Development Of A Software Module For Nc Programming In Industrial Environment

Mohamed Rahou[1], Fethi Sebaa[2]

[1](Higher School of Applied Sciences of Tlemcen, IS2M Laboratory, Algeria)
[2](Department of mechanical, IS2M Laboratory, University of Tlemcen, Algeria)

Abstract

In this paper, we developed a module that helps NC programming instructions machining standard in the CAM environment. This module allows learning to numerically as well as conversational programming mode. Two parties have developed. The first part deals with the incompatibility of functions preparatory commands such as FANUC, FAGOR, NUM, SINUMERIK, shooting, and milling. The second part deals with developing a module contributing to the adaptation of standard machining instructions in the CAM environment.

Keywords - *NC, CAM, Programing*

I. INTRODUCTION

The programming of the CNC is based on standard programming languages. It turns out that these standards are not complete. The directors of CNC (DNC) to adapt the specifications of their machines. The criteria describe programming languages , also called commands. Each builder DNC was trying, by all means, to differentiate its products from those of competition and develop the standard programming languages [1].

The non-uniformity of the language used creates confusion in preparing programs of different commands, leading to reluctance and inaction often when learning or programming of numerical control, especially in the case of a machine park varied languages, installing and doubt.

A set of tutorials enabling the discovery and language learning machine was developed, highlighting some functions preparatory core and CNC technology. We can quote EMCO, DENFORD, PROCAM, Tour Assistance, the ARDEM (Association for Research and Development Multimedia Computer Education). Which developed three tutorials (CONCEPT CN CN DIDA, IPMO), the association MECAPASSION, DS, SOLIDCONCEPTER software provides several commands, using the operator command is equivalent to choosing the post desired processor. Orders and REALMECA FAGOR offer intuitive tools to machines, using a group technology based on statements of form (dot machining operations). Other work has developing to provide the learner the basics of CNC programming but limited to one language [2,3].

This work aims to study the incompatibility of the most used commands NC and develop a teaching tool to support NC programming conversationally machining instructions in a CAM.

II. INCOMPATIBILITIES OF ADDRESS

The most common standard is the ISO standard (ISO 840), which defines the alphabet based on the ASCII code and additional measures that restrict the programming format (ISO 1056,1057,1058,1059)[4].

Table 1 shows some differences in codes with two names for the same code [5,6].

Table 1. Codes incompatibility based on a designation

Code	Turning	Milling
G76	Threading cycle	Bore
G90	Removal cycle	absolute programming
G92	Threading cycle	absolute programming
G94	Face turning cycle	Feed (minute)
G98	Feed (minute)	Return to the starting point

Some manufacturers of DNC (FANUC, FAGOR ...) use the same code in turning and milling for two different designations, such as for Fanuc and different codes for the same designation. These differences are even more pronounced in the case of FAGOR. To the SINUMERIK, the same codes are generally used for the same designations. As for the NUM control, using the ISO code, there have been no differences in designation for the same code.

Except for preparatory functions and auxiliary functions, the result after the statistical study of these addresses the following [7]:

- 37.50% use the same designation
- 25% use two names
- 08.33% use three designations
- 08.33% use four designations
- 08.33% use five names (addresses Q, H)
- 04.16% use seven nominations (address P)
- 04.16% use eight nominations (Address R)
- 04.16% use nine nominations (e-K)

Faced with a likely lack of consultation and fierce business competition, it continues to see a non-uniformity of language difficulties caused by the

programming of CNC. Hence the importance of developing a tool for NC programming for different orders.

III. DEVELOPED MODEL

To overcome the problems posed by these inconsistencies, a teaching module contributing to the adaptation of CNC machining instructions FANUC controls [12], SINUMERIK [9], NUM [10], FAGOR [11] was developed using the Visual Basic version 6.0 and a video editing software to create video files corresponding to machining operations.

In addition to using the program, two applications of machining turning and milling are visualized by video sequences corresponding to each block of the proposed program. This will also incorporate multimedia in the teaching of digital control. With the multimedia software used, the computer becomes a complete system for acquiring and editing video, without being forced to use specialized equipment and expensive.

