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STEP-NC: A New Programming Code for the CNC Machines

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The paper deals with the newest advances in standardization of manufacturing data management. The overview of previous work and projects and the latest developments on introducing the ISO 10303-238 and ISO 14649 standards is presented. The advantages of the new data models are shown and the plans on future work are presented.

Keywords:mnufacturing machines, CNC programming, ISO standardsg

0 INTRODUCTION

Ever since a CNC machine appeared on the market people have followed a dream of automatically manufacturing products. In the last 50 years CNC machines have evolved from simple, perforated paper belt controlled machines to today's high-end machines controlled by multiprocessor workstations. These modern controls are able to control very complex processes like 5-axis toolpaths, multitasking machines etc. Apart from this another big leap has happened. The CAD, CAPP and CAM software vendors realized that they need to have a standardized and open format to transfer information in the process of product and manufacturing planning.

Even though the CNC machines evolved substantially, the language to program them has not. Most of the CNC machines on this planet are still controlled by using a G-code format of NC program, which is standardized by an ISO 6983 from 1983.

During 1980s many different data formats for geometry transfer were proposed (SET, VDA, IGES) but none of them satisfied all the needs of software developers and users [4]. In the mid 1980s the international community started to develop the ISO 10303 group of standards which are today known under the name STEP. This standard is today widely recognized and accepted as de facto standard for geometry data transfer in the process of product design and manufacturing. The specification is open for all users and the format is alive, which means that it is adapting to new ascertaining of the field of CAD model data transfer.

In the field of CNC data transfer similar work started in the mid 1990s. Many different institutions all over the world have contributed to the development of a new standard to support seamless integration of CAD, CAM and CNC machines in the manufacturing process. The result of this work is the adoption of a new standard ISO 10303-238 in 2007.

1 CURRENT STATE IN THE FIELD OF CNC PROGRAMMING

Fig. 1 shows the current state of manufacturing data flow. Currently, most of CNC machines are programmed by some dialect of the NC language (ISO 6938). Programming with this language brings some advantages and many disadvantages.

Advantages are [7]:

- the language is very simple, easy to learn and understand,
- it is widespread and used all over the world.

With its simplicity it brings many disadvantages [3]

- long NC programs for simple geometry manufacturing,
- code is very unintuitive, most of them are coded as one later and a number (like G01),
- it does not contain enough information about the part, material and stock, so other type of data must be used to transfer this important information between CAD, CAM and CNC machine,
- it is almost impossible to back feed the information from CNC machine to the CAM and CAD software,
- it is almost impossible to run two different CNC machines by using the same NC program,
- spline interpolation is poorly integrated,

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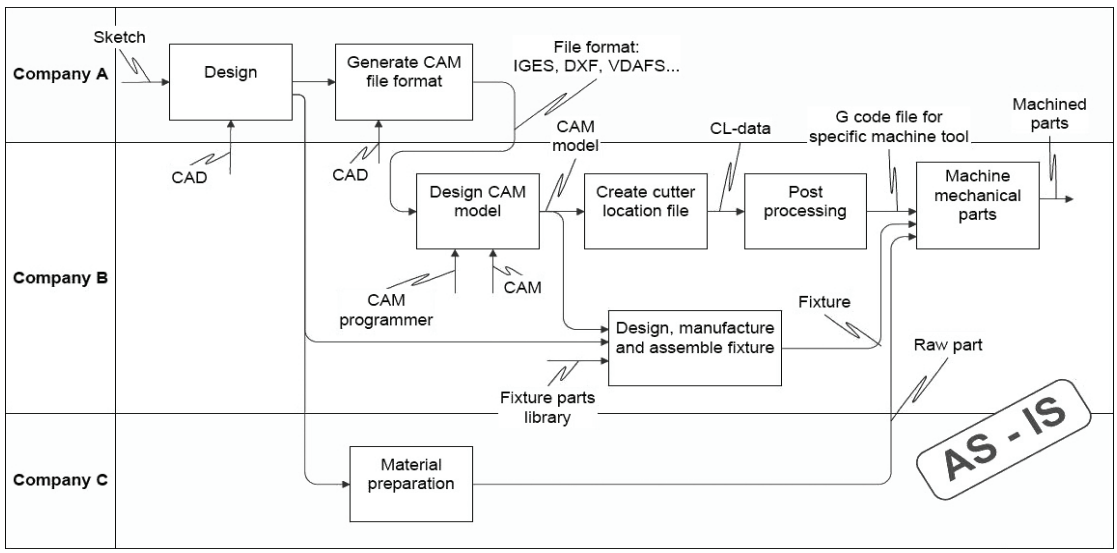

Fig. 1. *Current state of manufacturing data management* [1]

- poor support for kinematical features of 5axis machines.

