Intelligent Generation of a STEP-NC Program for Machining Prismatic Workpiece

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Abstract. Advances in CNC technology, coupled with improvements in computer systems provided the basis to review how computer-based systems can be used to allow universal fabrication. The G code no longer meets the current expectations of the program ever modern and complex machines. Indeed, the standard STEP-NC (STEP-Numerical Control) offers new solutions and integration of manufacturing in the full numerical chain. In this paper, we propose a new numerical control based on a standard high conceptual level STEP-NC machining process for prismatic parts, which aims to develop intelligent and powerful tools for sharing and data integration in CAD/CAM systems.

Keywords: STEP-NC, ISO-14649, G-code, CAD/CAM, CNC.

1 Introduction

Currently, a large number of developments have affected the field of mechanical production, the emergence of new CNC machines to the most advanced flexible machining cells, through the representation of new manufacturing processes. Indeed, machine tools and CNC will face new challenges of productivity, flexibility, interoperability, scalability and portability. The evolution of technology is in all components of the manufacturing cell and the CNC to the tool, but also by its full integration in CAD/CAM systems. Today, a new data standard known as STEP-NC [ISO 14649-10, 2004], which is to define non-complex way all the data of a product during its life cycle. In addition, it provides full interoperability and portability of this data to be interpreted by any type of computer system. The objective of this research is the development of a new digital channel based on a standard high conceptual level STEP-NC, used in the machining process for prismatic parts. The aim of this subject is to develop intelligent and powerful tools for sharing and integrating data into numerical chain of CAD-CAMCNC [Laguionie et al, 2009]. This work begins with the presentation of the STEP-NC program and the benefits over the conventional program code G. Then a proposal of the architecture of the model developed of generation of STEP-NC. The implementation of

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this model in the integrated design environment is developed by object-oriented programming, using the VBA CATIA interface. However, the validation of the results of the model developed is through case studies of prismatic work piece, and the compilation and verification of data programs of STEP-NC generated for all cases by STEPTOOLS, simulation of machining was performed with CATIA V5.

2 Programming STEP-NC

2.1 STEP-NC

The STEP-NC is a standard for data exchange for the G-code [Shin et al, 2007]. It is based on the STEP standard (Standard for the Exchange of Product data). It includes both the properties of STEP; integration of the design process, development, manufacture and maintenance during all life stages of the object and high level information for the manufacture of product. The standard STEP-NC is standardized through ISO 14649 (MRA: Application Reference Model) and the 10303 -AP238 application protocol (AIM: Application Interpreted Model). The ARM to define the data model corresponding to advanced users formalization of data to the user vocabulary. This modeling can be performed by EXPRESS-G (Fig. 1). STEP-NC is a new standard that has emerged in order to overcome the shortcomings of the current digital channel with regard to the production of CNC machine tool.

