STEP-NC Standard for Integration of CAD and CAM

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Abstract— This paper presents an overview about standard representation of product data for better communication and exchange between CAD/ CAM systems. For mechanical parts, standard use for description of product data is ISO 10303- Product data representation and exchange, also known as STEP. This make possible to use standard data during design, process planning and production stage to integrate CAD and CAM. The main barriers to realizing this concept is most of computer numerical control (CNC) machines, which use ISO 6983 G/M codes language. G/M codes lack the portability because of the following reasons. First it focus only on tool center path rather than machining process, second some vendors use language supplement with their own extensions programme which make it ambiguous. ISO 14649 called as STEP-NC is a new standards which extend STEP standard for NC. STEP-NC replaces G/M codes by remedies the above shortcomings. STEP-NC specifying machining processes rather than machine tool motion, using the concept of working steps. Working steps correspond to high-level machining features and associated process parameters. The controller of future CNC's are responsible for translating Working steps to axis motion.

***Index Terms*—CNC, CAD/CAM, G-code, Product Data Exchange, STEP-NC**

I. Introduction

The current standard to program NC machine tools has had no change significantly since the first NC (numerical control) machine was developed at M.I.T. (Massachusetts Institute of Technology), U.S.A in early 1950's. Today's Computer Numerical Control (CNC) machines and these early NC machines continue to use the same standard for programming known as G & M codes based on the ISO 6893 standard[1].

Since the 1970's significant developments have been made towards more automatic and reliable computer numerically controlled machines with new processes such as punching & nibbling, water jet cutting, and laser cutting which are now common place. The advent of the CNC brought a massive improvement in the capabilities of these machines. Currently CNC machines provide the ability of multi-axis, multi-tool, and multi-processes manufacture. These capabilities have made the programming task more and more difficult and off-
line software tools for CAD/CAM a necessity for efficient code generation. Though these developments have revolutionised CNC processes and capabilities, the programming language has basically stayed the same with G/M codes programming which was developed in the 1950's and later became the ISO 6983 standard that is based on the tool path and machine status description.

ISO 14649 is a new standard for data transferring between CAD/CAM systems and CNC machines. A major benefit of using ISO 14649 is its total conformity to ISO 10303 which is use for product data representation of mechanical parts known as STEP [2]. In fact, ISO 14649 popularly called as STEP-NC, STEP extended to NC [3]. It remedies the shortcomings of ISO 6983 by specifying the machining processes rather than machine tool motion by means of ''Working-step'', as the basic entity. Working-steps are effectively machining tasks that correspond to high-level machining features and associated process parameters. The future CNC controllers are responsible for translating Working-steps, instead of G, M codes, to axis motion and tool operations.

II. ISO 6983 and the Problems

According to ISO 6983, the CNC coding is based on the following stipulations [1]:

Preparatory functions:	From G0 to G99;
Miscellaneous commands:	M (also called Machine function);
Axis motion commands:	X, Y, Z, A , B, C;
Feed and speed commands:	F (Feed Rate), S (Spindle Speed);
Identification commands:	N (Block number); T (Cutting Tool).

Because of the efficiency in processing, precision in machining and easiness in operating, CNC machines have been widely used in manufacturing industries all over the world. Nevertheless, a number of problems are found with ISO 6983, which are summarized underneath.

1. The CAD data are not used directly on the machine, instead, they have to be processed by a machine-specific post-processor, only to obtain a set of low-level, incomplete data that makes verification and simulation difficult, if not impossible.
2. ISO 6983 does not support the spline data, which makes it incapable of controlling five or more axis milling.
3. There is limited control of program execution and it is difficult to change the program in the workshop.
4. It only supports one-way information flow from design to manufacturing; the changes in the shop floor cannot be directly fed back to the designer. Hence, invaluable experiences on the shop-floor can hardly be preserved.

5. Vendors usually supplement the language with extensions that are not covered in the limited scope of ISO 6983; hence the CNC programs are not exchangeable.
6. The standard defines the syntax of program statements, but in most cases leaves the semantics ambiguous.
7. The language focuses on programming the path of the cutter centre location (CL) with respect to the machine axes, rather than the machining tasks with respect to the part.

III. STEP-NC

In past various data formats have been developed for exchange of data. Their primary purpose is to exchange geometric data. The widely accepted formats are Initial Graphics Exchange Standard (IGES), Drawing Transfer File (DXF), and the Product Description Exchange for Standard (PDES).Published in 1994, ISO 10303 (STEP) defines the industrial automation systems and integration for representation and exchange of product data. Compare to its predecessor STEP provides a neutral mechanism by specifying a form capable of describing the entire product data throughout the life cycle of a product, independent of any particular system. STEP is not suitable not only for neutral file exchange, but also as a basis for archiving data and for implementing product databases.