The creation of videos is a three-step process:

i. Capture (acquisition) is the video recording on the hard disk.

ii. Edited (assembly): the classification of video scenes in the desired order, for example, the first reference block of the original machine, the second block tool change, and sound effects can add comments in the film.

iii. Create the film is the creation of the final video or video files.

A. Key features

This module has three main functions:

- Identification codes with writing syntax
- Search for code designations
- Applications in videos (turning and milling)

For the shooting, several sequences of machining operations were recorded in video files. These highlight machining cycles programmed with FANUC control cycles such as Turning-draft G71, G70 Turning-finishing, threading G76, G74 peck drilling, and the different linear and circular interpolations.

For milling, it was shown the zero offset, programming absolute or relative, the different linear and circular interpolations, and clearances of tools.

B. Identification of a designation

The first function allows identifying the designation of the preparatory function code G after choosing the command/operation and the desired G code.

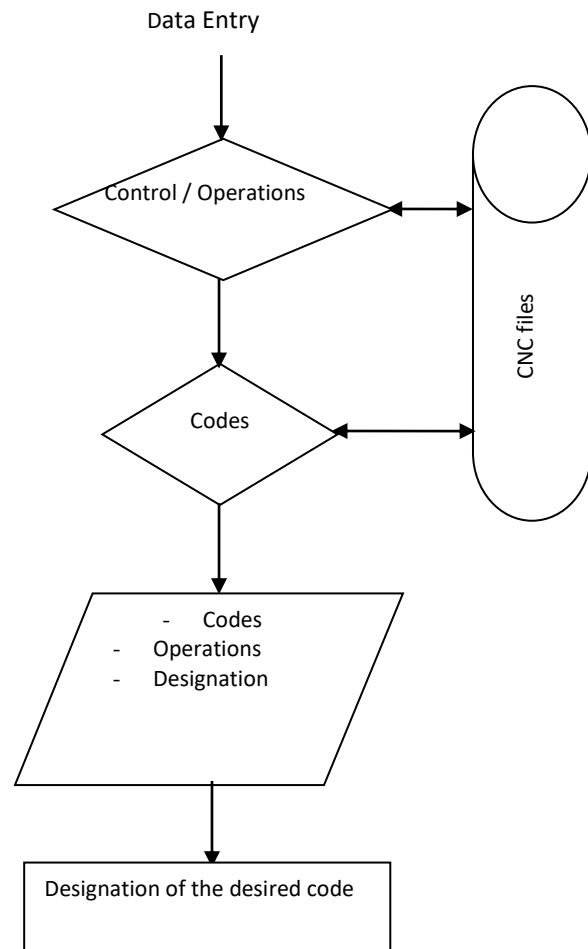The procedure of this function is illustrated by the steps of Figure 1 and Figure 2.

Fig.1. Mode operation of the first function

In case the code has not sought designation, the term "EXIST" is displayed in the text field reserved for the description of the code sought.
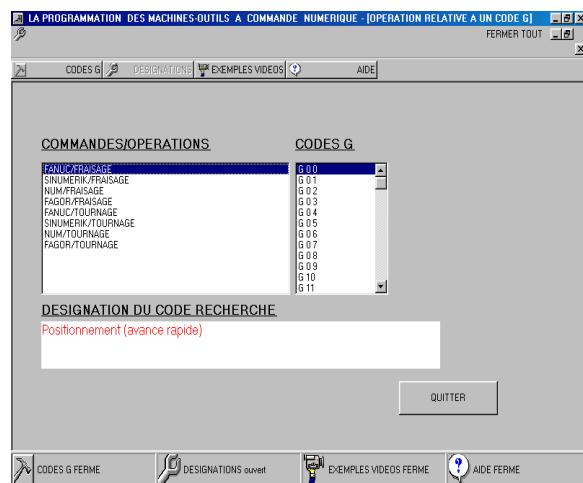

