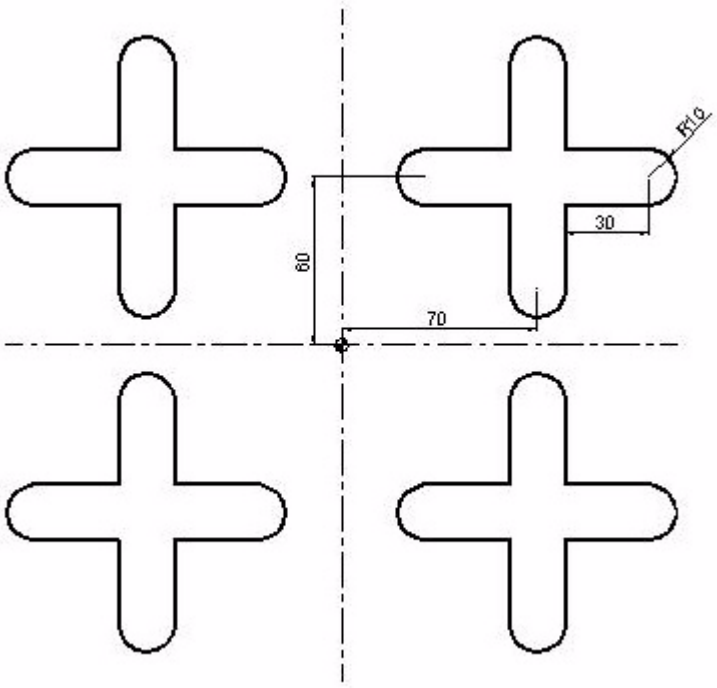
IJ It defines any point of the arc.

```
G0 Z100
T4D1
M6
S1000 M3
X-70 Y0
Z0
N1:
G1 G91 Z-5 F100
G90 G42 X-40 Y0 F1000
G30 I0 J0
G37 I10
G9 R40 Q0 I0 J-40
G30 I60 J0
G9 R20 Q0 I60 J20
G1 Y-40
G30 I90 J-40
G9 R10 Q0 I90 J-50
G1 Y0
G30 I30 J0
G9 R70 Q180 I30 J70
G1 Z20
G1 X-20 Y0
G1 Z-20
G3 X-20 Y0 I20 J0
G1 Z20
G38 I10
G1 G40 X-70 Y0
G1 Z-20
N2:
#RPT [N1,N2,3]
G0 Z100
M30
```