2 STEP FOR CNC MANUFACTURING – STEP-NC

A big advantage of STEP is its ability to support many different application protocols which can be used in wide range of applications. By using the same file formats these application protocols can easily be interchanged. The final goal for the developers of STEP standard is that it could support the entire lifecycle of a product.

STEP-NC is one of the application protocols of the STEP standard. Its main goal is to support the interchange of geometry and technological data between CAD, CAM and CNC controls.

3 MAIN FEATURES OF A STEP-NC APPLICATION PROTOCOL (AP-238)

In STEP-NC data format (STEP AP-238) the technological steps to manufacture a part are defined as a sequence of material removing operations. Each operation represents removing a chunk of material of the regular geometric shape. The standard supports all 2D geometric features (holes, pockets, grooves) and also 3D geometry (geometry confined by 3D surfaces). Each removal operation adds its geometric features, tolerances, type and size of a tool, etc. The main advantage of the STEP-NC is that the data format is independent from the type of machine and its control (Fig. 2). This means that the use of postprocessors will not be needed anymore and that same program can be run on any kind of machine and any kind of control [10].

The second important feature of STEP-NC is that it allows companies to use their current CAD, CAPP and CAM software without the need of new investments in the software.

Main advantages of new data format are as follows [10]:

- NC programming can be much more efficient. The work of NC programmers will change from the precise setting of parameters in CAM programs in order to get the right toolpath to the ordering of material removal operations and setting the right technological parameters
- STEP-NC will provide enough data that the control will be able to optimize the program according to the situation of the tools in the magazine
- The need for paper documents will be largely reduced. It will also allow the instant change of the 3D geometry directly on the control.
- All the data needed to produce the part will be stored in a single file.

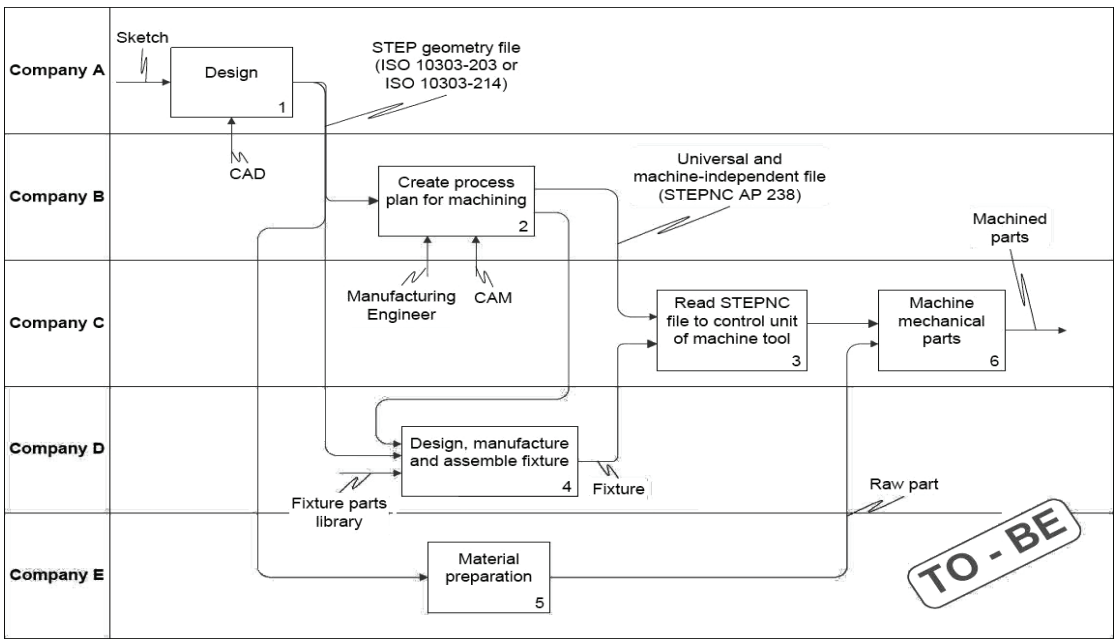

Fig. 2. *Manufacturing data management as proposed with STEP-NC* [1]

4 HISTORY OF STEP-NC RESEARCH

STEP-NC has been evolving for several years as the result of many research projects. These projects funded by industrial companies and state funded research institutions.

4.1 Project Optimal

The first project that started the path towards an industrial standard was OPTIMAL (Optimised Preparation of Manufacturing Information with Multi-Level CAM-CNC Coupling). This project was a part of ESPRIT 3 (Specific research and technological development program (EEC) in the field of information technologies, 1990-1994). It was founded by the European Union.

4.2 Project STEP-NC (EP 29708)

The project OPTIMAL was succeeded by the STEP-NC Compliant Data Interface for Numerical Controls (STEP-NC). This was the first pan-European project that resulted in first real data models (for milling, lathe and EDM). Each of the data model was tested in different usage scenarios. Real parts were machined by using STEP-NC data models. Partner companies (Table 1) came from many European countries: Germany, Switzerland, Great Britain, Sweden, France, and Italy [9].