STEP is not considered as an only way to transfer data between different CAD-systems, but it is also developed towards manufacturing information management. STEP-NC provides a full description of the part and manufacturing process as well-annotating CAD design data with manufacturing information about the stock, its cutting characteristics, and tool requirements. STEP-NC defines data representing working steps, a library of specific machining operations performed at the CNC, because of that any controller will be able to calculate the tool path based on definitions contained in formatted routines integrated within the controller itself [20].

STEP-NC is a new model of data transfer between CAD/CAM systems, which replaces G/M-codes. Contrary to the current NC programming standard ISO 6983 known by G/M codes, the ISO 14649 is not method for programming and does not describe the tool movements for a CNC machine. Instead, STEP-NC provides a object oriented data model for CNC's with a detailed and structural data interface that incorporates future base programming where there is a range of information such as feature to be machined, type of tools used, the operations to perform and the work plan [19].It is specifying machining processes rather than machine tool motion, using the object-oriented concept of working-steps. Working-steps correspond to high-level machining features and associated process parameters. CNCs are responsible for translating working-steps to axis motion and tool operation. A major benefit of STEP-NC is its use of existing data models from STEP. Basically, the standard is the smooth and seamless exchange of part information between CAD, CAM, and NC programming [4].

Main problem of Programming with G/M codes is, due to many different "dialects" and vendor-specific additions to the programming language, part programmes are not interchangeable between different controls. As a result, porting programs between machines is difficult as shown in Figure 1. STEP-NC provides a comprehensive model of the manufacturing process. It is object and feature oriented and describes the machining operations executed on the work piece, and not machine dependent axis motions. It will be running on different machine tools or controllers, it will provide good compatibility as shown in Figure 2. This compatibility will spare all data adaptations by postprocessors, if the new data model is correctly implemented on the NC controllers [20].

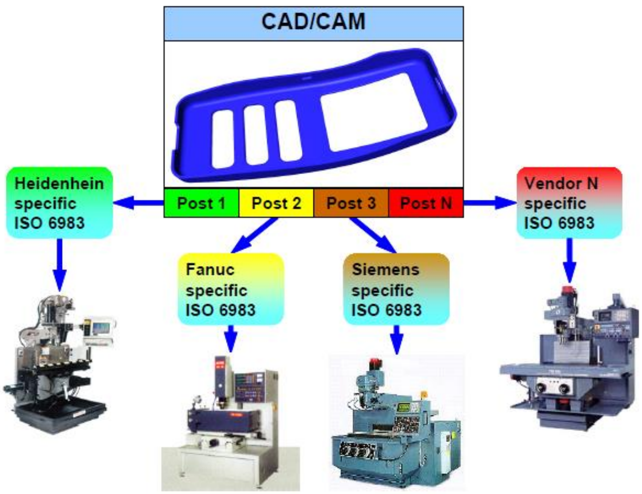

Figure 1. Current situation in CAM systems and CNC machines[20].

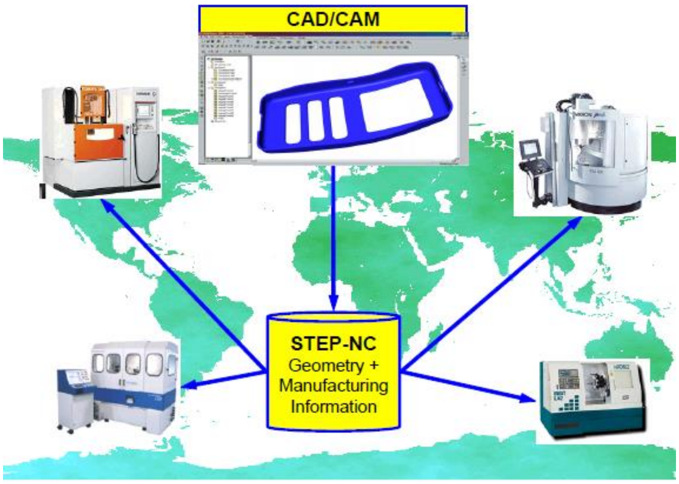

Figure 2. Tomorrow CAM and CNC[20]

Today in CAD system, the STEP data translation capabilities implemented are primarily support geometric models and configurations management as specified in Configuration controlled 3D designs of mechanical parts and assemblies (ISO 10303-203) and Core data for automotive mechanical design processes (ISO 10303-214). STEP extended by STEP-NC in manufacturing domain by adding data structures that support computer-aided manufacturing (CAM) and computerized numerical controller (CNC) requirements [7].