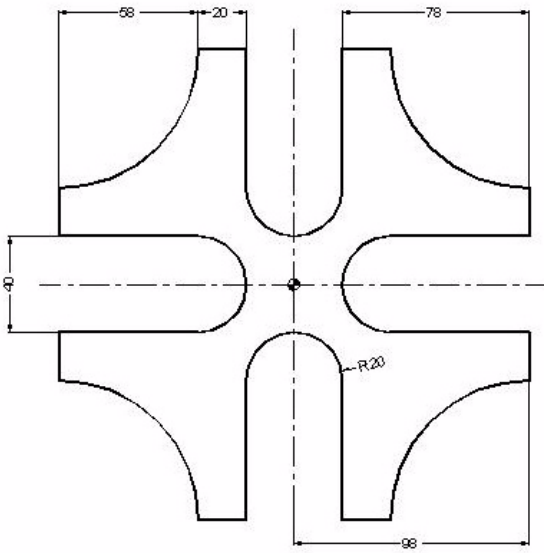
EXERCISE 9. MIRROR FUNCTION





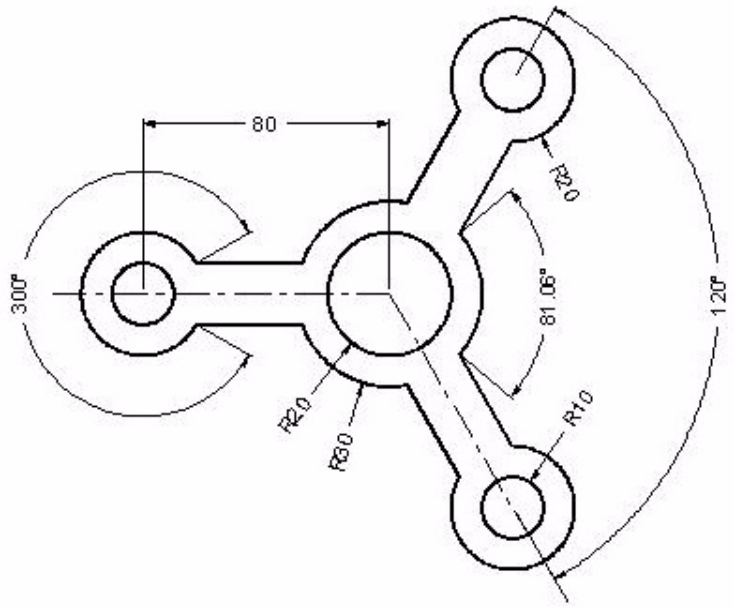
```
N1:
G0 Z100
T4D1
M6
S1000 M3
X100 Y20
Z0
 G1 Z-5 F100
G42 X100 Y50 F1000
 X110
G3 X110 Y70 R10
G1 X80
Y100
G3 X60 Y100 R10
G1 Y70
X30
G3 X30 Y50 R10
G1 X60
Y20
G3 X80 Y20 R10
G1 Y50
X100
G40 Y20
G0 Z100
N2:
G11                    Mirror function in X.
#RPT[N1,N2]
G10
G12                    Mirror function in Y.
#RPT[N1,N2]
G10                    Mirror function cancellation.
G11 G12
#RPT[N1,N2]
G10
M30
```

EXERCISE 10. COORDINATE (PATTERN) ROTATION




```
G0 Z100
T4D1
M6
S1000 M3
X120 Y0
Z0
N3:
G1 G91 Z-5 F100
G90 G42 X98 Y20 F1000
G37 I10
N1:
Y40
G2 X40 Y98 R58
G1 X20
Y40
G2 X-20 Y40 R20
G1 Y98
G73 Q90                 Coordinate rotation
N2:
#RPT[N1,N2,3]
G73                     Cancellation of coordinate rotation
G38 I10
 G40 X120 Y0
N4:
#RPT [N3,N4,5]
G0 Z100
M30
```

EXERCISE 11. COORDINATE (PATTERN) ROTATION IN POLAR



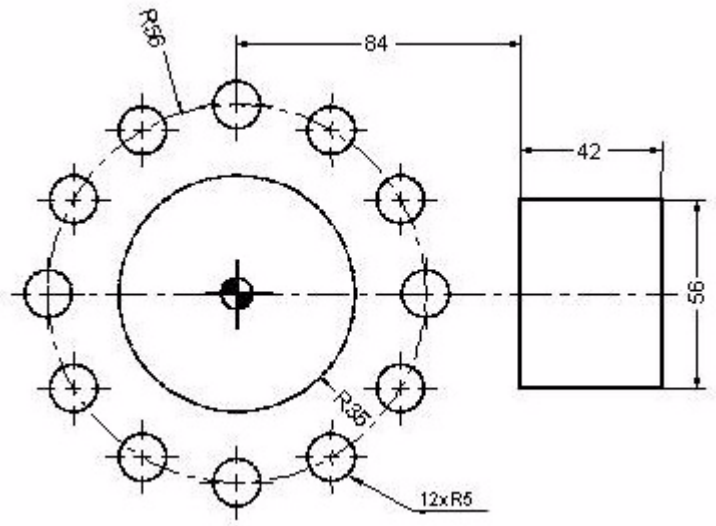

```
G0 Z100
T4D1
M6
S1000 M3
R60 Q120
Z0
N3:
G1 G91 Z-5 F100
G90 G42 R30 Q120 F1000
G37 I10
N1:
G3 Q160.53
G30 I-80 J0
G1 R20 Q30
G3 Q-30
G30 I0 J0
G1 R30 Q-160.53
G3 Q-120
G73 Q120
N2:
#RPT[N1,N2,2]
G73
G38 I10
G30 I0 J0
 G40 G1 R60 Q120
N4:
#RPT [N3,N4,5]
G0 Z100
G99 X0 Y0
G88 Z2 I-30 D2 J20 B3
```

```
G0 G80 Z100
G99 R80 Q180
G88 Z2 I-30 D2 J10 B3
G91 Q120
G91 Q120
G90 G0 G80 Z100
M30
```

