

A NEW SET OF CANNED CYCLES FOR CNC MILLING MACHINES

Sotiris L. Omirou¹

¹Department of Mechanical Engineering, Frederick University, Nicosia, Cyprus

Abstract

Canned cycles are a powerful manual programming tool for conveniently performing repetitive CNC machine operations. Canned cycles is an option that can be purchased from most CNC manufacturers. However, the number and type of canned cycles is generally limited to certain machining tasks and applied to the most common geometric shapes being unable to accommodate the increasing needs of the manufacturing industry. This paper comes to suggest a new set of canned cycles that extends the feature generating capabilities of CNC milling machines. The development of the new canned cycles is based on parametric programming, an advanced programming technique available on all modern CNC machines. The effectiveness of the proposed new canned cycles is verified by simulation tests for each one of them.

Keywords: - CNC, Canned cycles, Pocket machining, Cutter path generation.

1. INTRODUCTION

Canned cycles are a set of pre-programmed instructions permanently stored in the machine controller that automate many of the required repetitive tasks. Their use, eliminates the need for many lines of programming, reduces the programming time and simplifies the whole programming process [1]. All CNC machining controls come with a set of helpful machining canned cycles. These canned cycles are programmed using the G/M code language by entering a certain G code together with any required variable information. Most machine tool controllers are equipped with standard canned cycles such as pocketing, slotting, drilling, threading, etc

In particular as regards the pocketing, pocket milling is one of the common machining operations used for removing the material inside a closed boundary on a flat surface of a workpiece to a fixed depth. Rectangular and circular form of pockets are the standard types of pockets found as canned cycles in the programming capabilities of modern CNC systems. Other forms of pockets are handled by using a CAD/CAM system to generate the necessary tool path. However, CAD/CAM systems turn to the segmentation

technique to generate the required tool path. Thus, they first approximate the geometric shape of the pocket with a series of short segments of lines and arcs and then use the existing linear and circular motion commands to drive the tool along them. Inevitably, the generated voluminous part programs with these motions lead to continuous succession of accelerations and decelerations of the driving motors causing feedrate variations, decrease machining accuracy and require more machining time.

To avoid this drawback, research is focused in developing various methods for automatic CNC pocket machining bypassing the CAD/CAM dependency [2-6]. This paper, following the present intention of research engineers to take advantage of the hardware capabilities of modern CNC systems, proposes a set of three pairs of new canned cycles that can be embodied in the control of a CNC milling machine, enriching its instruction set.

Namely, the proposed canned cycles are the following: (a) Spherical pocket / boss, (b) Toroidal pocket / boss, (c) Conical pocket / boss (Fig. 1):

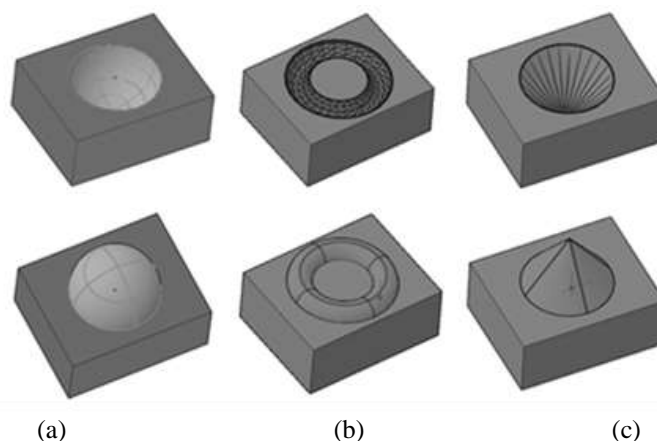


Fig.1: The proposed new CNC canned cycles

The rest of the paper is organized as follows: Section 2 describes in detail the mechanism of defining and using parameters in the environment of Heidenhain control CNC System. Sections 3, 4 and 5 present the development of the three pairs of canned cycles respectively. Finally, section 6 summarizes the contribution of this paper to manufacturing process.

2. PARAMETRIC PROGRAMMING

Parametric programming constitutes an innovative programming technique of G/M-code programming language. This innovation simplifies the CNC procedures dramatically, lessens the user’s dependence on CAD/CAM and contributes to bear minimum cycle time [7]. As production quantities grow, every second of program execution time becomes more critical. An important factor that minimizes the cycle time is how the program is formatted. While CAM systems have come a long way, it can be difficult, if not impossible, to cause some CAM systems to output CNC programs as efficiently as can be done with parametric programming technique. A series of significant machining applications have become feasible through the employment of parametric programming technique [8 -10].

Applying the technique of parametric programming, a part program can be written using variables, arithmetic and logic statements similar to usual programming languages such as Basic, Pascal or C. The specific is that in the case of CNC programming, all these parametric expressions are used to represent the machine axis position (x, y, z, a, etc.), feedrate, and spindle speed functions.

2. 1 Parameters in Heidenhain Control System

Like other computer programming languages, parametric programming comes in several versions. The most popular are those of Heidenhain [11] and Fanuc (Custom Macro B) [12, 13] controls. Others include User Task (from Okuma), Q Routine (from Sodick), and Advanced Programming Language [APL] (from G& L). The present work uses the latest version of Heidenhain control, called iTNC 530.

In the Heidenhain control [8], parameters are designated by the letter Q and a number between 0 and 99. Using Q parameters, an entire family of parts can be accommodated by a single part program. In such a case, Q parameters are entered in the part program instead of fixed numerical values. For example, several parts may require machining a pocket (Fig. 1) with different sizes. A single parametric program can be called up to machine the specific feature. Before the execution of the parametric program, the numerical values for each pocket are transferred to the respective parameters. This approach eliminates the redundant codes in part program and substantially reduces the size of the program and programming time.

Q parameters also enable the programming of contours that are defined with mathematical functions or can be used to make the execution of machining steps depend on logical conditions. According to the implementing function, Q parameters are classified in three function groups (table 1.):

Table 1: Codification of parameters in Heidenhain control

Function group	Operation	Code	Example	Explanation
Basic arithmetic	assign	D00	D00 Q1 P01 60	assigns number 60 to parameter Q1
	addition	D01	D01 Q2 P01 Q3 P02 5	assigns the sum of Q3 and 5 to parameter Q2
	subtraction	D02	D02 Q3 P01 Q4 P02 1	assigns the difference Q4 -1 to parameter Q3
	multiplication	D03	D03 Q4 P01 Q5 P02 3	assigns the product Q5x3 to parameter Q4
	division	D04	D04 Q5 P01 Q6 P02 Q7	assigns the quotient Q6/Q7 to parameter Q5
	square root	D05	D05 Q6 P01 Q7	assigns the square root of Q7 to parameter Q6
Trigonometric functions	sine	D06	D06 Q7 P01 Q8	assigns the sin of angle Q8 to parameter Q7
	cosine	D07	D07 Q8 P01 Q9	assigns the cos of angle Q9 to parameter Q8
	root sum of squares	D08	D08 Q9 P01 5 P02 4	assigns the result for values 5 and 4 to Q9
Conditional jumps	If equal jump	D09	D09 P01 Q1 P02 Q2 P03 1	If Q1 = Q2, jump to the given label (1).
	If not equal jump	D10	D10 P01 Q3 P02 Q4 P03 2	If Q3 ≠ Q4, jump to the given label (2).
	if greater than jump	D11	D11 P01 Q5 P02 Q6 P03 3	If Q5 > Q6, jump to the given label (3).
	If less than jump	D12	D12 P01 Q7 P02 Q8 P03 4	If Q7 < Q8, jump to the given label (4).

As it is deduced by the examples above, Q parameters and fixed numerical values can be freely mixed within a part program.

3. DEVELOPMENT OF THE SPHERICAL POCKET / BOSS CANNED CYCLES.

3.1 Tool Path Planning for Spherical Pocket (Fig. 2)

A tool commonly used for generating the specific form of surfaces is a spherical end milling cutter (ball end mill).