The data models were tested by four real-life scenarios: drilling and milling, free surface milling, wood and glass machining and WEDM. The main result of this project is the development of software tools that show the capability of proposed data models [9]:

- simulation and verification of STEP-NC toolpaths (OpenMind)
- module for importing STEP-NC toolpaths into the CAM program (Dassault)
- database for distribution of files between different software tools (WZL)

4.3 Project Super Model

Super Model project was run from 1999 to 2002 by the company STEP Tools Inc, New York, USA. It was financed by the National Institute for Standardization (NIST) [4].

This project meant that American companies also joined the path of new standard development. Each year new features were presented in a special presentation show. In the last presentation software tools that enabled manufacturing by using ISO 10303-238 data model [4] were shown.

Table2. *Partners in the IMS 97006 project* [9]

Partner Company	Task
Development of CNC controls	
Agie (Switzerland)	Development of STEP-NC control for WEDM machines
CADCAMation (Italy)	Development of CAM module for WEDM machining
CMS (Italy)	Development of the protype STEP-NC supported machine for woodworking
Dassault (France)	Development of CAM module for 2,5D milling
Open Mind (Germany)	Development of CAM module for 2,5D milling
Osai (France)	Development of STEP-NC control for woodworking machines
Siemens (Germany)	Development of STEP-NC control 2,5D milling
Ral life testing	
DeimlerChrysler (Germany)	Milling and drilling on CNC machines
Volvo (Sweden)	Milling and drilling on CNC machines
Proghetti (Italy)	Milling CNC router machines
Wyss (Switzerland)	WEDM machining
Research institutions	
EIG i-tech (Switzerland)	Development of software tools or WEDM
EPFL (Switzerland)	Development of software tools or WEDM
ISW Stuttgart (Germany)	Development of data model for lathe
WZL Aachen (Germany)	Development of data model for lathe and milling

This presentation also showed the conversion from AP203 (STEP) to the AP238 (STEP-NC) and milling of the part on a NC machine. CAD model of the machined part is shown in the Fig. 3 [4].

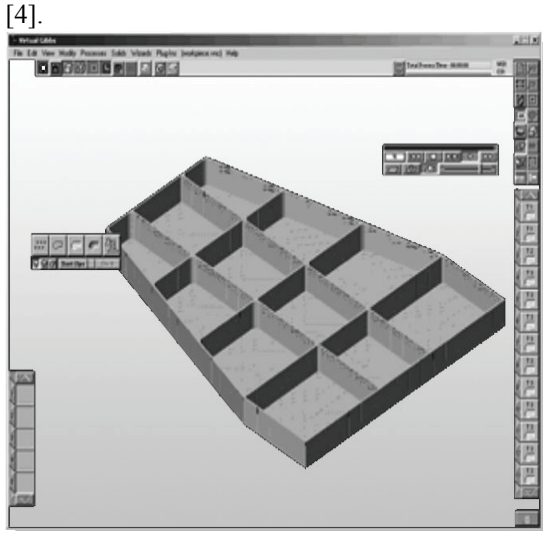

Fig. 3. *Part was machined at the last presentation of the Super Model project* [4]

4.4 Project IMS 97006 (Step-Compliant Data Interface For Numeric Controls)

This project was running from 2001 to 2003 and was a major step forward because it brought the unification of different partners that were previously independently developing STEP-NC standard. Partners from EU, USA, South Korea and Switzerland were involved in this project (Table 2) [12].

This unification of aspirations brought harmonization of experiences from previous projects. This harmonization resulted in two different standards: ISO 10303-238 and ISO 14469 [12].

5 LATEST DEVELOPMENTS IN STEP-NC DATA MODEL RESEARCH

Even though STEP-NC is currently supported as an active industrial standard it does not mean that its development is finished. The research is being done by several industrial and academic institutions all over the world.

Table2. *Partners in the IMS 97006 project* [4]

Geographical area	EU	Switzerland	Korea	USA
End user	Daimler-Chrysler Volvo, Franci	Dereninger, Wyss	Samsung	
CNC Machines producer	CMS	AGIE		
NC Controls producer	Siemens, OSAI			
CAM Software producer	OpenMind, Dassault	CADCAMation	Cubictek	STEP Tools, Inc.
Research institution	WZL, ISW, KTH	EPFL, EIG i-tech	ERC-ACI, KIST, NRL-SNT	
Number of partners	11	7	5	1

5.1 Research in Germany

The majority of research work in Germany is being dedicated to the development and implementation of STEP compliant CAPP/CAM software applications. The leading research institutions are ISW in Stuttgart and WZL Laboratory of the RWTH institute in Aachen. The ISW laboratory has developed STEPTurn, which is a prototype software application for programming of turning applications using STEP-NC data model. Its main advantage is the automatic planning of the manufacturing steps sequence. To achieve this the system uses two types of priority pairs. The 0/1 pair represents a distinct necessity to finish step 1 before continuing to step 2. In this case step 1 has the priority 1 and the step 2 has the priority 0. The +/+ pair is used when it is not important which activity is finished first. The use of this system to achieve process plan is shown on the Fig. 4 [4]

The Werkzeugmaschinenlabor (WZL) laboratory at the RWTH Aachen is mainly researching STEP compliant NC controls and CAM software (Fig. 5.). One of its products is graphic user interface for working with STEP-NC files [4].