EXERCISE 12. CANNED CYCLES 1

Programming cycles always has the following sequence:

1. Prior positioning (starting plane).
2. Type of withdrawal (G98/G99) and XY position.
3. Cycle definition.
4. Cycle cancellation (G90) and withdrawal.

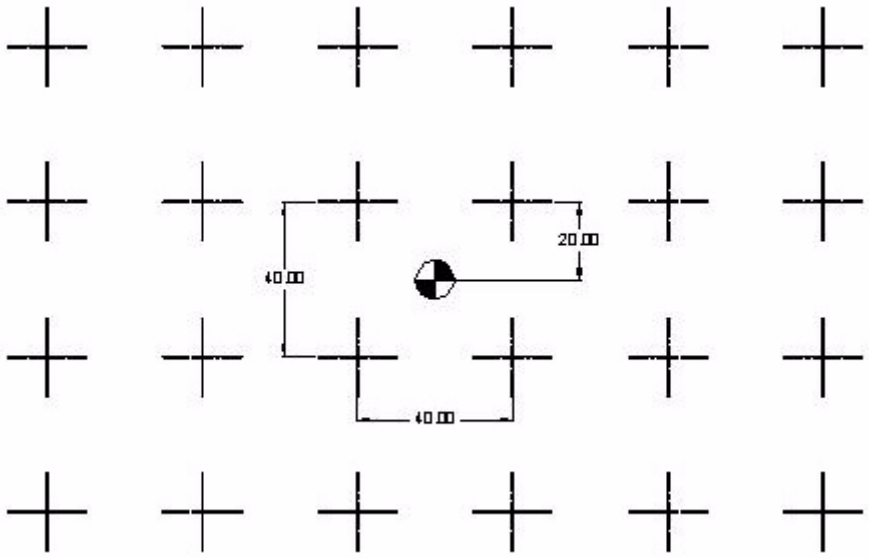



```
G0 Z100
T4 D1
M6
S1000 M3
G99 X0 Y0 F1000
G88 Z2 I-10 D2 J35 B3 L0.5 H500 V50      Circular pocket canned cycle.
G0 G80 Z100
X105 Y0
G87 Z2 I-10 D2 J21 K28 B3 L1 H480 V30    Rectangular pocket canned cycle.
G0 G80 Z100
T11 D1
M6
X0 Y56 G81 Z2 I-10                       Direct drilling.
N1:
G91 Q30 ->ANGULAR INCREMENT
N2:
#RPT[N1,N2,10]                           Angular repetition.
G90 G0 G80 Z100
M30
```

EXERCISE 13. CANNED CYCLES 2

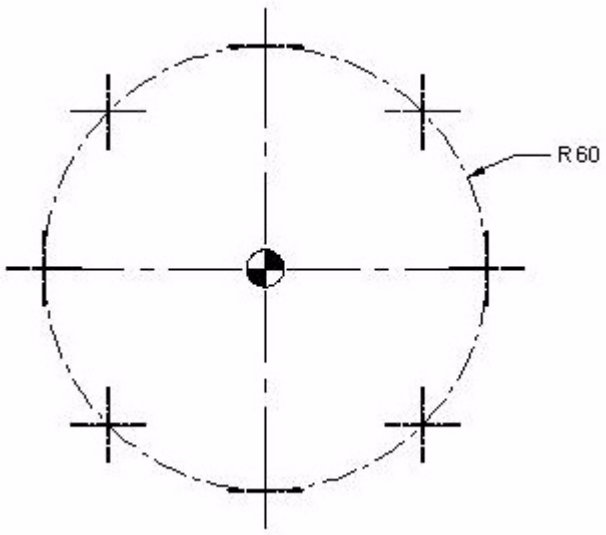
Any cycle, once defined, may be repeated in several ways using multiple machining.