A convenient tool center path consists of a series of small circular arcs of prescribed step angle in the XZ plane, followed by full circular motion in the XY plane, until the bottom of the pocket is reached. More specifically, the whole process plan is described below:

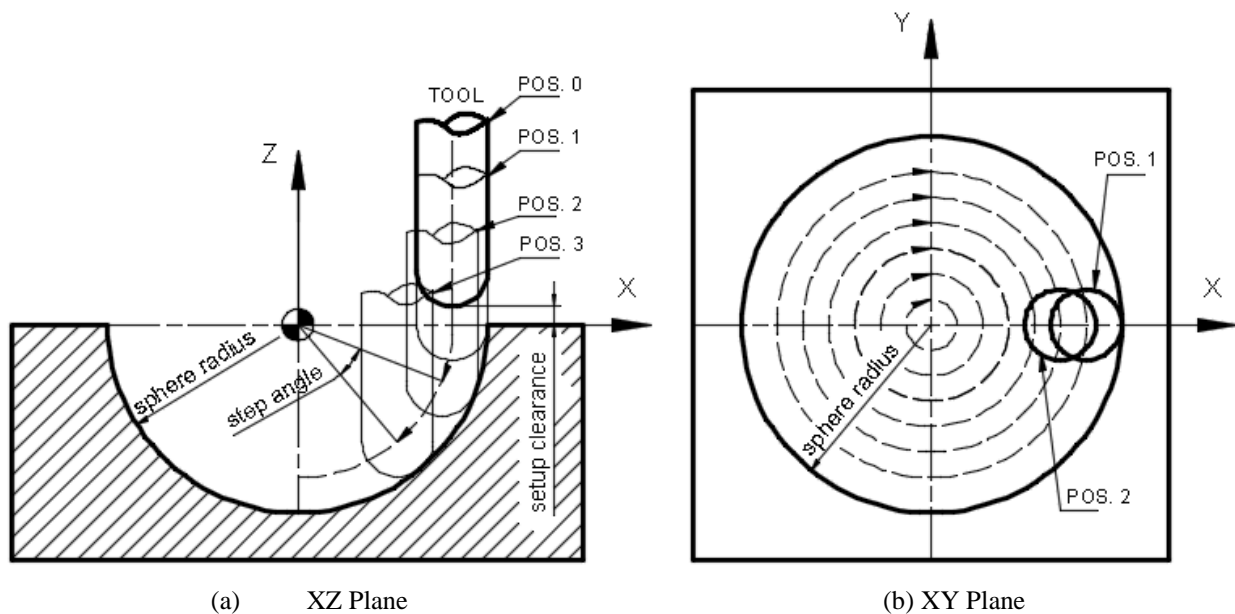


Fig.2: Tool path planning for a spherical pocket

Process Plan

a. Tool Approach

With the zero reference point selected at the center of the pocket, the tool is initially pre-positioned to starting position (POS. 0), defined by the following coordinates:

$$X = \text{sphere radius } R - \text{tool radius } r, \quad Y = 0 \text{ mm}, \quad Z = \text{setup clearance.}$$

From this position the tool is plunged in the material with the value of tool radius (POS. 1).

b. Machining

The machining of the spherical cavity is achieved by repeating the following two successive motions:

1. Circular motion in XZ plane along a small circular arc (the two first motions are shown in figure 2.a by consecutive positions POS1, POS2 and POS3).
2. Full circular motion in XY plane with the center of this motion at the center of the sphere and radius the distance between the current tool

position and the sphere center (shown with dashed line in Fig. 2.b).

The two motions keep being repeated as the angular position of the tool in the XZ plane remains smaller than 90° .

c. Tool Retraction

Reaching the 90° angle, the sequence of the two motions stops and the tool is rapidly retracted to the initial setup clearance or to a safe height above the center of the pocket.

The programmable step angle is a factor offering the possibility to control the surface quality of the spherical pocket. The smaller the programmed step angle, the higher the quality of the surface is achieved.

3.2 Tool Path Planning for Spherical Boss (Fig. 3):

The tool path planning for a spherical boss is similar to the case of the spherical pocket. Thus, the tool center path consists of a series of small circular arcs of prescribed step angle in the XZ plane, followed by full circular motion in the XY plane, until the bottom of the boss is reached. More specifically, the whole process plan is described below:

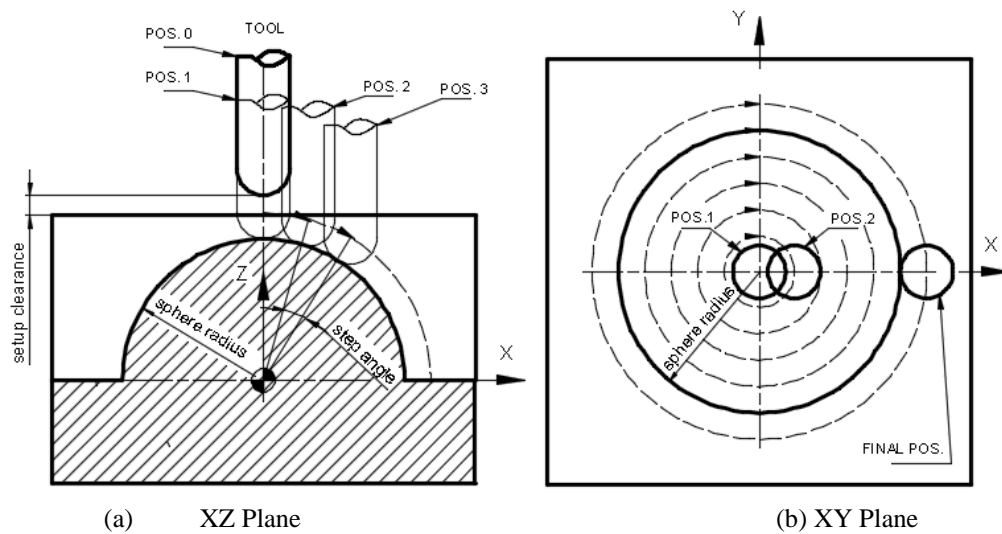


Fig.3: Tool path planning for a spherical boss

Process Plan

a. Tool Approach

With the zero reference point selected at the center of the spherical boss, the tool is initially pre-positioned to starting position (POS. 0), defined by the following coordinates:

$$X = 0 \text{ mm} \quad Y = 0 \text{ mm}, \quad Z = \text{setup clearance.}$$

From this position the tool is vertically plunged in the material to $Z = \text{sphere radius}$ (POS. 1).

b. Machining

The machining of the spherical boss is achieved by repeating the following two successive motions:

1. Circular motion in XZ plane along a small circular arc (the two first motions are shown in Fig. 3.a by consecutive positions POS1, POS2 and POS3).
2. Full circular motion in XY plane with the center of this motion at the center of the sphere and radius the distance between the current tool position and the sphere center (shown with dashed line in Fig. 3.b).

The two motions keep being repeated as the angular position of the tool in the XZ plane remains smaller than 90° .

c. Tool Retraction

Reaching the 90° angle, the sequence of the two motions stops and the tool is rapidly retracted to the initial setup clearance or to a safe height.

The programmable step angle is a factor offering the possibility to control the surface quality of the spherical boss. The smaller the programmed step angle, the higher the quality of the surface is achieved.

3.3 Parametric G-Code Programs and Test Results for Spherical Pocket / Boss

The flowchart shown in Fig. 4 constitutes the common basis of developing the parametric G-code programs for machining any spherical pocket or boss.. The flowchart incorporates initialization of the CNC milling machine, definition of the required parameters and tool motion control as previously described in the process plans.

For the needs of the simulation test, the required Q parameters included in the INPUT DATA box of the flowchart are assigned with the values shown in table 2.

Table 2: Definition of Q parameters – spherical pocket/boss

Q parameter	Meaning	Assigned value
Q1	Tool radius	5 mm
Q2	Sphere radius	90 mm
Q3	Current angle	0°
Q4	Step angle	5°
Q5	Setup clearance	2 mm
Q6	Feedrate	200 mm/min

The parametric G-code programs for a spherical pocket and a spherical boss are shown on the left side of figures 5 and 6 respectively. Both programs are consistent to the flowchart of Fig. 4 and according to the parameters of table 2. The result of the simulation test is shown on the right side of the same figures. To demonstrate the capability of the programs to carry out roughing and finishing operations, the parameter of step angle Q4 in the case of the spherical boss was decreased from 5° to 1° . The higher machining accuracy obtained in this case can be clearly seen on the right side of Fig.6.