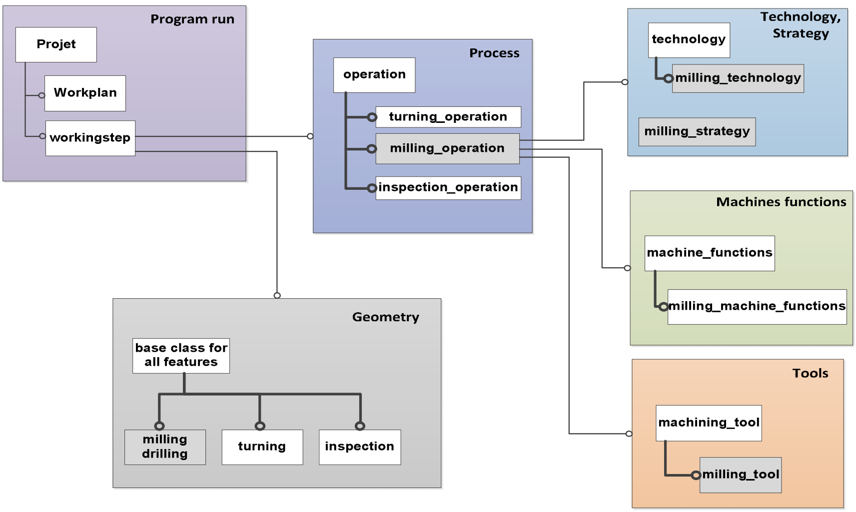


Fig. 1 EXPRESS-G model for STEP-NC

2.2 Comparison of G-code and STEP-NC

The increased productivity on CNC machine involves improving programming thereof. The language which is currently based programming that dates from the early 80s with the 6983 standard, which defines the principles of the G code programming this program joins a path in accordance with the movement of machine tool axes, rather than to focus on the needs of the machining in accordance with the workpiece. It is programmed by auxiliary functions (M-type) and technological (F, S, etc.). But it has certain inconvenience to the views of the new machining strategies and creates a break in the numerical chain at the manufacture. Among these drawbacks, the semantics can sometimes be blurred and manufacturers sometimes add language extensions to fill the gaps and adapt to changing technologies and the portability of a program then proves not between different manufacturers. There are also the flow of information is unidirectional: the lack of feedback (Feedback) possible production design causes difficulties of communication and correction [Xu, 2006]. Similarly, the preservation and capitalization of experiences prove complicated (Fig. 2).

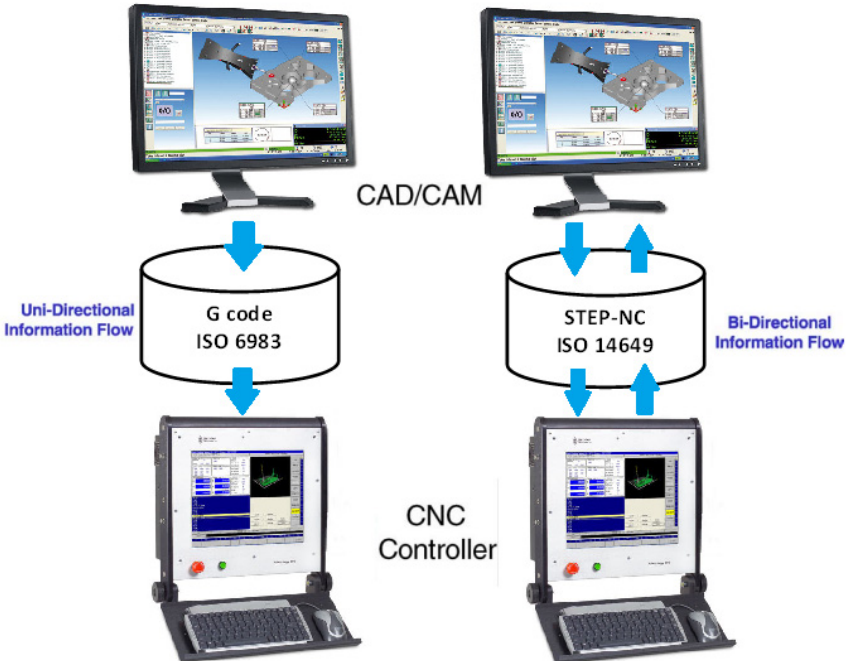


Fig. 2 Bi-directional information flow with STEP-NC

The STEP-NC [Rauch, 2007] [Rauch et al, 2012] reduces machining time for parts small and medium series due to intelligent optimization, also, it can be integrated into the controllers. Indeed, in this new programming, the post-processor will be eliminated as the interface requires no information specific to the machine, so the STEP-NC file is transportable from one machine to another without the need for adaptation, because it predominantly contains generic information treatable by all interpreters.

2.3 Structure of STEP-NC

The contents of the ISO 14649 [ISO 14649 Part 11, 2004] information consist of: task description, description of the technology, a description of the tool, and description of the geometry. The job description describes the logical sequence of executable tasks and data types. Details of each "working step" are covered in the description of the technology in reference to the description of the tool and the description of the geometry. A graphical summary of the information available in the STEP-NC program Express-G [Newmen et al, 2009] is shown in Fig. 3 below:

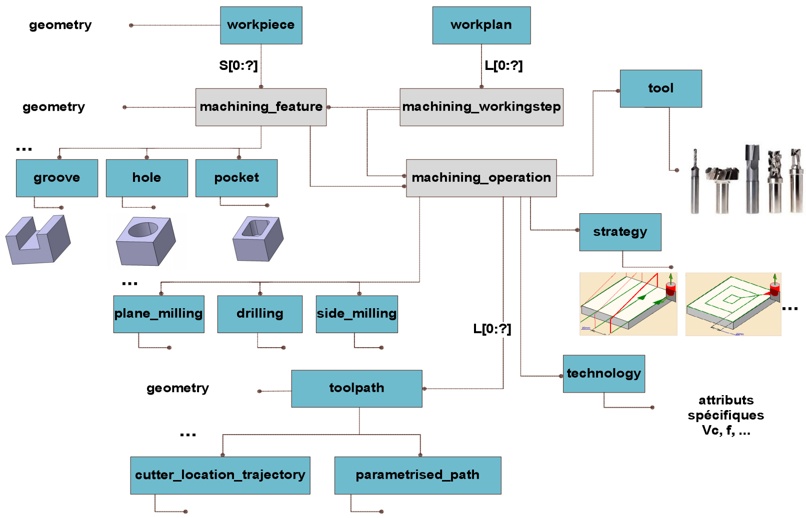


Fig. 3 Geometry, technology and process information in STEP-NC

All of these part data is broken down into three parts:

- ✓ A header section (Header) on general program presentation information (author, date, my tumble... etc.).
- ✓ Information of geometry, which concern the machining features.
- ✓ Information for machining: for each operation on a processing entity, the STEP-NC file defines the set of information such as the type of operation, the tools, the cutting parameters and machining strategies.

Fig. 3 shows part of the internal structure of STEP-NC data. The exert from a STEP-NC file **[ISO10303-21]** is shown as follows:

```
ISO-10303-21;
HEADER;
FILE DESCRIPTION((‘A STEP-NC testing file’),’1’);
FILE NAME(‘sample part1.stp’,$,(‘AUMS’),(‘‘), ’Prototype Mill’,’ ’,’ ’);
FILE SCHEMA((‘STEP-NC milling schema’));
ENDSEC;
DATA;
// Project and workplan
#1=PROJECT(‘Contour’,#2,(#3));
#2=WORKPLAN(‘Work plan’,(#4),$,#5);
#3=WORKPIECE(‘Workpiece’,#6,0.01,$,$,#8,());
// Working steps
#4=MACHINING WORKINGSTEP(‘Rough Contour’,#13,#16,#17);
#5=SETUP(‘main setup’,#44,#48,(#51));
#6=MATERIAL(‘ST-50’,’Steel’,(#7));
#7=PROPERTY PARAMETER(‘E=200000 N/mm^2’);
#8=BLOCK(‘Block’,#9,260.000,210.000,110.000);
// Geometric data
#9=AXIS2 PLACEMENT 3D (‘BLOCK’,#10,#11,#12);
.....
// Manufacturing features
#16=GENERAL OUTSIDE PROFILE (‘Profile’,#3,(#17),#18,#22,$,$,$,$,#23,$,$);
// Operation data
#17=SIDE ROUGH MILLING($,$,’Contour profile’,#38,10.000,#39,#40,#43,$,$,$,20.000,5.000,0.000);
#18=AXIS2 PLACEMENT 3D(‘Position of contour’,
#19,#20,#21);
#19=CARTESIAN POINT(‘Position of contour’,(40.000,90.000,100.000));
#20=DIRECTION(‘‘,(0.0,0.0,1.0));
#21=DIRECTION(‘‘,(1.0,0.0,0.0));
#22=TOLERANCED LENGTH MEASURE(20.000,$,$,$);
#23=COMPOSITE CURVE (‘Contour Profile’,(#24,#25,#56),.F.);
.....
// Tool data
#40=CUTTING TOOL(‘End mill 10 mm’,#41,(),(50.000),50.000);
#41=TAPERED ENDMILL(#42,3,.RIGHT.,.F.,$,$);
#42=TOOL DIMENSION(10.000,$,$,$,$,$,$);
// Machining technology
#43=MILLING TECHNOLOGY($,.TCP.,$,3.3333,$,0.10,.T.,.F.,.F.);
#44=AXIS2 PLACEMENT 3D(‘Reference point to Machine zero,#45,#46,#47);
#45=CARTESIAN POINT (‘‘,(20.000,30.000,10.000));
.....
#56=COMPOSITE CURVE SEGMENT (.CONTINUOUS.,.T.,#57);
#57=POLYLINE(‘Second cut of the contour’, (#29,#30,#31,#32,#33,#27));
ENDSEC;
END-ISO-10303-21;
```