1. G160 - Multiple positioning in a straight line.
2. G161 - Multiple positioning in a parallelogram pattern.
3. G162 - Multiple positioning in a grid pattern.
4. G163 - Multiple positioning in a circular pattern.
5. G165 - Multiple positioning using an arc chord.



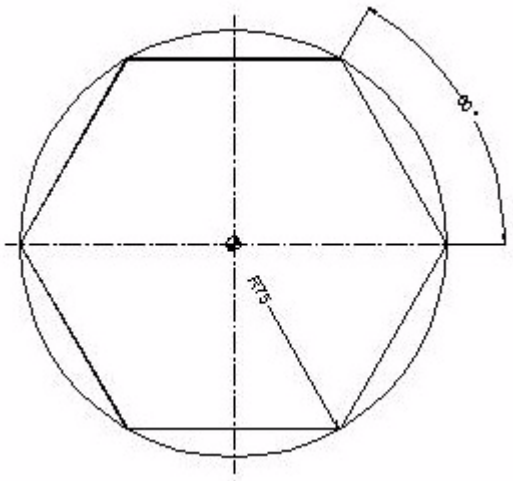

```
G0 Z100
T6 D1
M6
S1000 M3
G99 X-100 Y60 F1000      Coordinate of the first drilling point (hole).
G81 Z2 I-10
G162 I40 K6 J-40 D4      Multiple machining in a grid pattern.
G0 G80 Z100
M30
```

EXERCISE 14. CANNED CYCLES 3



G0 Z100
T6 D1
M6
S1000 M3
G99 X-42.4264 Y-42.4264 F1000Coordinate of the first drilling point (hole).
G81 Z2 I-10
G163 X42.4264 Y42.4264 I45
G0 G80 Z100
M30

EXERCISE 15. ANGULAR REPETITION




```
G0 Z100
T4D1
M6
S1000 M3
X100 Y0
Z2
G1 Z0 F175
N1:G91 Z-5
G90G42 X75 Y0          Repetition of down movements.
N3:G91 Q60             Polar programming of the first side.
N4:
#RPT [N3,N4,5]         Angular repetition of the sides.
G90 G40 X100 Y0
N2:
#RPT[N1,N2,4]          Repetition of down movements.
G0 Z100
M30
```

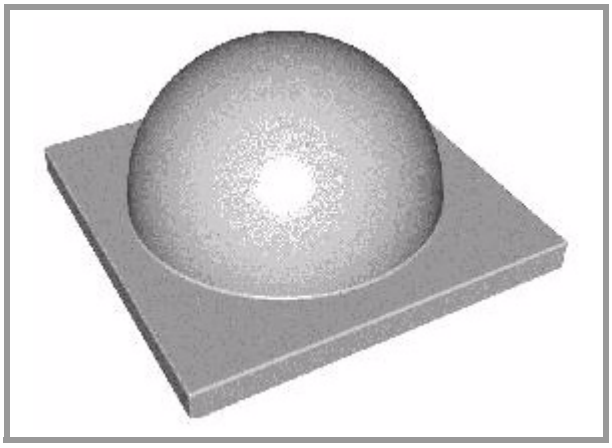
PARAMETRIC PROGRAMMING 4

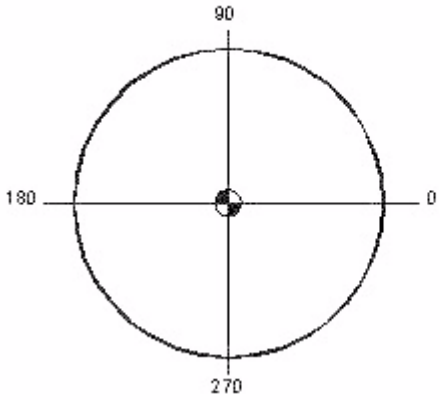
Parametric programming basically consists in assigning values to certain parameters, identified with the letter "P", to perform the necessary operations in order to different shapes on the same part.