Fig.2.Identification designation

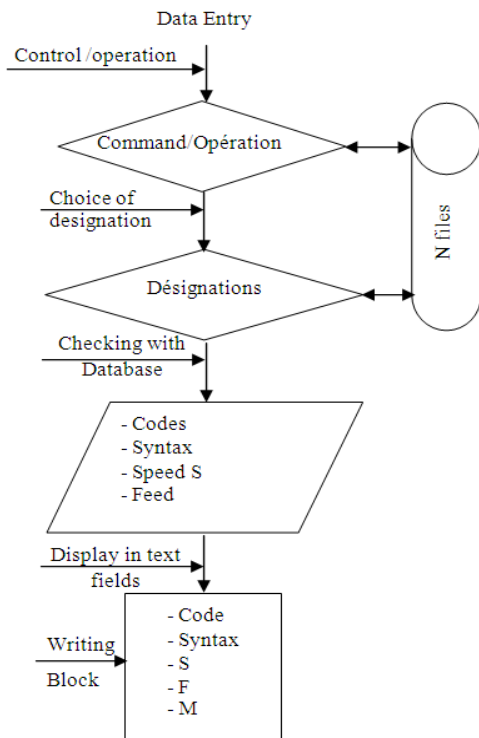*C. Identification code with syntax.*

Fig.3.Mode operation of the second function

The second function allows the identification of the syntax of a program G-code from the desired name, and the command/operation is illustrated by the steps of Figure 3 and Figure 4 display.

Programming with Visual Basic 6.0 software is sequential programming, advancing step by step. The program should check all the parameters from the files of the database. If, after reading a parameter of the syntax of the code looking for does not exist in the file, the text field will be closed to the programmer. The latter cannot enter data. By cons in the case of M functions, the corresponding text field is still open. The screen of Figure 4 shows the closure of the text field reserved for S because it cannot be programmed to rotate in the same block in the case of linear interpolation G01.
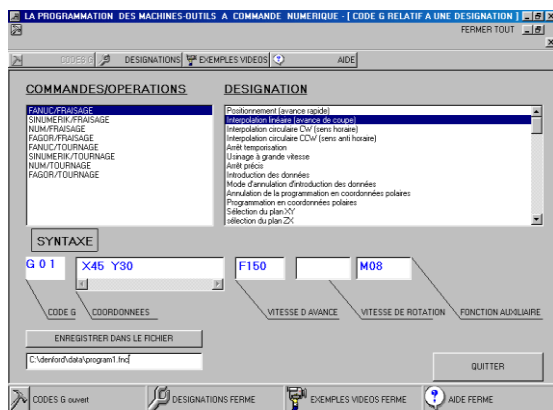

Fig.4. Identification of a G-code

After writing the code and syntax, it is asked to introduce the coordinates of the point to be reached, the value of the feed rate, and the corresponding auxiliary function, as shown in the screen of fig 4. Once the block of writing, it is possible to save a file (C: \ .... \ Data \ file name) belonging to a CAM software by clicking after the end of each block on the Save button in the FILE.

After you click save the file, the block in question is directly inserted into a pre-established in CAD-CAM software using the command FANUC

O 0001 (program number)
G21 G91 G28 X0 Y0 Z0 / U0 W0 ;
M06 T01;
G 92 X… Y… Z… ;
M03 S… ;
G00 G90 X Y / X Z ;
G 01 X45 Y30 F150   M08 ;
G28 G91 X0 Y0 Z0 / U0 W0 ;
M05 M30 ;

D. Video examples

The screen of Figure 5 shows two examples of program turning and milling. Just click on the example so that the program wanted to view is displayed in the BLOCK (BLOCK).

Each block in the program year is the video file. In what follows, we limit ourselves to present only a few sequences of programs for turning and milling.

- Example programming milling

The screen of Figure 5 also has the original machine tool position (MO) after using the cutter tool magazine, corresponding to the block N001 of the proposed milling program.
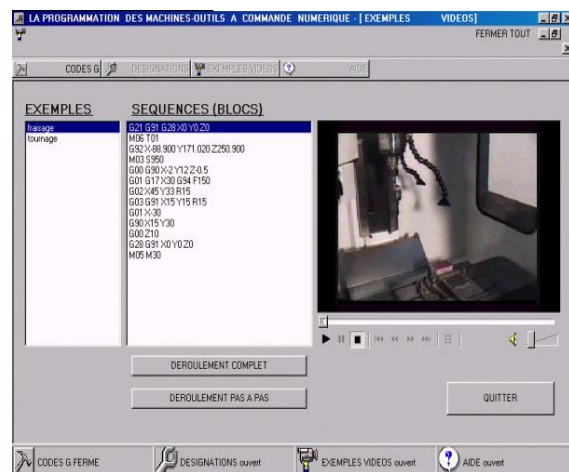