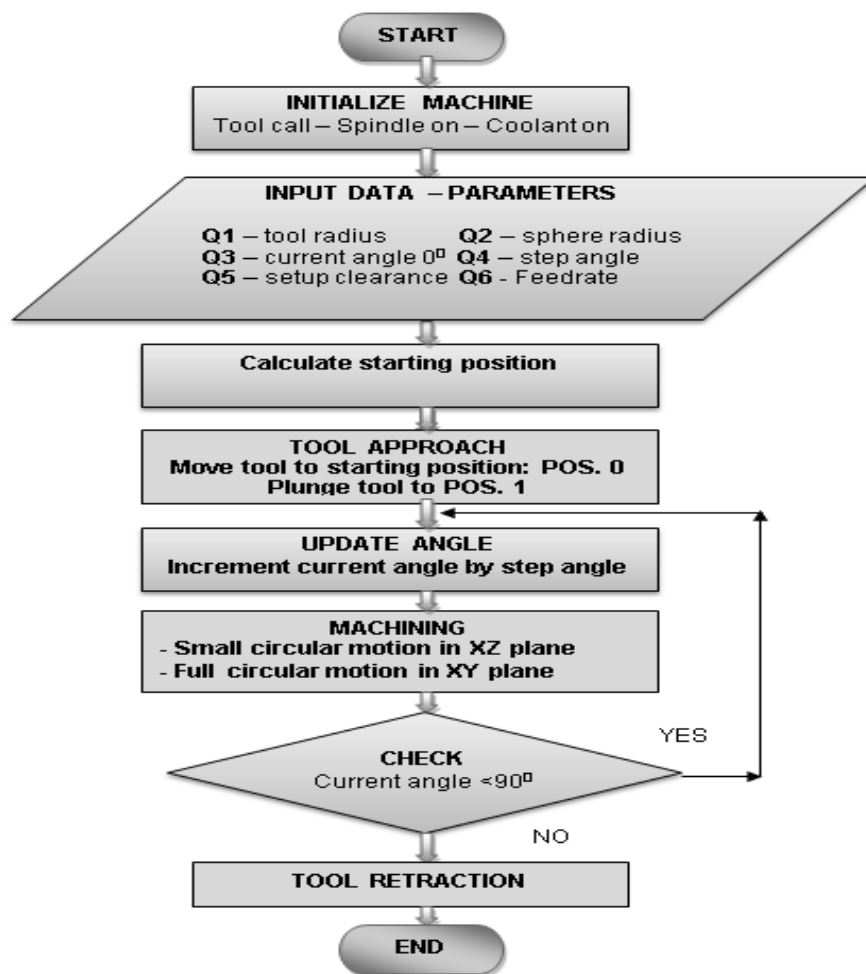


Fig.4: Flowchart of the spherical pocket / boss parametric programs

Manual operation	Test run
<pre> % spherical pocket G71 * N10 G30 G17 X-100 Y-100 Z-100* N20 G31 X+100 Y+100 Z+0* N30 T1 G17 S2000* N40 M13* N50 D00 Q1 P01 +5* N60 D00 Q2 P01 +90* N70 D00 Q3 P01 +0* N80 D00 Q4 P01 +5* N90 D00 Q5 P01 +2* N100 D00 Q6 P01 +200* N110 D02 Q7 P01 +02 P02 +01* N120 G90 G00 X+07 Y+0 Z+05* N130 G01 Z-01 F06* N140 G98 L1* N150 G13 G91 H+04 G90 I+0 K+0* N160 G02 G91 X+0 Y+0 G90 I+0 J+0* N170 D01 Q3 P01 +03 P02 +04* N180 D12 P01 +03 P02 +90 P03 1* N190 G90 G00 Z+2* N200 M30* N99999999 %spherical pocket G71 * </pre>	

Fig.5: Simulation test for a spherical pocket - step angle 5° (Heidenhain control)

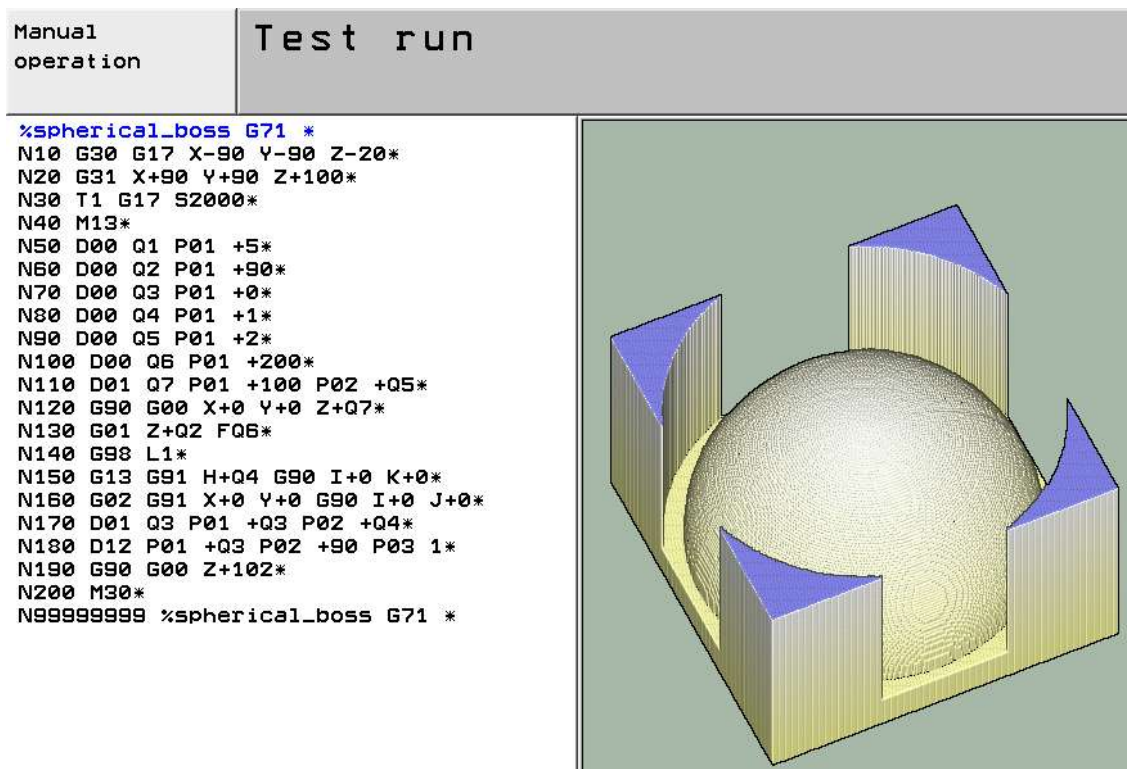


Fig.6: Simulation test for a spherical boss - step angle 1° (Heidenhain control)

4. DEVELOPMENT OF THE TOROIDAL POCKET / BOSS CANNED CYCLES.

4.1 Tool Path Planning for Toroidal Pocket (Fig. 7)

Using a spherical end milling cutter as the appropriate for the case tool, the machining of a toroidal pocket can be achieved according to the following process plan:

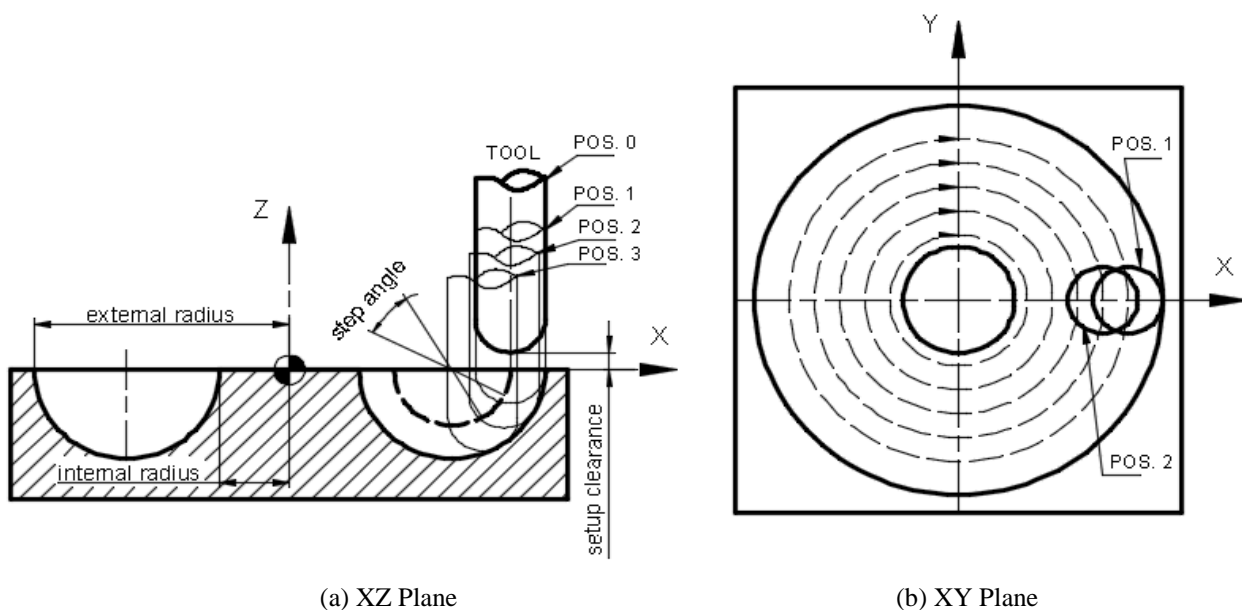


Fig.7: Tool path planning for a toroidal pocket

Process Plan

a. Tool Approach

With the zero reference point selected at the center of the toroid, the tool is initially pre-positioned to starting position (POS. 0), defined by the following coordinates:

$$X = \text{external radius } R - \text{tool radius } r, \quad Y = 0 \text{ mm}, \\ Z = \text{setup clearance}$$

From this position the tool is plunged in the material with the value of tool radius (POS. 1).

b. Machining

The machining of the toroidal cavity is achieved by repeating the following two successive motions:

1. Circular motion in XZ plane along a small circular arc (the two first motions are shown in Fig. 7.a by consecutive positions POS1, POS2 and POS3).

2. Full circular motion in XY plane with the center of this motion at the center of the toroid and radius the distance between the current tool position and the toroid center (shown with dashed line in Fig. 7.b).

The two motions keep being repeated as the angular position of the tool, in the XZ plane, remains smaller than 180°.

c. Tool Retraction

Reaching the 180° angle, the sequence of the two motions stops and the tool is rapidly retracted to the initial setup clearance or to a safe height.

4.2 Tool Path Planning for Toroidal Boss (Fig. 8)

Using a spherical end milling cutter as the appropriate for the case tool, the machining of a toroidal boss can be achieved according to the following process plan:

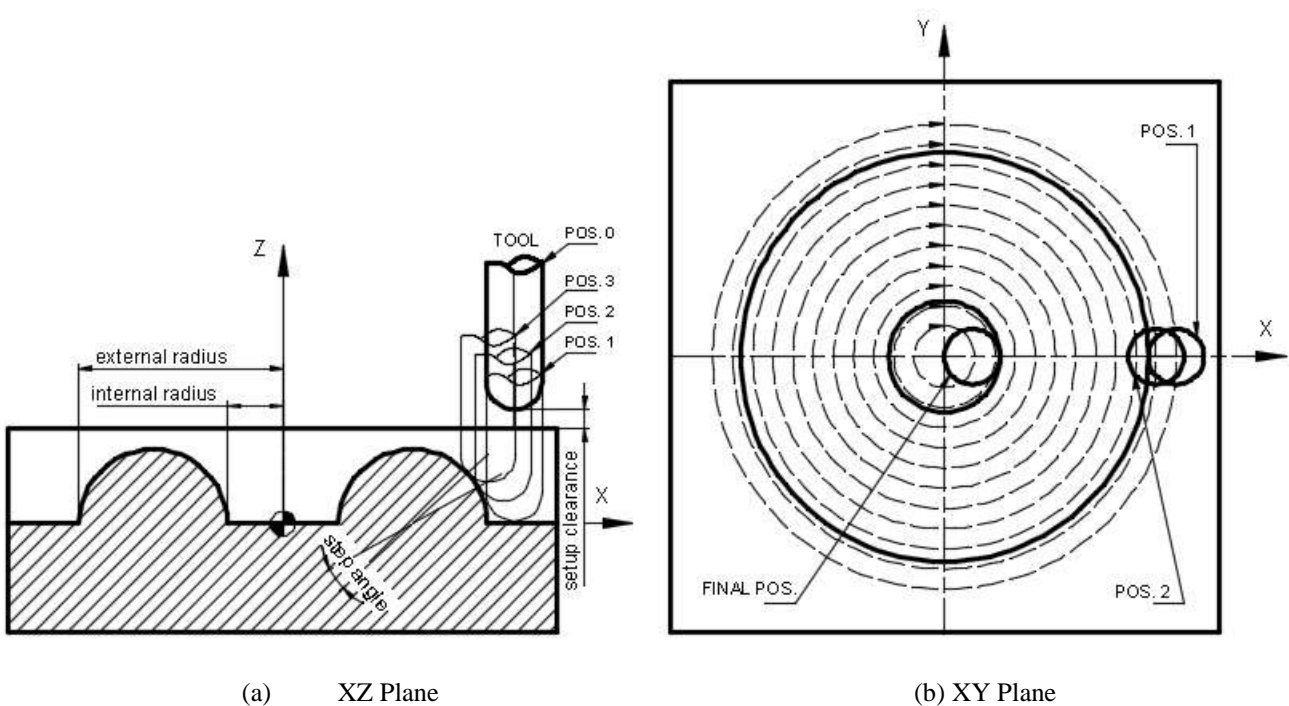


Fig.8: Tool path planning for a toroidal boss

Process Plan

a. Tool Approach

With the zero reference point selected at the center of the toroid, the tool is initially pre-positioned to starting position (POS. 0), defined by the following coordinates:

$$X = \text{external radius } R + \text{tool radius } r, \quad Y = 0 \text{ mm}, \quad Z = \text{setup clearance.}$$

From this position the tool is vertically plunged in the material to Z = 0 (POS. 1).

b. Machining

The machining of the toroidal boss is achieved by repeating the following two successive motions:

1. Circular motion in XZ plane along a small circular arc (the two first motions are shown in Fig. 8.a by consecutive positions POS1, POS2 and POS3).

2. Full circular motion in XY plane with the center of this motion at the center of the toroid and radius the distance between the current tool position and the toroid center (shown with dashed line in Fig. 8.b).

The two motions keep being repeated as the angular position of the tool, in the XZ plane, remains smaller than 180° .

c. Tool Retraction

Reaching the 180° angle, the sequence of the two motions stops and the tool is rapidly retracted to the initial setup clearance or to a safe height.

4.3 Parametric G-Code Programs and Test Results for Toroidal Pocket/Boss

The flowchart shown in Fig. 9 constitutes the common basis of developing the parametric G-code programs for machining any toroidal pocket or boss. The flowchart incorporates initialization of the CNC milling machine, definition of the required parameters and tool motion control as previously described in the process plans.

For the needs of the simulation test, the required Q parameters included in the INPUT DATA box of the flowchart are assigned with the values shown in table 3.

Table 3: Definition of Q parameters – toroidal pocket/boss

Q parameter	Meaning	Assigned value
Q1	Tool radius	5 mm
Q2	External radius	100 mm
Q3	Internal radius	60 mm
Q4	Current angle	0°
Q5	Step angle	5°
Q6	Setup clearance	2 mm
Q7	Feedrate	200 mm/min

The parametric G-code programs for a toroidal pocket and a toroidal boss are shown on the left side of figures 10 and 11 respectively. Both programs are consistent to the flowchart of Fig. 9 and according to the parameters of table 3. After running the simulation test, the machined parts were cross-sectioned for better observation and verification of the desired tool path. As it can be seen, in both cases, the generated geometry exactly responds to the expected geometrical form of the workpieces.