Fig. 4 Example STEP-NC physical file

3 Creation of the Interface of STEP-NC Module and Validation

In programming STEP-NC manufacturing parameters such as cutting conditions (cutting speed, feed rate... etc.) [Cuenca et al, 2011] are given by the user in the program or on the machine to use, since the one of its advantages is compatible with all CNC machine tools that contain a STEP-NC processor. This programming is required of all production data for used and ordered according to the relationships between machining features and raw part with the diagram STEP-NC. The flow of work in the generation system developed STEP-NC [Haddad et al, 2014] is followed by the process and the structure of the chart of Fig. 5:

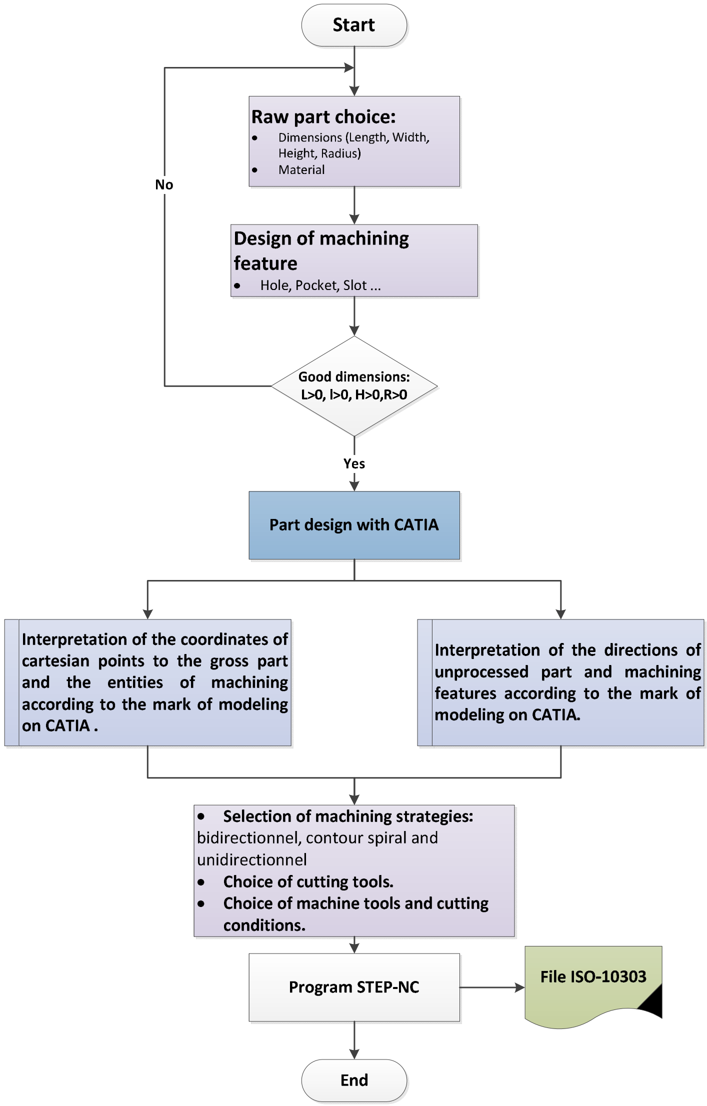


Fig. 5 Flowchart program for development STEP-NC from the part data

4 Steps of the Model Developed

The generation of the model developed STEP-NC consists of several modules to generate the STEP-NC program:

- A modeling module and rough material.
- A modeling module of machining features and automatic creation of finished part CATIA.
- A module to interpret the directions of orientation of the part in the same frame.
- A module to display the STEP-NC program.
- A simulation module and validation of STEP-NC CATIA.

The relationships between these modules are defined in Fig. 6. These modules are integrated in a database. Using these modules, the flow of work in the generation of the model developed STEP-NC is followed by the algorithm in Fig. 5.