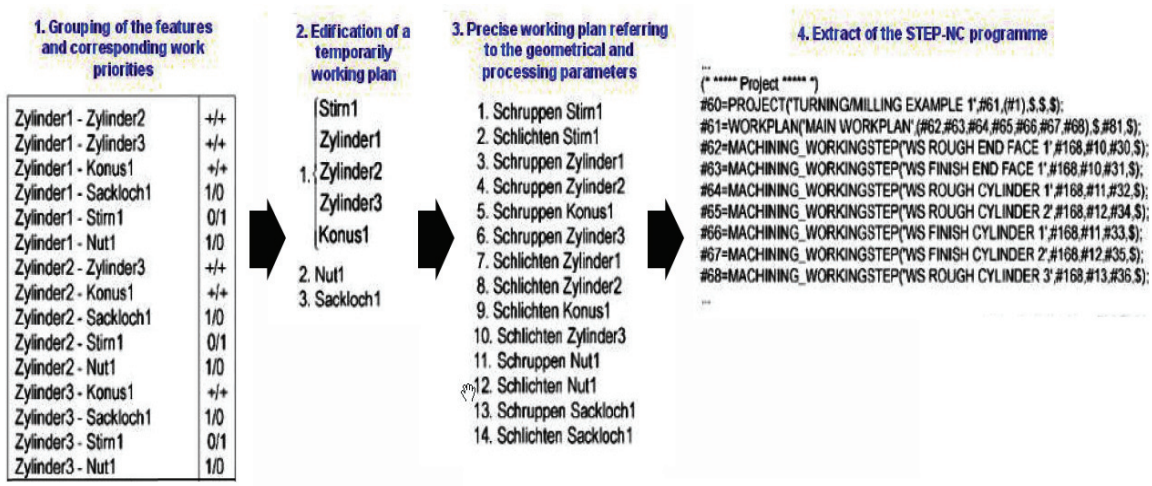

Fig. 4. *Process planning in the STEPTurn* [4]

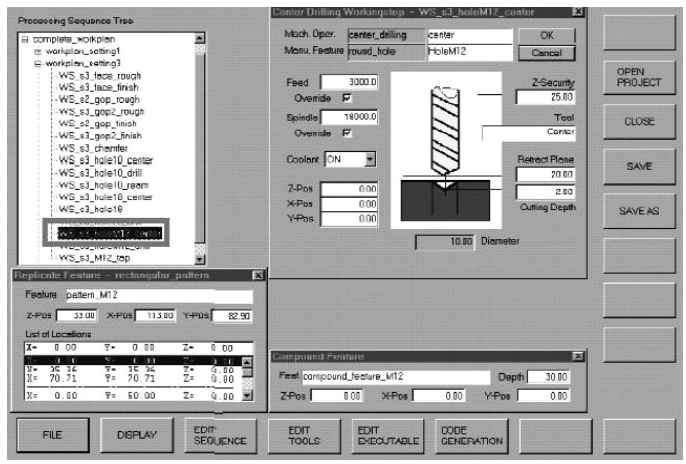

Fig. 5. *Graphical user interface for working with STEP-NC files developed at WZL-RWTH Aachen*[4]

5.2 Research in Switzerland

Switzerland is one of the leading countries in STEP-NC research. Many companies and research institutes have joined the process of STEP-NC development: Starrag, AGIE, AMT, CADCAMation, EIG i-tech, Derendinger, EPFL and Wyss. Most of the work is devoted to STEP-NC support for WEDM. The data model and the compatible WEDM systems were developed (Fig. 6). SolidWorks was used for test part modeling. The software module for STEP-NC support was developed by AlphaCAM [4].