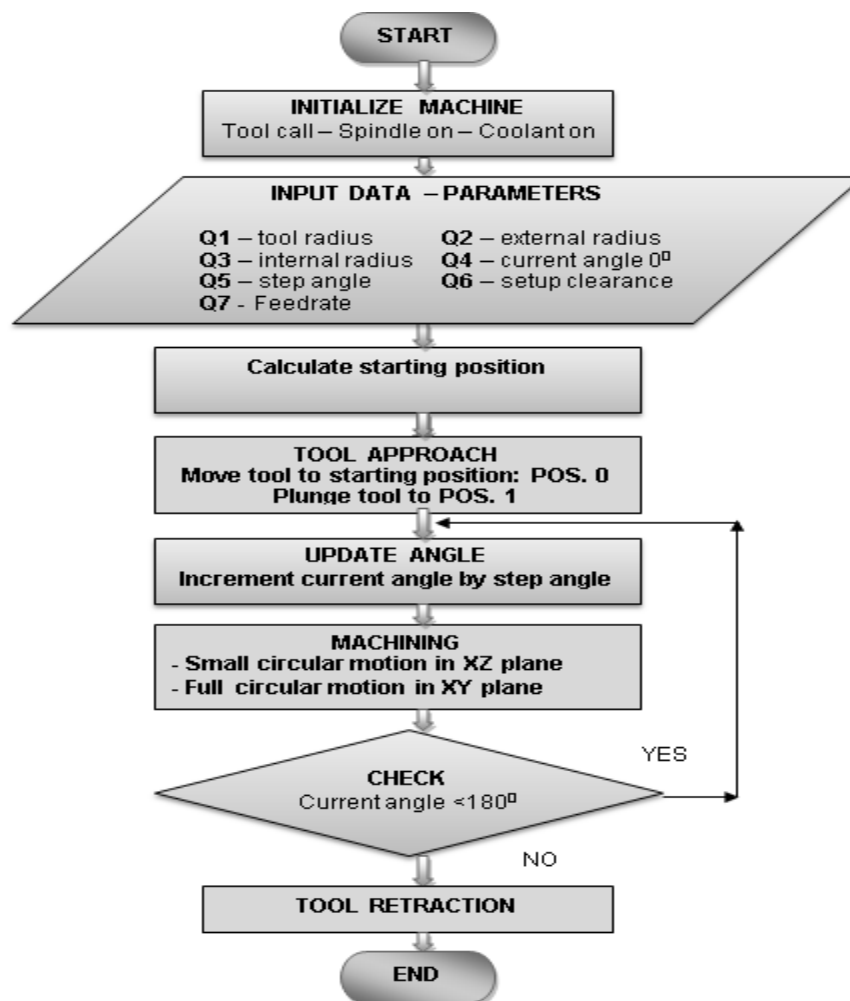


Fig.9: Flowchart of the toroidal pocket / boss parametric programs

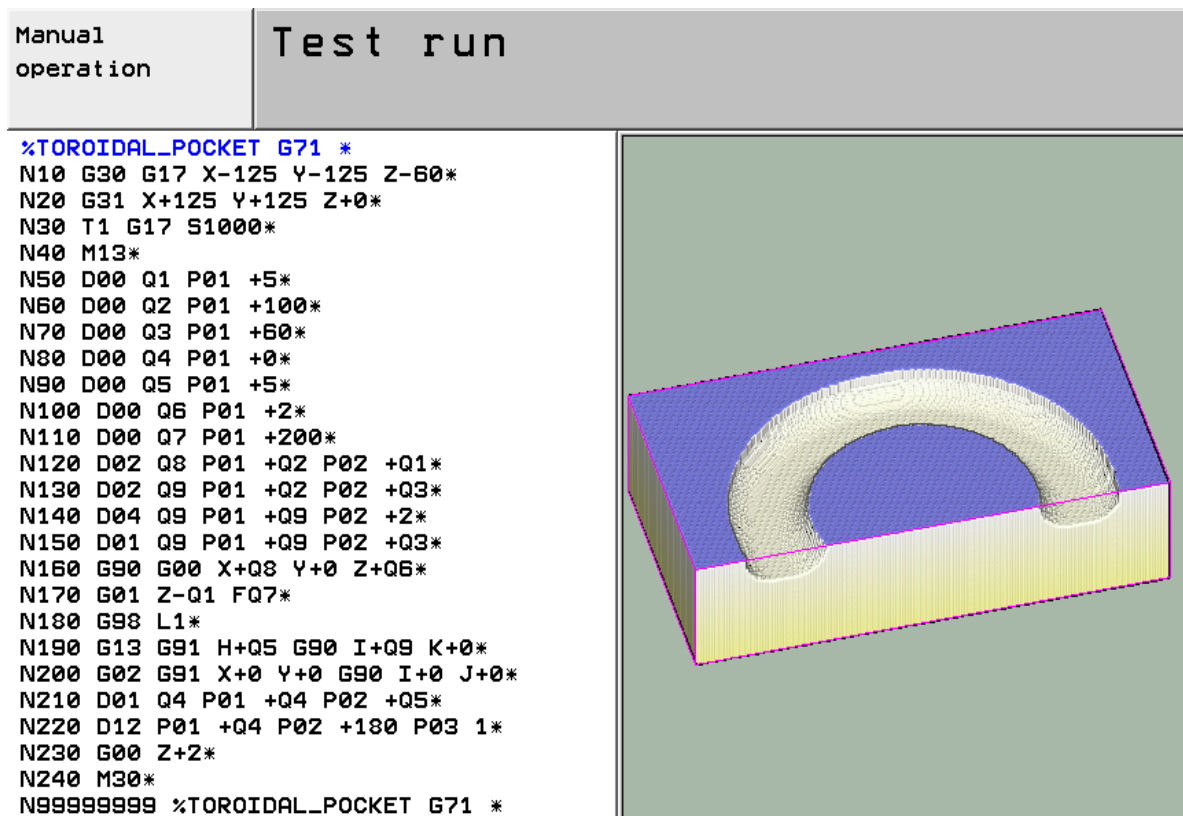


Fig.10: Simulation test for a toroidal pocket - step angle 5° - section view - (Heidenhain control)

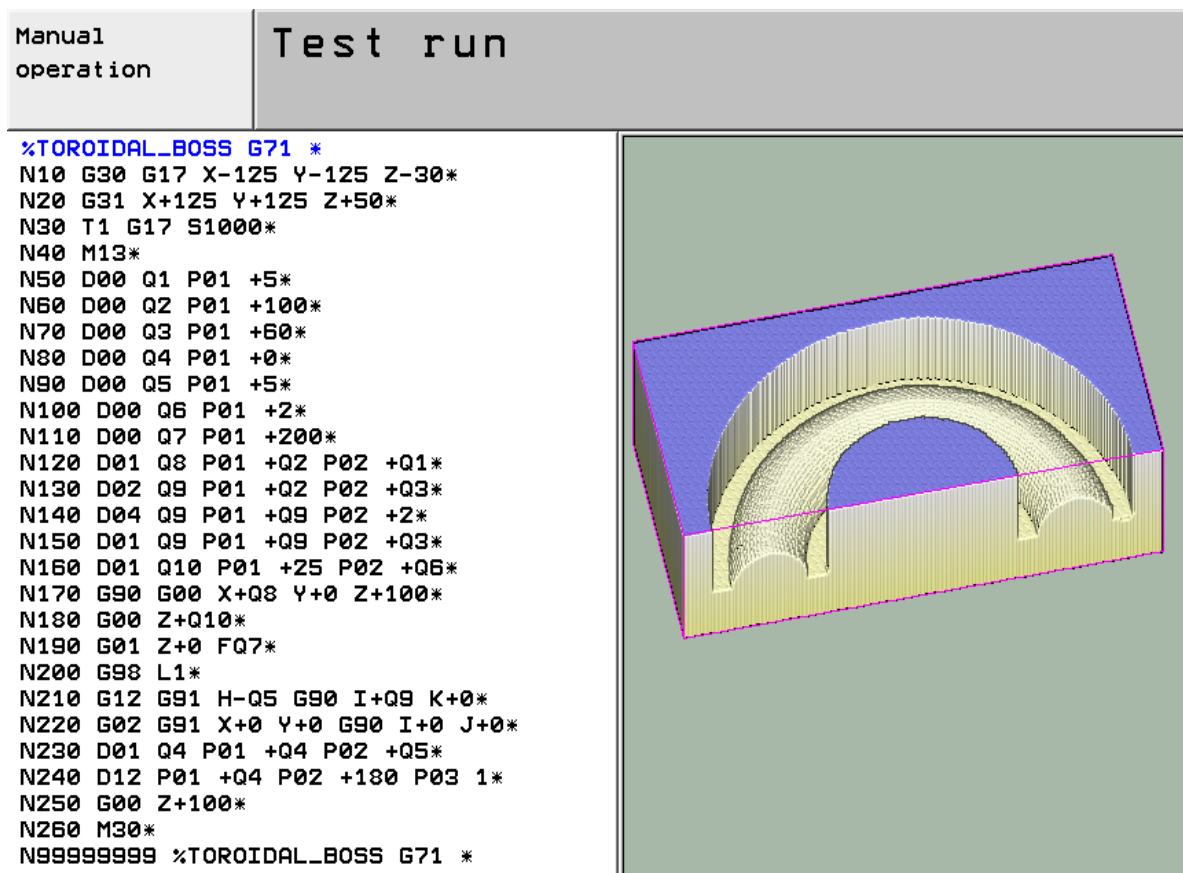


Fig.11: Simulation test for a toroidal boss - step angle 5° - section view - (Heidenhain control)