A parametric program is basically made up of three parts:

1. Assignment.
2. Operation.
3. Comparison.

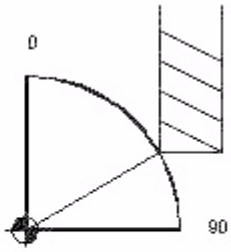
EXERCISE 1. SEMI-SPHERE





SIN P101 = Z/R

COS P101 = X/R



P100 = Initial radius
P101 = Initial angle
P102 = Final angle
P103 = Incremental angle
P104 = Tool radius

P110 = P100 * SIN P101
Z=R*SIN P101

P111 = P100 * COS P101
X = R*COS P101

PARAMETER ASSIGNMENT

P100=60	Radius of the semi-sphere.
P101=90	Initial angle.
P102=0	Final angle.
P103=0.5	Incremental angle.
P104=8	Tool radius.

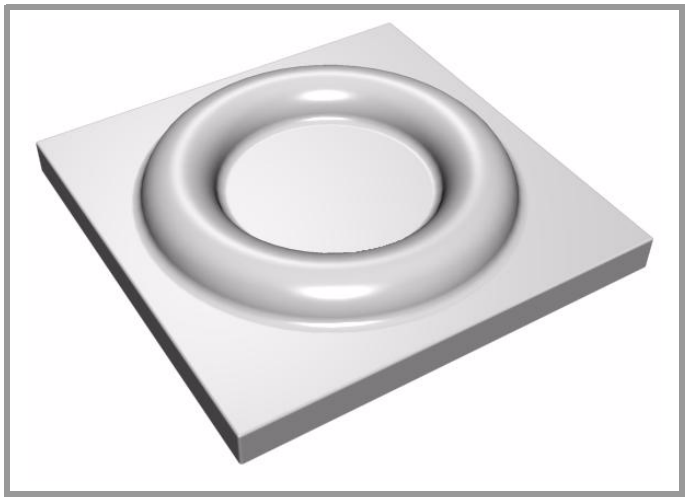
PROGRAM

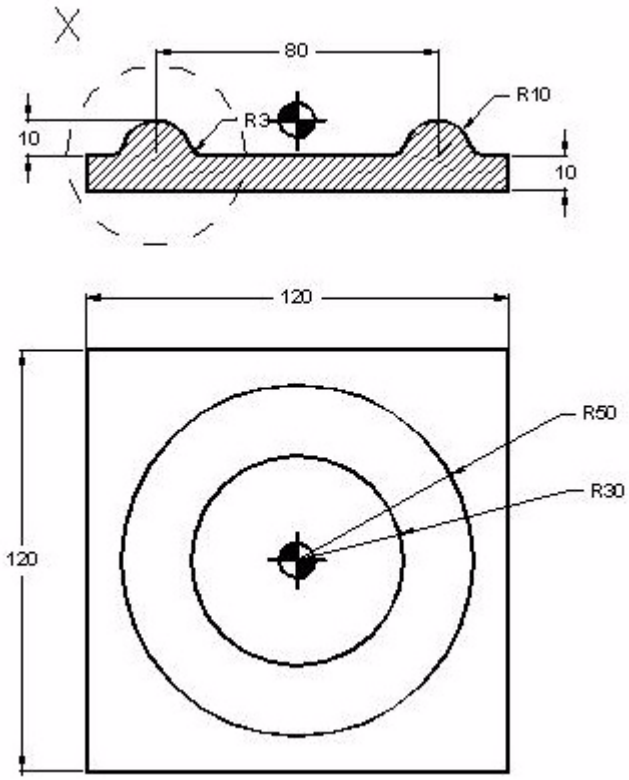
```
G0 Z100
T12D1
M6
S1000 M3
X0 Y0
N1: P120= P100*COS [P101] P121=P100*SIN [P101]   XZ position.
P120=P120+P104                                   Tool compensation.
P121=P121-P100                                   Zero up.
G1 XP120 ZP121 F1000
G2 Q360
N2:
P101=P101-P103                                   Angular decrement.
```