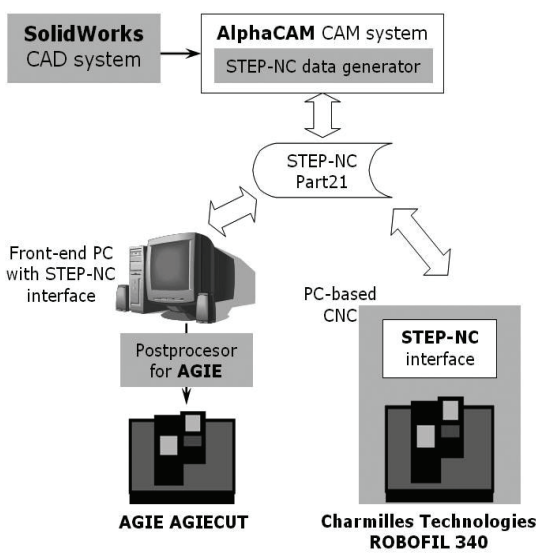

Fig. 6. *A diagram of prototype STEP-NC WEDM* [4]

5.3 Research in USA

Research in the USA is led by three different segments of industrial research [4]:

- government research institutions: NIST (National Institute of Standards and Technology)
- small privately owned companies: Step Tools inc., New York, NY
- major corporations: Boeing, Seattle, WA

Research by the NIST:

NIST focuses mainly on the development of the FBICS (Feature-Based Inspection and Control System) computer system, which is generally more focused to the CAPP field. The connection between FBICS and STEP-NC is that this program's final output is formatted by the STEP-NC data model instructions (Fig. 7.).

Main tasks in the development of FBCIS are [6]:

- demonstration of feature based environment for product development and measuring,
- it is the test bed for dealing with issues in the introduction of the feature based production to the real working environment,
- it is a test of real life usability of STEP-NC data model.

FBCIS is a computer program that incorporates functions of the NC control, functions for the process planning and modules for machining simulation and measuring. The entire program can be loaded directly to the CNC machine's control unit and can be used as a standard control. Its main features are the automatic creation of the machining procedure and its realization on the machine [6].

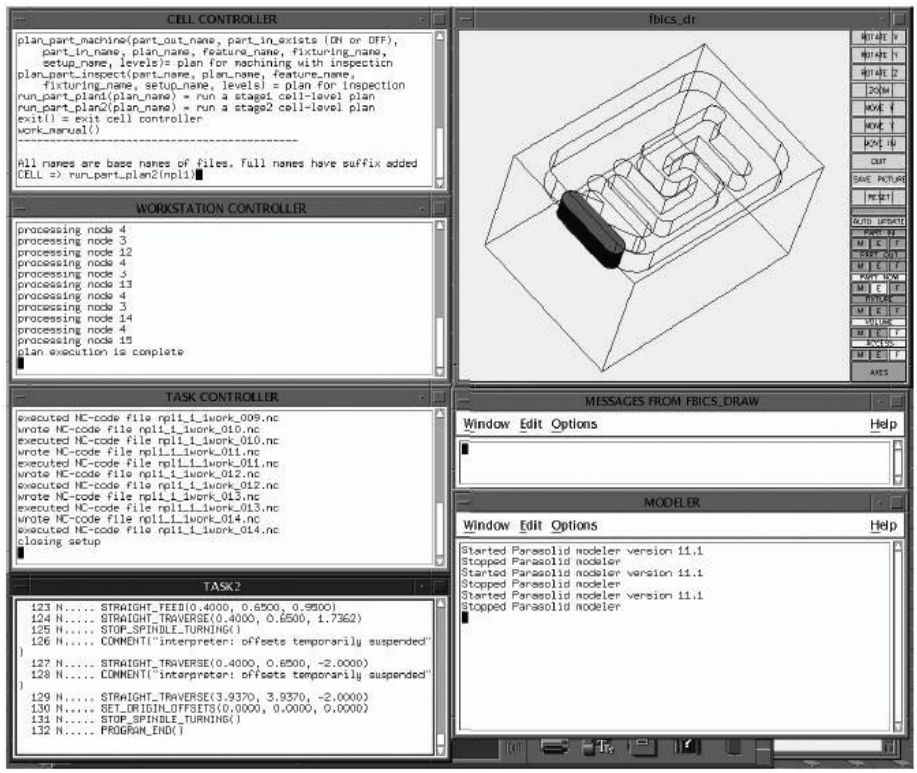

Fig. 7. *FBCIS's graphic user interface system [6]*

Research by the STEP Tools Inc:

STEP Tools Inc. is a privately owned small company which specializes in the field of STEP-NC standard development and support. They have developed a substantial amount of software solutions that are used for testing and using STEP-NC data model in the real life applications. These software solutions are [10]:

- STEP-NC Write,
- ST-MACHINE,
- STEP-NC Explorer,
- ST- Developer.

The important fact is that this library is issued under GNU General Public License (GPL) and is therefore, freely available [10].

ST-Machine is a computer program which is meant to be loaded to a machine's control unit. By using this program every machine can be changed to a STEP-NC compliant machine [10].

STEP-NC Explorer is a computer program which is used for reading and opening of the STEP-NC files on the PC computers. Besides opening STEP-NC files it can also convert NC programs from Mastercam, Catia, Unigraphics NX and Pro Engineer to STEP-NC [10].