5. DEVELOPMENT OF THE CONICAL POCKET / BOSS CANNED CYCLES

5.1 Tool Path Planning for Conical Pocket (Fig. 12)

Using a flat end mill as the appropriate for the case tool, the machining of a conical pocket can be achieved according to the following process plan:

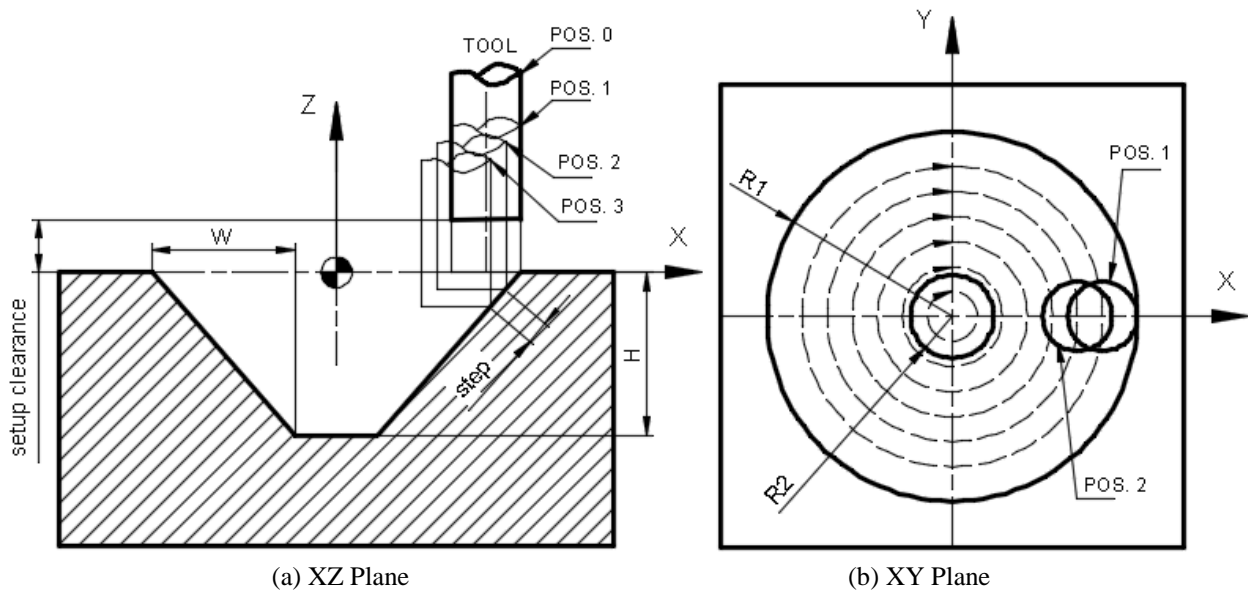


Fig.12: Tool path planning for a conical pocket

Process Plan

a. Tool Approach

With the zero reference point selected at the top center of the conical pocket, the tool is initially pre-positioned to starting position (POS. 0), defined by the following coordinates:

$$X = \text{top radius } R1 - \text{tool radius } r, Y = 0 \text{ mm}, Z = \text{setup clearance.}$$

From this position the tool is driven downwards to contact the top surface. (POS. 1)

b. Machining

The machining of the conical pocket is composed of two stages: Machining of the internal conical surface and removal of the residual material in cylindrical form (Fig. 13)

B.1. Machining of the Internal Conical Surface

The machining of the internal conical surface is achieved by repeating the following two successive motions:

1. Linear motion in XZ plane along a small segment of the cone generatrix (the two first motions are

shown in Fig. 12.a by consecutive positions POS1, POS2 and POS3). This motion is composed of two steps, one in X axis and one in Z axis which are executed simultaneously following the slope of the cone. The size of each step can be regulated by the user via a number stored in a parameter. This number divides both the depth H and the width W of the cone and the results are used as the respective sizes of the two steps in the axes X and Z respectively

2. Full circular motion in XY plane with the center of this motion at the center of the cone and radius the distance between the current tool position and the cone center (shown with dashed line in Fig. 12.b).

The two motions keep being repeated as the position of the tool, in the XZ plane, remains smaller than the depth H of the pocket. Reaching the bottom of the pocket at depth H, the sequence of the two motions stops and the tool is rapidly retracted to the initial setup clearance.

B.2. Removal of the Residual Material in Cylindrical Form

Since the residual material is in a cylindrical form, it can be considered as the internal material of a

circular pocket. Thus, its removal can be easily achieved by employing the standard circular pocket canned cycle which is available in all modern CNC milling machines. Practically, immediately after the end of the conical surface machining, the tool is driven at the top center of the residual cylinder and to the previously defined setup clearance. From this point the standard circular pocket canned cycle undertakes the

task of removing the residual material. The standard parameters of the circular pocket cycle are assigned with the current dimensions of the cylinder.

c. Tool Retraction

With the end of the circular pocket canned cycle the tool is rapidly retracted to a safe height.

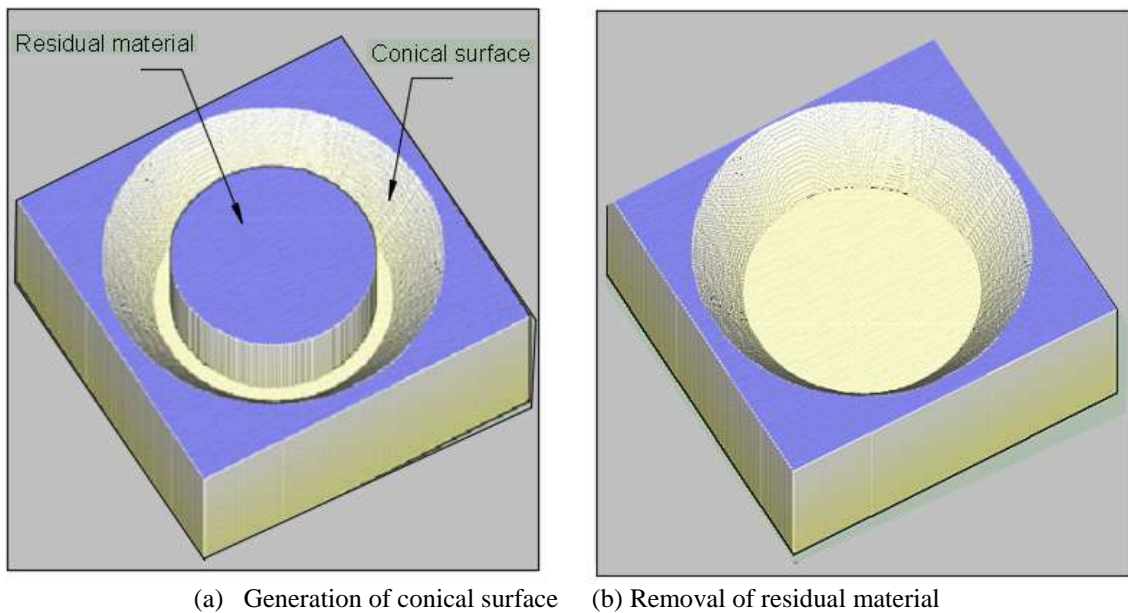


Fig. 13: Machining of a conical pocket in two stages

5.2 Tool Path Planning for Conical Boss (Fig. 14)

Using a flat end mill as the appropriate for the case tool, the machining of a conical pocket can be achieved according to the following process plan:

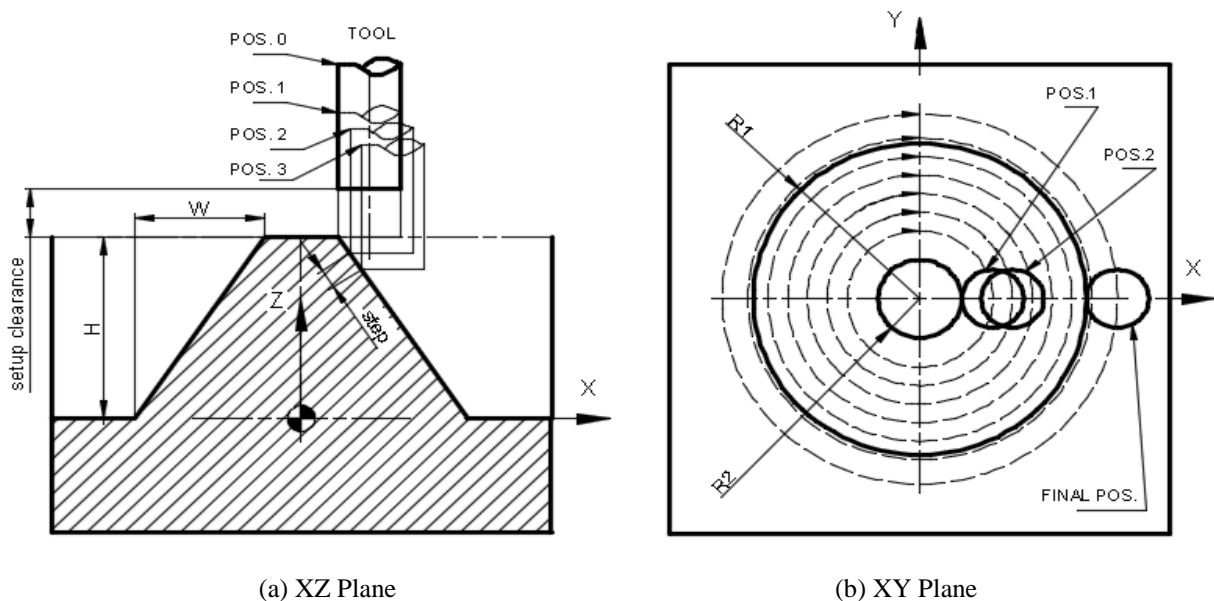


Fig.14: Tool path planning for a conical boss

Process Plan

a. Tool Approach

With the zero reference point selected at the bottom center of the conical boss, the tool is initially pre-positioned to starting position (POS. 0), defined by the following coordinates:

$$X = \text{top radius } R2 + \text{tool radius } r, \quad Y = 0$$

$$\text{mm, } Z = \text{setup clearance.}$$

From this position the tool is driven downwards to contact the top surface. (POS. 1)

b. Machining

The machining of the conical cavity is achieved by repeating the following two successive motions:

1. Linear motion in XZ plane along a small segment of the generatrix (the two first motions are shown in Fig. 14.a by consecutive positions POS1, POS2 and POS3). The size of the motion is calculated in the same way with the case of conical pocket described above.
2. Full circular motion in XY plane with the center of this motion at the center of the cone and radius the distance between the current tool position and the cone center (shown with dashed line in Fig. 14.b).

The two motions keep being repeated as the position of the tool, in the XZ plane, remains smaller than the depth H of the pocket.

c. Tool Retraction

Reaching the bottom of the boss at depth H, the sequence of the two motions stops and the tool is rapidly retracted to the initial setup clearance or to a safe height.

follows the end of the common flow, for the case of conical pocket.

For the needs of the simulation test, the required Q parameters included in the INPUT DATA box of the flowchart are assigned with the values shown in table 4.

Table 4: Definition of Q parameters – conical pocket/boss

Q parameter	Meaning	Assigned value
Q1	Tool radius	5 mm
Q2	R1	100 mm
Q3	R2	60 mm
Q4	H	80 mm
Q5	Precision factor	100
Q6	Current depth	0 mm
Q7	Setup clearance	4 mm
Q8	Feedrate	200 mm/min

The parametric G-code programs for a conical pocket and a conical boss are shown on the left side of figures 16 and 17 respectively. Both programs are consistent to the flowchart of Fig. 15 and according to the parameters of table 4. The result of the simulation test is shown on the right side of the same figures.

5.3 Parametric G-Code Programs and Test Results for Conical Pocket / Boss

The flowchart shown in Fig. 15 constitutes the common basis of developing the parametric G-code programs for machining any conical pocket or boss. The flowchart incorporates initialization of the CNC milling machine, definition of the required parameters and tool motion control as previously described in the process plans. The flowchart does not include the required circular canned cycle that