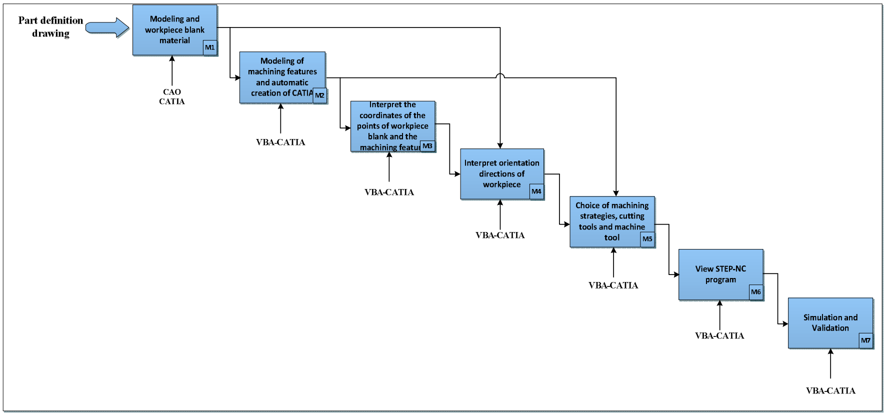


Fig. 6 Architectural Overview of the generation system of STEP-NC

5 Case Study

The application we have proposed to validate the developed generation computer system STEP-NC, is a workpiece containing single prismatic machining features of type " hole ", Pocket’’ and " Slot '. The following table shows the characteristics of this feature and the associated blank.

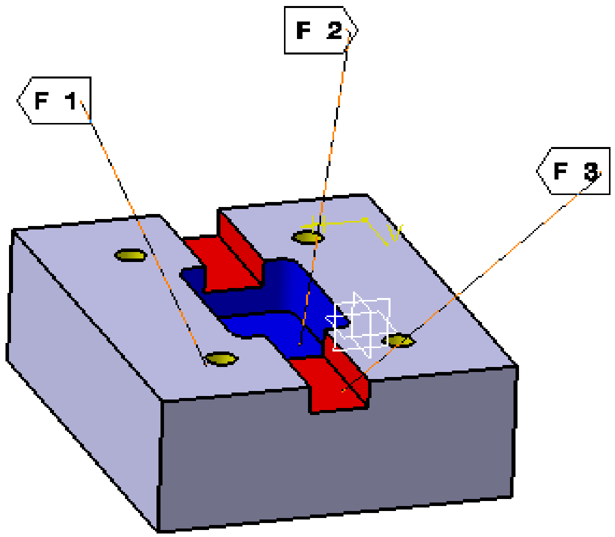


Fig. 7 Example of prismatic workpiece

Table 1 The characteristics of machining units and the blank

	Dimension	Tolerance	Cutting tool	Machine tool
F 1	Hole diameter 20 mm and 100 mm deep	±0.1	Drill (Ø20 mm)	3-axis milling machine
F 2	Pocket width 40 mm and 10 mm deep	±0.2	Pocket cutter (Ø10 mm)	3-axis milling machine
F 3	Slot width 80 mm and 30 mm deep	±0.2	Slot cutter ARS (Ø20 mm)	3-axis milling machine

When starting the computer application displays the menu in Fig. 8 for part modeling with the choice of its form (prismatic or cylindrical), its dimensions and material.

After the creation of the piece on CATIA in proceeds to the next step in the interpretation of the coordinates of the nodes and vertices of the blank and entities "Hole" “Pocket”, and "Slot". And in the menu of Fig. 10, the position is interpreted by the workpiece origin and orientation according to the directions.