ST-developer is a suite of software development tools. It offers these options [10]:

- DLL libraries for reading, writing and manipulating the STEP-NC files,
- it includes ST-Viewer for STEP-NC files viewing,
- Tools for opening AP-203, AP-209 and AP-214 files.

Research at The Boeing Company:

Boeing manufactures a large number of aircraft parts on many different 5-axis machine tools. Currently all their NC programs are coded in ISO 6983 NC code and are different from machine to machine. Because different 5-axis machines have different axis configurations this means that their 5-axis NC programs can be substantially different. If they want to change the machining of a part from one machine to the other, another postprocessing is required to get an NC program for the specific machine tool. Because the company is using STEP as a format

for 3D model data exchange with great success they have decided to run a pilot project of using STEP-NC data format on several different machine tools [4].

5.4 Research in South Korea

In South Korea there are two groups that are working on the development of STEP-NC: RL-SNT (National Research Laboratory for STEP-NC Technology) in Pohang University of Science & Technology and ERC-ACI (Engineering Research Center for Advance Control and Instrumentation) in Seoul National University. NRL-SNT has been developing STEP-NC compliant controls since 1997 and has developed Korean STEP-NC system. Its main component is a module for computer assisted NC programming - Shop-Floor Programming (SFP) (Fig. 8). As shown on this module can create STEP-NC of both standardized types (ISO 14649 and ISO 10303-238) [4].

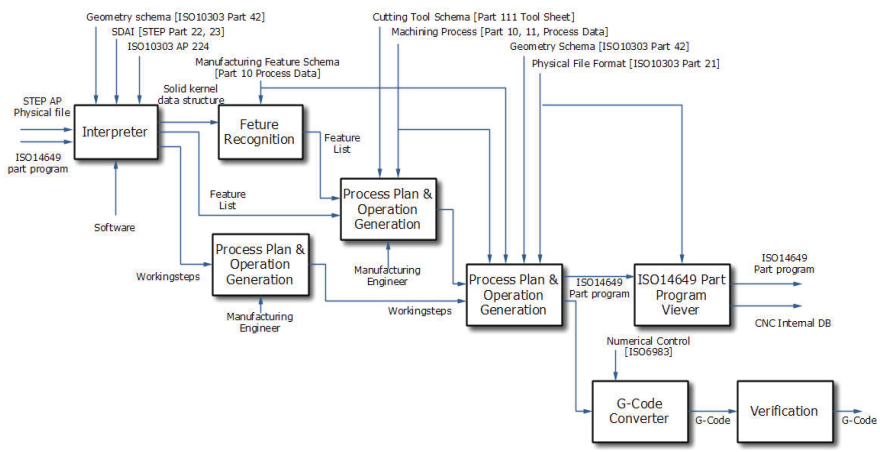

Fig. 8. *Diagram of SFP module [4]*

6 CURRENT SUPPORT FOR STEP-NC THROUGH INDUSTRIAL STANDARDS

Currently there are two standards that cover the field of STEP-NC: ISO 14649 and ISO 10303-238. Each of them covers its own portion data exchange between different steps in product development and manufacturing.

6.1 ISO 14649

- This standard became active in the years 2003-2005. Its main purpose is to replace the ISO 6938 as the standard for NC program data formatting. It is planned that in the initial phases of standard implementation the CNC machines would support both standards [3]. Fig. 9 shows the initial phase of STEP-NC implementation. The deficiencies of ISO 6938 are compensated by describing working process with different working steps which are needed to produce the geometry instead of tool movement trajectory. These working steps represent higher level geometry elements combined with all the process parameters needed in order to manufacture them. NC controls become responsible for translating these working steps into actual tool movement.

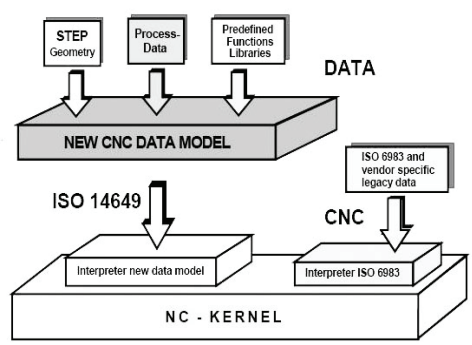

Fig. 9. *The NC control can support both standard (ISO 6938 and ISO 14649)* [3]

This data structure also allows using the same STEP-NC program on any type of machine without the use of postprocessors [3].

ISO 14649 data format is very different from the old NC-code. It is well structured and object oriented [3]. Its main elements are (Fig. 10):

- header,
- geometrical features of the part,
- technological working steps for manufacturing (each working step is composed by many feature/technology operations).