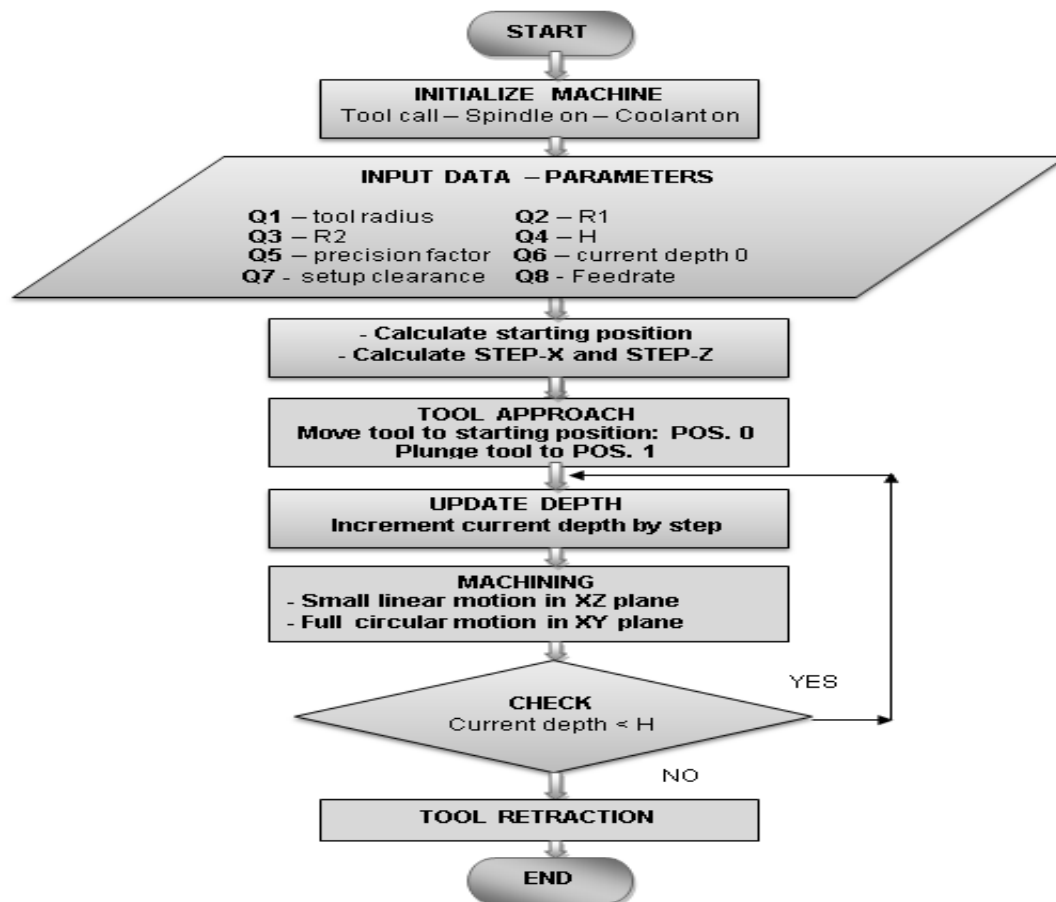


Fig.15: Flowchart of the conical pocket / boss parametric programs

Manual operation	Test run
<pre> %CONICAL_POCKET G71 * N10 G30 G17 X-110 Y-110 Z-100* N20 G31 X+110 Y+110 Z+0* N30 T1 G17 S2000* N40 M13* N50 D00 Q1 P01 +5* N60 D00 Q2 P01 +100* N70 D00 Q3 P01 +60* N80 D00 Q4 P01 +80* N90 D00 Q5 P01 +100* N100 D00 Q6 P01 +0* N110 D00 Q7 P01 +4* N120 D00 Q8 P01 +200* N130 D02 Q9 P01 +Q2 P02 +Q1* N140 D02 Q10 P01 +Q2 P02 +Q3* N150 D04 Q11 P01 +Q10 P02 +Q5* N160 D04 Q12 P01 +Q4 P02 +Q5* N170 G90 G00 X+Q9 Y+0 Z+Q7* N180 G01 Z+0 FQ8* N190 G98 L1* N200 G91 G01 X-Q11 Z-Q12* N210 G02 G91 X+0 Y+0 G90 I+0 J+0* N220 D01 Q6 P01 +Q6 P02 +Q12* N230 D12 P01 +Q6 P02 +Q4 P03 1* N240 G90 G00 Z+4* N250 G00 X+0 Y+0* N260 G77 P01 4 P02 -Q4 P03 5 P04 200 P05 Q3 P06 200* N270 M99* N280 G00 Z+100* N290 M30* N999999999 %CONICAL_POCKET G71 * </pre>	

Fig.16: Simulation test for a conical pocket - (Heidenhain control)

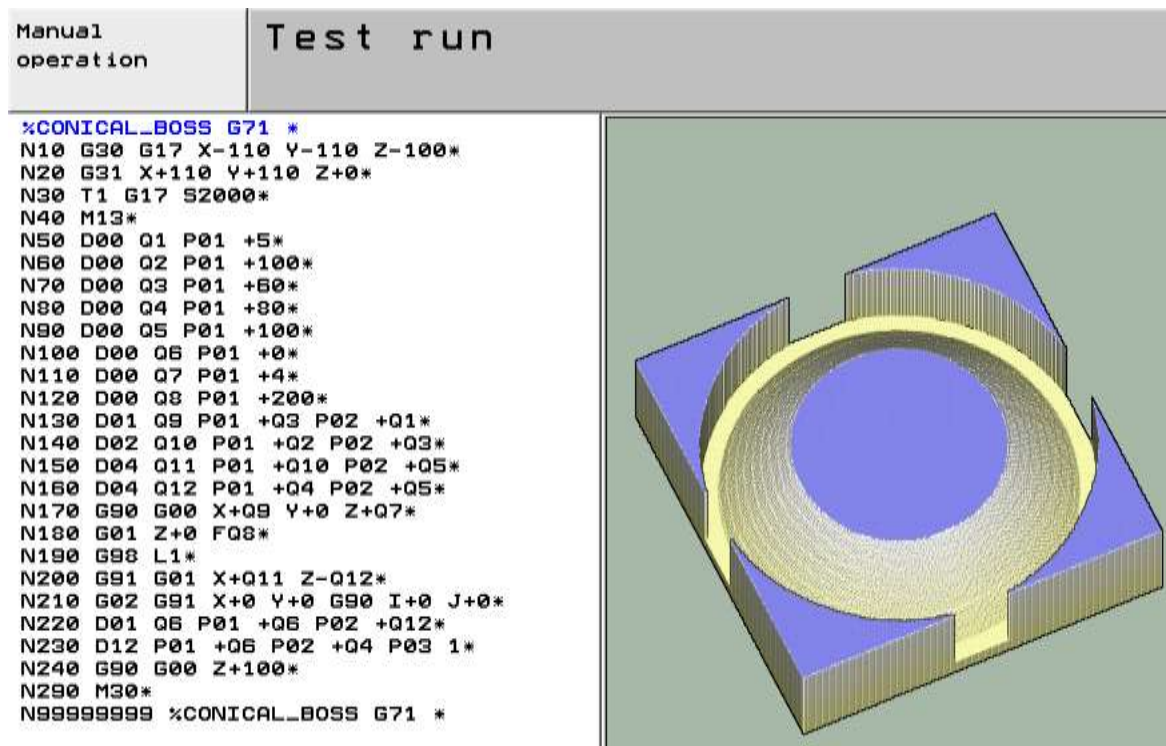


Fig.17: Simulation test for a conical boss - (Heidenhain control)

6. CONCLUSION

This paper presented a set of three new canned cycles that can extend the feature generating capabilities of CNC milling machines. Their development has been made possible by exploiting the possibility of parametric programming, an advanced CNC programming technique. The G-code parametric programs were developed in the environment of Heidenhain control system. However, they can be also easily translated in any other CNC control possessing the capability of parametric programming.

All programs are short in size since they do not exceed the number of thirty command lines each. The programs can be loaded in the storage device of the control system and called-up, as it stands for the standard canned cycles, each time a need is raised for machining one of the three pairs of the features described.

The parametric character of the programs offers them the capability to accommodate the machining of the three pairs of geometrical forms at any desired size, with the required in each case accuracy, the suitable tool size and optimum feedrate. The effectiveness of the proposed new canned cycles was verified by simulation tests for each separate case.

Certainly, there are more geometries than the presented in this paper that are useful in manufacturing needs and can be formulated parametrically as canned cycles. The methodology presented here can constitute a model for development of more canned cycles.

REFERENCES

- [1] Stenerson J. and Curran K., "Computer Numerical Control – Operation and Programming", Prentice Hall, 2005.
- [2] Zhiyang Yao and Satyandra K. Gupta, Cutter path generation for 2.5D milling by combining multiple different cutter path patterns, International Journal of Production Research, Vol. 42/11 pp. 2141–2161, 2004.
- [3] Park S.C and Chung Y.C., Offset tool-path linking for pocket machining, Computer-Aided Design, Volume 34, Issue 4, pp. 299-308; 2002.
- [4] Park S.C. and Choi B.K., Uncut free pocketing tool-paths generation using pair-wise offset algorithm, Computer-Aided Design, Volume 33, Issue 10, pp. 739-746; 2001.
- [5] Pateloup, V., Duc, E. and Ray, P., Corner optimization for pocket machining, International Journal of Machine Tools and Manufacture, 44 (12), pp.1343-1353; 2004.
- [6] D'Souza, R.M., Sequin, C. and Wright, P.K., Automated tool sequence selection for 3-axis machining of free-form pockets, Computer-Aided Design, 36 (7), pp. 595-605; 2004.
- [7] Mike Lynch, Parametric Programming for Computer Numerical Control Machine Tools and Touch Probes, Society of Manufacturing Engineers, 1997.
- [8] Sotiris L. Omirou. and Marios M. Fyrillas, A general G-code algorithm for deep hole drilling, Journal Paper Journal of Manufacturing Science and Production, Volume 15, Issue 2, pp. 225–237, 2015.

- [9] Sotiris L. Omirou, A CNC Parametric Programming Method for Manufacturing of Axisymmetric Mould Cavities, Journal Paper Journal of Manufacturing Science and Production, Volume 16, Issue 3, pp. 173-181, 2016.
- [10] Phung V. Application of Variable Parametric Programming in Precision Machining Processes. In: Seyfried P., Kunzmann H., McKeown P., Weck M. (eds) Progress in Precision Engineering, pp. 290-298, Springer, Berlin, Heidelberg, 1991
- [11] Heidenhain iTNC 530 User's Manual.
- [12] S. K. Sinha, CNC Programming Using Fanuc Custom Macro B, The McGraw-Hill Companies, Inc., 2010.
- [13] Mohd Asif Hasan, Computer Numerical Control Machines: An Introduction to Parametric Programming using Custom -Macro B, Journal of Basic and Applied Engineering Research, Vol.2, pp. 1566-1569, 2015.

BIOGRAPHIE



Dr. Sotiris Omirou is an Associate professor at the Department of Mechanical Engineering of Frederick University in Cyprus. He has over 30 years teaching experience in tertiary education. His research interests are focused on CAD/CAM and CNC manufacturing methods. He has published a significant number of scientific papers in international peer-reviewed journals and he is the author of two technical books.