Fig.5. Program block N001 milling machine zero position

The screen of Figure 6 shows a linear interpolation with programming for corresponding to the block N009.
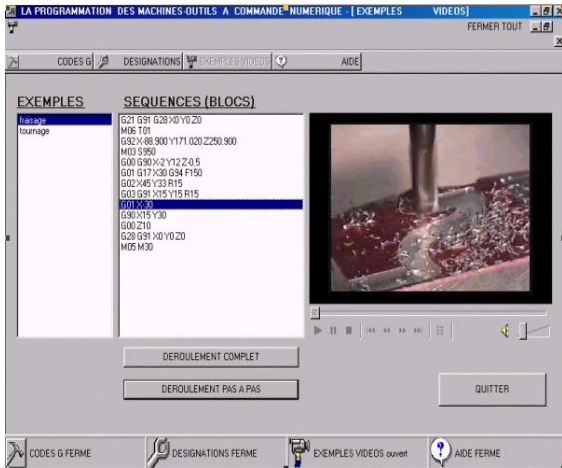
Fig. 6. Block N009 program programming on linear interpolation

- *Programming example for turning*

The screen of Figure 7 shows the original position machine tool (MO) after resetting the machine (reset) and catch the reference.
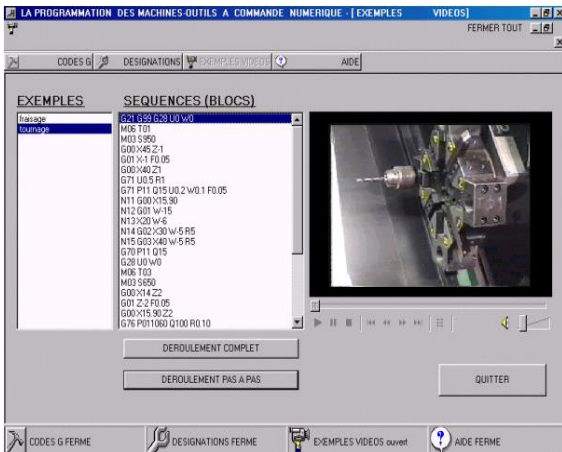

Fig. 7. N001 original block machine

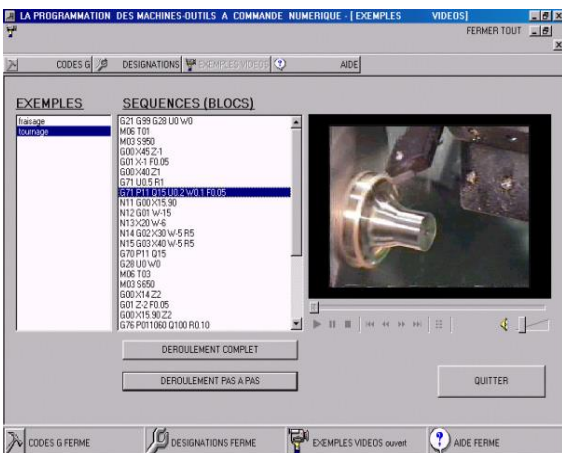The screens 8 to 10 are machining cycles shooting.
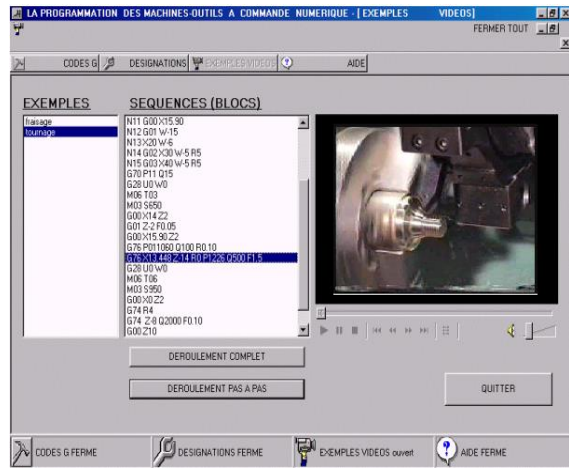