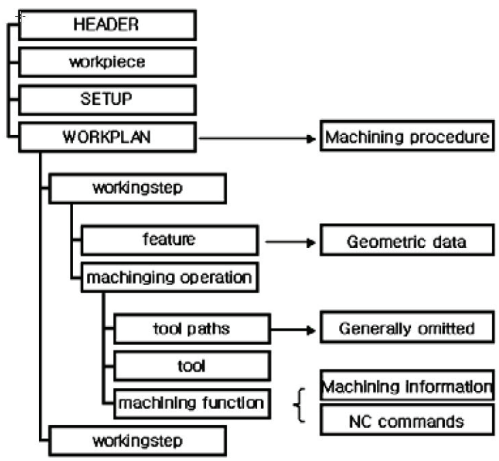

Fig. 10. *Graphical overview of ISO 14649 data organization [13]*

Table 3 shows how all the data in the file are written in plain text using special coding [2].

Table 3. *A simple example of ISO 14649 data structure [2]*

ISO 14694 data format	Explanation
Header	
1=Project(WorkPlan#10);	Select Workkplan at #10
#10=WorkPlan(#20,#35,#71,....);	Workplan is consisted of: WorkStep1 at #20, WorkStep2 at #35,…
.	
#20=WorkStep1(,#21(Feature), #22(Manufacturing))	WorkStep1: manufacture geometry feature #21 using technological parameters #22
#21=Hole('Tap M6',,,,,,,,,,);	geometry feature #21
#22=Drilling(#..(Tool),,,#..(Technology),#..;	technological parameters #22
.	
#35= WorkStep2 (.....);	WorkStep1
END-ISO-10303-21;	End of program

6.2 ISO 10303-238

This standard was officially released on 30. 4. 2007 as ISO 10303-238:2007. Its main difference compared to ISO 14649 is that this data model also includes a 3D CAD model of the product. It is basically an upgrade of an ISO 14649 standard. The Table 4 clearly shows that a large portion of AP238's data structure implementation was copied from previously released ISO 14649 [5].

Table 4. *Origins of different elements of ISO 10303-238 data model [8]*

Element of AP238 data model	Element origin
Part 3D geometry	All STEP application protocols share the same data model
Dimensions and tolerances	STEP GT&D
Measures Project data Stock data Geometry features Working steps sequence Toolpaths	ISO 14649 Part10
Milling operations Milling tools	ISO 14649 Part11 ISO 14649 Part111
Lathe operations Lathe tools	ISO 14649 Part12 ISO 14649 Part121

The data model of AP238 is structured into three main parts [10]:

- part 3D representation,
- working plan for manufacturing,
- technological parameters for manufacturing.

A part 3D representation uses the same data model as other STEP application models. A 3D Part is described by using ISO 10303 standard data model, also all the information about owners and creation dates are included. This data can be

created by any STEP compatible CAD program [5].

Besides the 3D representation the standard foresees that the part is assembled by using interconnected features [11]. Each feature is a distinct geometric shape of a special type: 2.5D element (hole, pocket, groove…), transitional element (radius, chamfer…), and element of repeating (mirror, copy…), region element (3D surfaces), lathe element (grooves…) and others. Some of 2.5D features are shown on the Fig. 11.

Working plan for manufacturing is "the heart" of the AP238 application protocol. It contains information about manufacturing steps which is needed to manufacture the part. Fig. 12 shows the plan, which is divided into several working steps. They are instructions that designate which strategies and parameters are used to manufacture each feature [10].

7 DIFFERENCES BETWEEN ISO 14649 AND ISO 10303-238

ISO has issued two STEP-NC standards. Each of them has its strengths as its disadvantages. The main difference is the level of integration of the STEP architecture. The comparison is shown in Table 5 [4].

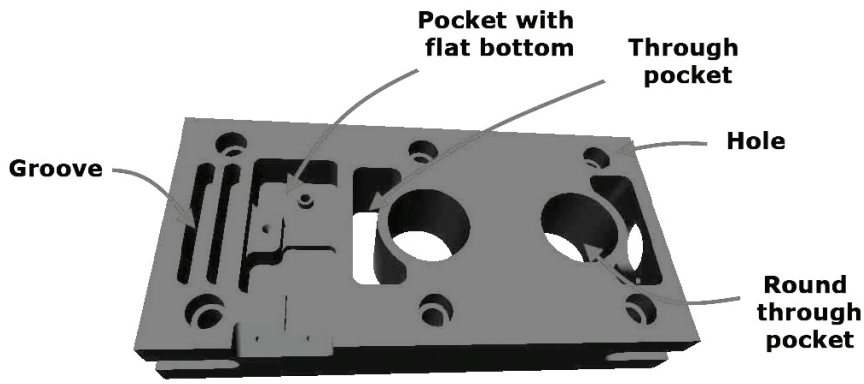

Fig. 11. *A part with 2.5D features* [8]