Sharing of common data structures between STEP parts are become possible by STEP architecture. A CAD system gives output of geometry, design features, and product identification data in STEP format. Then a CAM system input and use that data it to develop the detailed process data needed to manufacture the part. There is no requirement for CAM systems to redefine the geometry and features because these data structures are shared between STEP parts. The CAM systems output geometry, process sequence, features and tool requirement data in STEP-NC format, which are used by CNC machine Controller [7]. This offers following advantages over traditional methods:

1. STEP-NC provides a complete and structured data model, linked with geometric and technological information, so that no information is lost between the different stages of the process.
2. STEP-NC allows for complete safety checking because safety areas for fixtures can be defined as part of the setup.
3. Documentation may be easily generated by the CAM or CNC system to show the state of the part before and after each working step.
4. STEP-NC is easy to generate; specific tool paths need not to be defined in advance.
5. A STEP-NC file is not machine specific; the STEP-NC file can be manufactured on any machine that meets the tooling requirements.
6. XML files can be used for information transfer hence enable Web based manufacturing or e-manufacturing.
7. Machining time for small to medium sized job lots can be reduced because intelligent optimization can be built into the CNC controller.
8. Modification at the shop-floor can be saved and fed back to the design department hence bi-directional information flow from CAD/CAM to NC is achieved.

IV. Research work in STEP-NC

There are several research project carried out by university and companies to developed STEP-NC. The global research in the areas of STEP-NC has been primarily coordinated under a single IMS (Intelligent Manufacturing System) project in 2002, which effectively entails an international package of actions. Table 1 shows the participants and the distribution of the technological scope within the proposed project. Partners from four regions are invited to participate in the project: EU, Switzerland, Korea and USA. The regional coordinators are Siemens (EU), CADCAMation (Switzerland), STEP Tools (USA) and ERC-ACI (Korea). Siemens is also the inter regional coordinator. The most active research activities are found among the partners in the EU regions, USA and Korea. They covered the manufacturers of all systems related to the data interface (CAM-systems, controls, and machine tools), the users and academic institutions [6].

V. STEP-NC data format

The fundamental principle of the STEP-NC Data Model is the object-oriented view of programming in terms of manufacturing features, instead of direct coding of sequences of axis motions and tool functions as per defined in ISO 6983. The objects in this case are manufacturing features and their associated process data. The Data Model is a layer that provides a standard interface between the controller interpreter and the different sources of data supplied. The interpreted data are supplied to the CNC kernel, where axis motions and machine functionality are generated. This new model can accept data from several sources. The data can be generated by a CAD/CAM system, part of libraries and/ or graphic user interface. The data generated from CAD/CAM systems or graphical user interfaces is normally processed automatically with computers, so feature definitions, technology descriptions, readability and completeness have been given higher priority than compressed coding. The program format follows the same format for STEP as described in a Physical File Format (ISO 10303 Part 21). Fig. 3 shows the internal structure of STEP-NC data. The first section of the part program is the header section marked by the keyword ‘‘HEADER’’. In this header, some general information and comments concerning the part program are given, e.g. filename, author, date and organization. The second and main section of the program file is the data section marked by the keyword ‘‘DATA’’. This section contains all the information about manufacturing tasks and geometries. The contents of the data section are further divided into three parts: work plan and executables, technology description, and geometry description. The project entity serves as a starting point for executing the part program. Each part program must include an instance of this entity in a ‘‘DATA’’ section of an ISO 10303 Part 21 file. This instance should contain a main work-plan that contains sequenced subsets of executables (executable manufacturing tasks or commands) and may also include information of work-piece to be machined [2, 3]. The executable is the base entity of all executable objects. There are three types of executables: Working-step, NC function and program structure. The work-plan combines several executables in a linear order or depending on given conditions if conditional controls are used. In STEPNC, all geometrical data for work-piece, set-ups and manufacturing features are described based on ISO 10303 parts 21 and 42/43 This data is also used directly by the CNC machines to avoid conversions between different data formats that may result in reduced accuracy. Tooling information including tool type,

tool geometry and expected tool life, etc. is also included in a STEP-NC file. The exert from a STEP-NC file is shown as follows [6]:

```
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,$,$,$,20.000,5.000,0.000);
#18=AXIS2 _PLACEMENT 3D(‘Position of contour’,
#19,#20,#21);
#19=CARTESIAN POINT(‘Position of contour’,
(40.000,90.000,100.000));
#20=DIRECTION(‘‘,(0.0,0.0,1.0));
#21=DIRECTION(‘‘,(1.0,0.0,0.0));
#22=TOLERAN-
CED_LENGTH _MEASURE(20.000,$,$,$);
#23=COMPOSITE CURVE(‘Contour Profile’,
(#24,#25,#56),.F.);
… … … …
// Tool data
```

```
#40=CUTTING TOOL('Endmill
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;
```

TABLE 1. THE PARTICIPANTS AND THE DISTRIBUTION OF THE TECHNOLOGICAL SCOPE.[6]