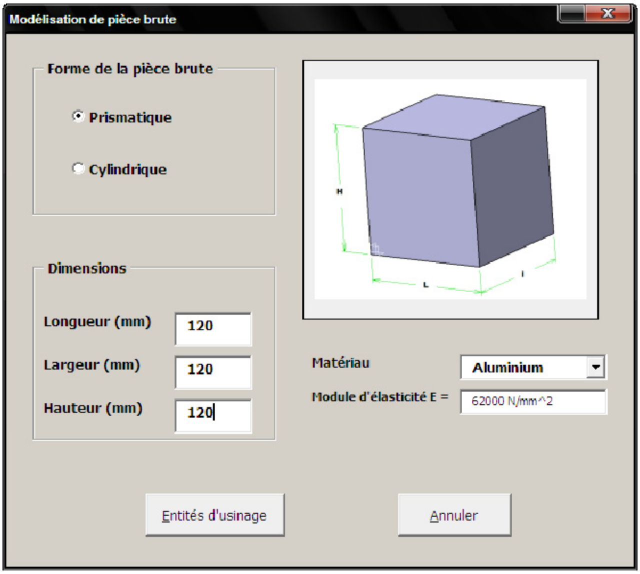
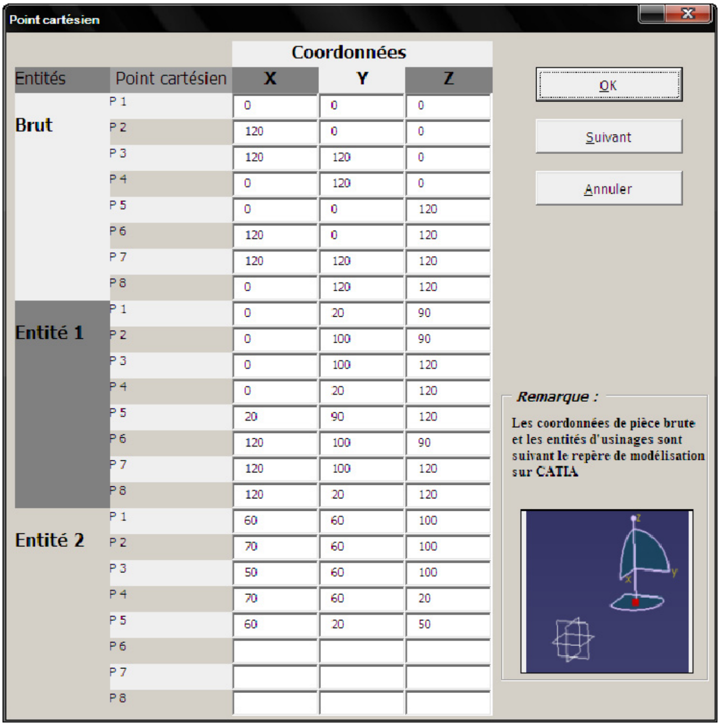


Fig. 8 Raw part modeling



Entités	Point cartésien	Coordonnées X	Y	Z
Brut	P 1	0	0	0
	P 2	120	0	0
	P 3	120	120	0
	P 4	0	120	0
	P 5	0	0	120
	P 6	120	0	120
	P 7	120	120	120
	P 8	0	120	120
Entité 1	P 1	0	20	90
	P 2	0	100	90
	P 3	0	100	120
	P 4	0	20	120
	P 5	20	90	120
	P 6	120	100	90
	P 7	120	100	120
	P 8	120	20	120
Entité 2	P 1	60	60	100
	P 2	70	60	100
	P 3	50	60	100
	P 4	70	60	20
	P 5	60	20	50
	P 6			
	P 7			
	P 8			

Fig. 9 Interpretation of the coordinates of the nodes for raw part and machining features

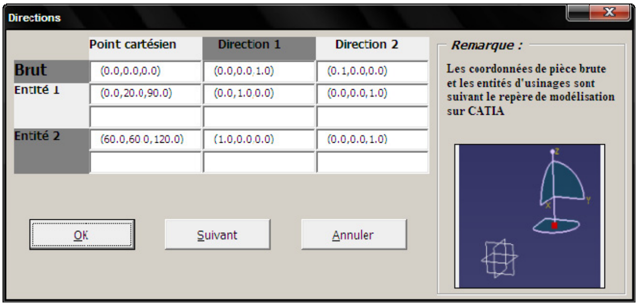


Fig. 10 Interpretation of the directions of orientation for raw part and machining features