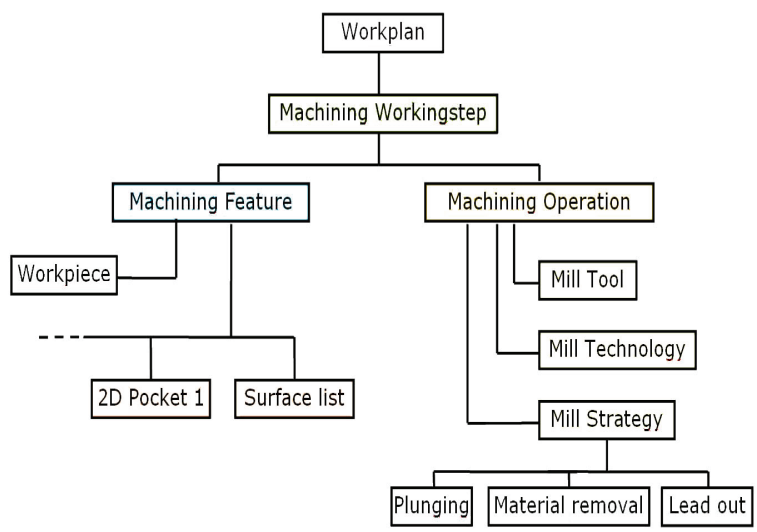

Fig. 12. *Manufacturing working plan* [4]

Table 5. *Comparison between ISO 14649 and ISO 10303-238 [4]*

Comparison criteria	ISO 14649	ISO 10303-238
File size	10-times smaller than ISO 10303-238	10-times bigger than ISO 14649
Difficulty for programming	Easy	Complicated
Is the file readable?	Very hard	Impossible
STEP compatibility	Partially	Fully
Data consistency	Original geometry data is lost	Original geometry data remains

ISO 14649 in ISO 10303-238 can be used as two different methods for the implementation of STEP standard into manufacturing. ISO 14649 is intended for applications where CAM software has total access to all the data from the production, whereas STEP AP-238 is intended for total integration of CAD, CAM and manufacturing. Even though STEP AP238 is more advanced this means that the implementation is much more complex and that it will take much longer for its practical realization [4].

8 CHALENGES FOR FUTURE RESEARCH

STEP-NC is still not developed enough to be used in real engineering practice. The data model is still showing some deficiencies, which will have to be resolved before the standard can be globally accepted [4].

8.1 Feedback to Previous Steps in Product Development

Improvement of data feedback from current step to all the previous steps of the product lifecycle is one of the best solutions to improve product quality, shorten the production time and lower the production costs. Currently a global use of G-code format is very convenient from the aspect of creating NC programs for the CNC machines but it is quite useless when we want the feedback from production to the previous steps of a product lifecycle. STEP-NC is big step forward because of several reasons [4]:

- STEP-NC includes information about manufacturing process on a much higher level than G-code. This allows for seamless feedback from NC machine to the CAD and CAM.
- STEP-NC allows the transfer of data about actual manufacturing parameters back to the process of design and process planning. This allows the implementation of the "design-for-manufacturing".

8.2 Tool modelling

Currently STEP-NC does not allow the tool compensation as G-code with G41 and G42 does. If the tool radius is minimally changed, the toolpath must be calculated again. For this standard to be widely accepted it will have to allow small changes in tool geometry without the need of toolpath recalculation [4].

8.3 Tolerances

Tolerances are the basic principle by which the interchangeability and functionality of the products can be guaranteed. In addition, they have an extremely important influence on the production by the production costs and productivity point of view. If we want the STEP-NC to become a globally accepted standard it must also support the information about tolerances of the part manufacturing. To this moment the STEP-NC has not yet solved this issue suitably.

The main issue of integration of tolerances into the data model is their double role in the part representation. When used in assembly the tolerances are connected to the part sizes and are therefore, called construction tolerances. In manufacturing other types of tolerances are used and they are called production tolerances. Often these are not equal to the construction tolerances. Clearly shows the difference between construction and production tolerances. [4].

9 CONCLUSION

STEP-NC is currently an internationally supported industrial standard for data exchange in the process of product design and manufacturing. Before it will actually be accepted in real life it

will take some more time. I believe the reasons for this are as follows:

- ISO 6938 standard is well established and widely used. Its main advantage is great simplicity. For some applications it is easier to use ISO 6983 than STEP-NC,
- STEP-NC is still not developed enough and it still lacks some features that are inevitable in the manufacturing process,
- CNC machines and controls producers have not sent to the market any product that would be STEP-NC compliant, therefore, the users have not even started to think there is any other way than ISO 6938.

Even though the STEP-NC is not used yet in real world's applications at all it is a step in the right direction. TCP/IP standard is a clear example what standardization can bring to the industrial field. All the hardware and software in the billions of products exchange data all over the world without any problems. The reason for this is the standardization of data exchange protocols. The process of product lifetime management really needs the same kind of standardization. It will bring the world together, minimize design and manufacturing costs and make the process easier to manage.

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