COMPARISON

```
$IF P101 > P102 $GOTO N1
P101=P102
#RPT [N1,N2]
G0 Z100
M30
```

EXERCISE 2. TOROID (DONUT)





PARAMETER ASSIGNMENT

```
P100=-90
P101=90
P102=1
P103=10
P104=3
P105=-P103
P106=40
P120=P103+P104
```

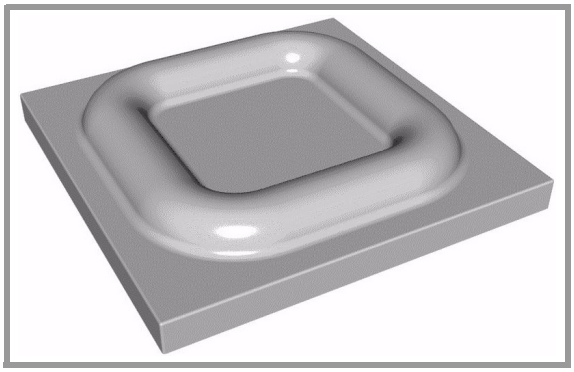
PROGRAM

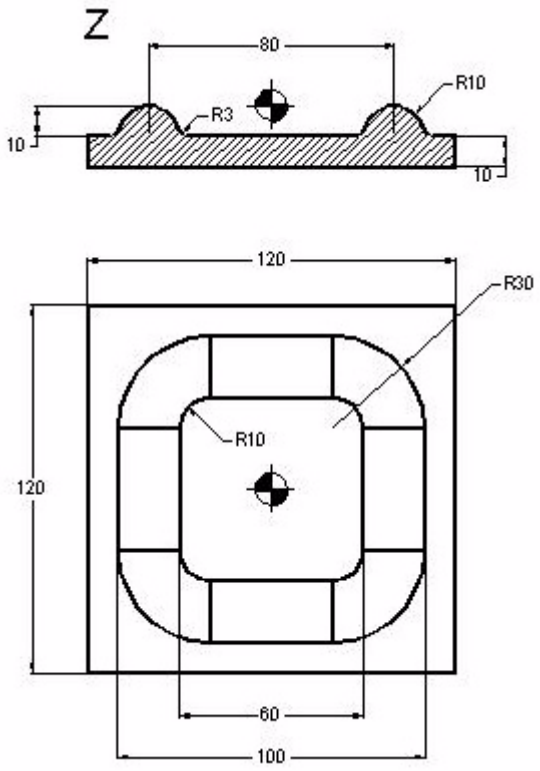
```
G0 Z100
T12D1
M6
S1000 M3
X0 Y0
N1:G18
G30 IP105 JP106
G1 RP120 QP100 F1000
G17
G30 I0 J0
G3 Q360
N2:
P100=P100+P102
```