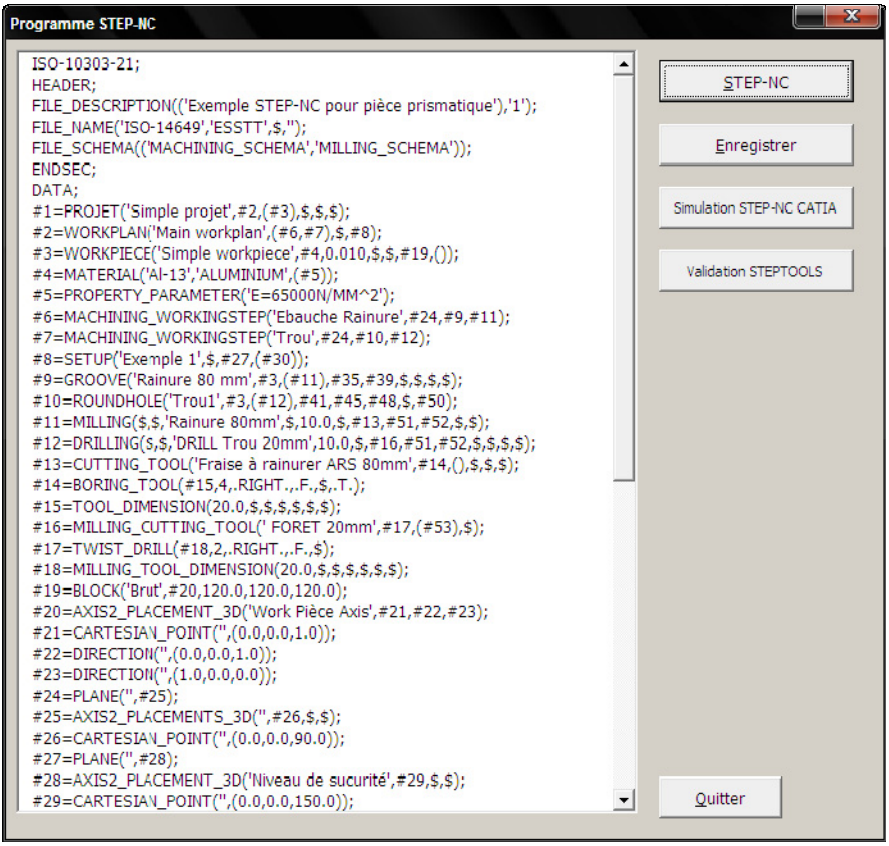


Fig. 11 Generated of STEP-NC (ISO 14649) part program

The application program result is displayed in a menu as shown in Fig. 11, which can be saved as a STEP file [ISO 10303-21]. Then we made the validation of our program by a compiler STEPTOOLS as shown in the following Fig. 12.

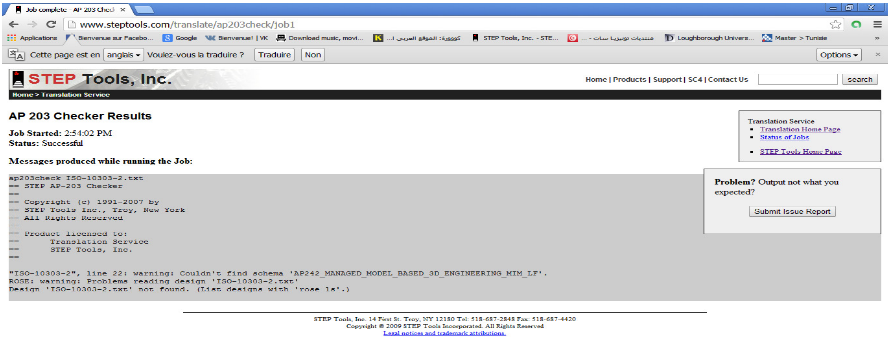


Fig. 12 Validation by STEPTOOLS compiler

6 Conclusion

This work is concerned with the integration of the entities based programming for processing on CNC machines in a context multi process. Programming STEP-NC can fill the skips of G-code and fully integrate CAD-CAM-CNC system. Indeed, in this paper we propose a development approach of STEP-NC allows manufacturers to use the new generation of machine tools based on STEP NC programming. The STEP-NC uses a high-level language and is based on the implementation of a single file. It allows two-way flow of data and the eradication of post-processors and code-G. However, the programming method proposed STEP-NC is carried out with CATIA VBA module. This intelligent programming method allows using workpiece data and knowledge of machining experts. The implementation of this module was developed with programming VBA, EXPRESS-G and CATIA.

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