Fig. 8. N008 block removal cycle-draft

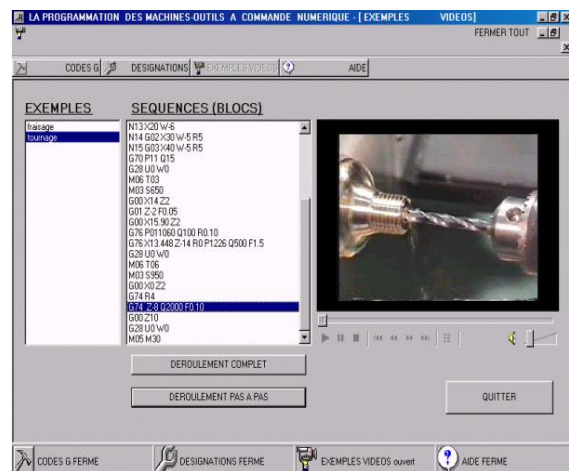**Fig. 9. N022 block threading cycle**

Fig.10. N28 block drilling peck

IV. CONCLUSION

In this work, the goal is to develop a tool for programming to teach or learn NC, especially in a machine park varied languages and avoiding dependence on a single manufacturer or technology.

In addition to the presentation of the three functions of the module, multimedia software was used for the preparation of two video films of turning and milling applications.

It is also possible to call only one video file to explain a sequence or block release as a tool, a tool change, an interpolation, a machining cycle. The integration of multimedia in training and learning of the CNC will facilitate the explanation and understanding of different directions and machining cycles and the preparation of the workplace by the determination of origin.

REFERENCES

[1] C. MARTY, C. CASSAGNES, P. MARIN, septembre 1993, " *la Pratique de la Commande Numérique des Machines Outils* ", Editions LAVOISIER, Paris.

[2] SEBAA F, CHEIKH A, " *Module d'adaptation des instructions d'usinage des MOCN* "JEM 03,Annaba ,2003.

[3] FANUC, 1995, " *Manuels de tournage* "B62634/02 et fraisage B62644/02, FANUC GE LTD, Tokyo.

[4] EMCO WIN, 2003, *"Manuel de programmation sinumerik "*810/820 (DIN 66025), EMCO, Autriche.

[5] J-P. URSO, mars 1999, " *Commande Numérique Programmation"*, Editions CASTEILLA, Paris.

[6] FAGOR, 1998, *" Manuels techniques de tournage et fraisage "*8025 / 8030 Ref (9701), FAGOR automation, France.

[7] SEBAA. F., RAHOU M. , CHEIKH A., *" Contribution à la programmation des adresses CN pour différents langages dans un environement FAO"*, 9ème Congrés de Mécanique, Marrakech, Maroc,2009.

[8] Rakesh Prajapati , Avadhoot Rajurkar , Vijay kumar Chaudhary. *"Tool Path Optimization of Contouring Operation and Machining Strategies for Turbo Machinery Blades".* International Journal of Engineering Trends and Technology (IJETT). V4(5):1731-1737 May 2013.

[9] F. SEBAA, M. RAHOU, *" Statistical Study of NC Address"*, Mechanical and mechanics engineering, Vol 13, No 3-J (2013) Online ISSN: 2249-4596 Print ISSN: 0975-5861.

[10] M. RAHOU, F. SEBAA *," Development of a Tool fo Programming the Machining Instructions in a CAM Environment"*, Mechanical and mechanics engineering, Vol 13, No 4-J (2013) , Online ISSN: 2249-4596 Print ISSN: 0975-5861.

[11] M. RAHOU, F. SEBAA, A. CHEIKH *" Comparative study between nc programs "*Istanbul international conference on progres in applied science 2017 – 4 - 6 january 2017

[12] M. RAHOU, F. SEBAA, A. CHEIKH, F. RAHOU , *" A Tool for Programming CNC Machining "*, ASME, DOI: 10.1115/1.859902.paper221 , 2011,ISBN 9780791859902