COMPARISON

```
$IF P100<P101 $GOTO N1
P100=P101
#RPT [N1,N2]
G0 Z100
M30
```

EXERCISE 3. ASHTRAY





PARAMETER ASSIGNMENT

```
P100=-90
P101=90
P102=1
P103=10
P104=3
P105=-P103
P106=40
P120=P103+P104
```

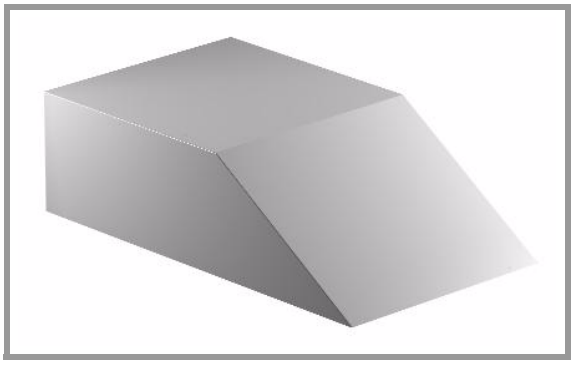
PROGRAM

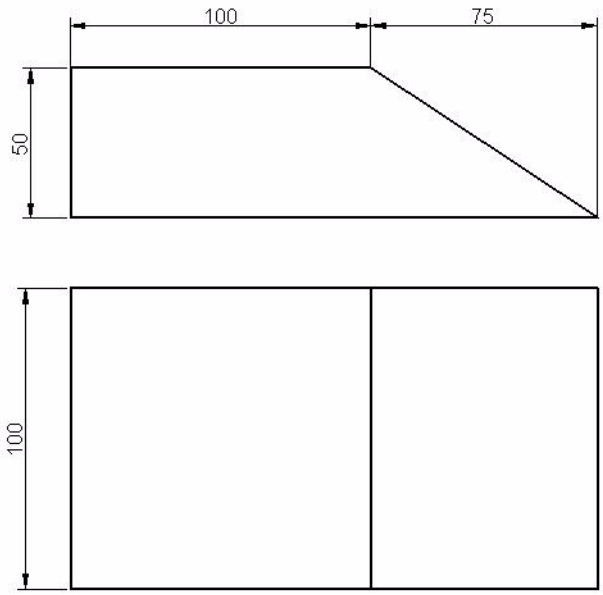
```
G0 Z100
T12D1
M6
S1000 M3
X0 Y0
N1:G18
G30 IP105 JP106
G1 RP120 QP100 F1000
G17
G1 Y20
G6 G3 Q90 I20 J20
G1 X-20
G6 G3 Q180 I-20 J20
G1 Y-20
G6 G3 Q-90 I-20 J-20
G1 X20
G6 G3 Q0 I20 J-20
G1 Y0
N2:
P100=P100+P102
```

COMPARISON

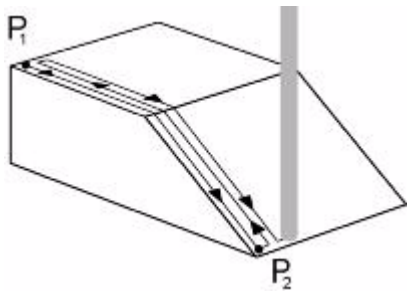
```
$IF P100<P101 $GOTO N1
 P100=P101
#RPT [N1,N2]
G0 Z100
M30
```

EXERCISE 4. WEDGE





Program a wedge by assigning parameters. Then, by using positioning and increments, do a comparison between the initial point and the final point to be reached.

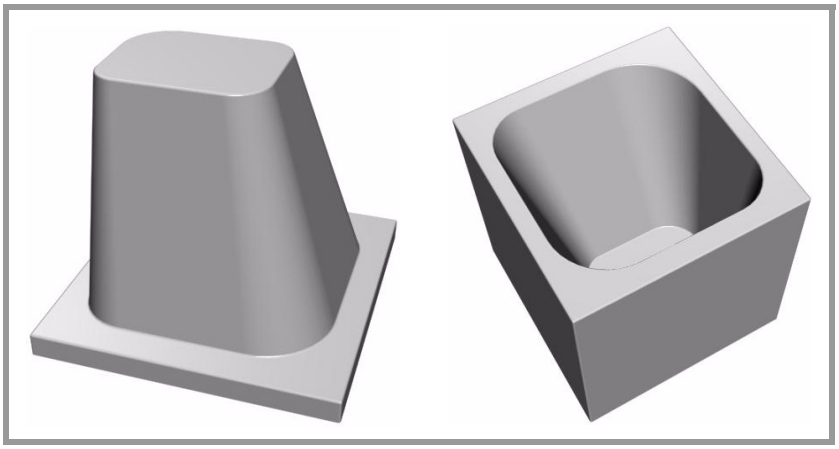


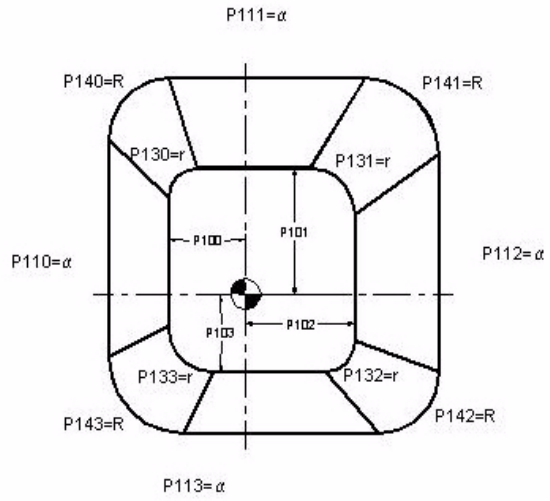
Parametric programming is handy when trying to change the assignment of parameters to obtain the desired dimensions using the same program.

DOING THE EXERCISE

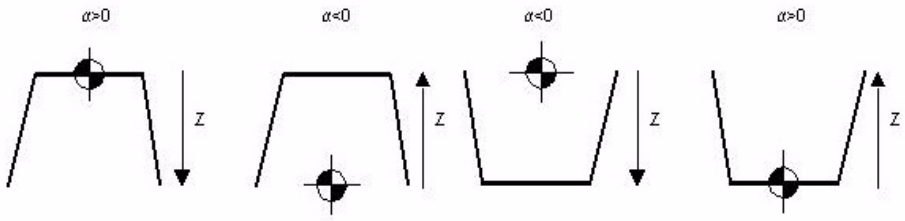
```
P100=100Length in X
P101=100                Lenth in Y
P102=75                 Movement in X
P103=50                 Depth
P106=2
G0 Z100                 Z position
T4 D1
M6                      Calling a tool
S1000 M3
Y0
N1:                     Label number 1
X0
Z0
G1 XP100 F1000          Initial X position
G1 G91 XP102 Z-P103
G90 YP106
G1 Z10
N2:
P106=P106+2
$IF P106<P101  $GOTO N1 Comparison. If P106 is smaller than P101, the tool returns to label 1
#RPT[N1,N2]             Repetition. Last pass
G0 Z100
M30
```

EXERCISE 5. POCKETS WITH 4 SIDES AND 4 DIFFERENT RADII





P100 = Width -X
P101 = Width +Y
P102 = Width +X
P103 = Width -Y
P104 = Increment in "Z"
P105 = Initial "Z" coordinate
P106 = Final "Z" coordinate
P107 = Tool offset "D"



PARAMETER ASSIGNMENT

```
P102=50 P103=40                          External sides.
P107=5                                   Tool radius.
P125=80 P126=60 P127=50 P128=70          Angles.
P130=5 P131=7 P132=4 P133=8              Small radii.
P140=10 P141=12 P142=15 P143=17          Large radii.
P120=0 P121=1 P122=30
P150=P122-P120 P151=P150/P121 P152=FUP[P151]
P160=P140-P130 P161=P141-P131 P162=P142-P132 P163=P143-P133
P140=P140+P107 P141=P141+P107 P142=P142+P107 P143=P143+P107
P164=P160/P152 P165=P161/P152 P166=P162/P152 P167=P163/P152
G0 Z100
T4 D1
M6
N1: P170=P120/TAN[P125] P171=P120/TAN[P126] P172=P120/TAN[P127] P173=P120/TAN[P128]
P180=P100-P170 P181=P101-P171 P182=P102-P172 P183=P103-P173
```

PROGRAM

```
G01 X-P180 Y0 Z-P120 F2000
YP181
G36 IP140
XP182
G36 IP141
Y-P183
G36 IP142
X-P180
G36 IP143
Y0
N2:
P120=P120+P121
P140=P140-P164 P141=P141-P165 P142=P142-P166 P143=P143-P167
```

COMPARISON

```
$IF P120<P122 $GOTO N1
P120=P122
P140=P130+P107 P141=P131+P107 P142=P132+P107 P143=P133+P107
#RPT[N1,N2]
G00 Z50
M30
```