Region	**EU**	**Switzerland**	**Korea**	**USA**
Technologies covered	Milling, Contour cutting, Turning & Inspection	Wire/Sink EDM	Rapid Prototyping	AIM for Milling & Turning
End-user	• Daimler Chrysler • Volvo • Franci • Progetti	• Derendinger • Wyss	• Samsung	• Boeing • Lockheed Martin • General Electric • General Dynamics • General Motors
Machine tool manufacturer	• CMS	• AGIE • Starrag		
Control manufacturer	• Siemens • OSAI • Fidia			
CAM manufacturer	• Open Mind • Dassault	• CADCAMatio n	• Cubictek	• STEP Tools • Gibbs • BA Solutions Numerical Control Services
Research institute	• WZL • ISW • KTH	• EPFL • EIG I-tech	• ERC-ACI • KIST • NRL-SNT	• Louisiana Centre for Manufacturing Sciences • Lawrence Livermore National Laboratories
Association	• CECIMO	• AMT		• NIST • Department of Energy • Army.s National Automotive Centre

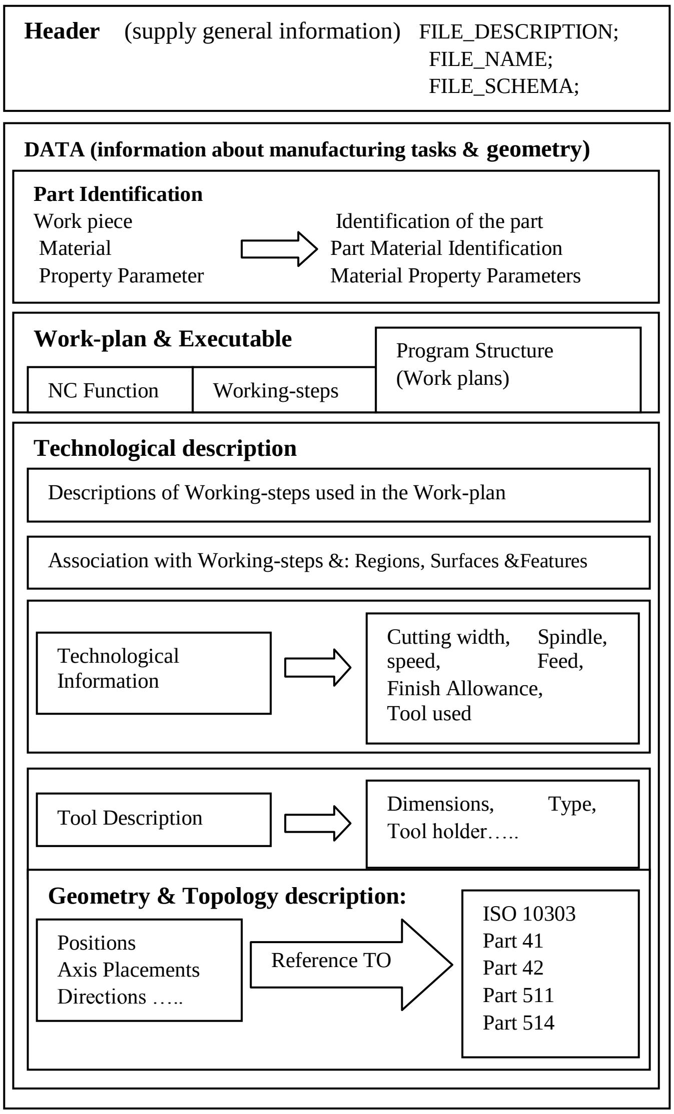

Figure 3- The general structure of a STEP compliant NC code [3].

VI. Challenges for STEP-NC

Although STEP-NC is appealing and gathering more and more interest from researchers, there are some challenges to overcome before it can be realized.

- Certain capabilities of CAM systems are moved to CNC controllers by adopting STEPNC. It results in much more complicated controllers and requires more knowledge from the machine operators.
- Before STEP-NC can be practically used in manufacturing new STEP-NC compliant controllers for CNC machines have to be developed first. CNC controller manufacturers then have to redesign the structures and strategies of their controllers to make it happen.
- As STEP-NC is gradually implemented on the CNC machines, the databases for milling, turning, EDM and other STEP-NC supported processes need to be developed and standardized.
- Security issues have to be sufficiently addressed when transmitting STEP-NC via Internet or LAN to enable e-manufacturing.
- Resistance can be expected from the manufacturing industry for the reason that ISO 6983 has deeply rooted in both people's minds and machinery's structure.

VII. Opportunities with STEP-NC

STEP-NC offers good opportunities for different parties. These opportunities, should they be missed, will be transformed into threats. On other end, companies who grab the opportunities will surely excel. For instance, the manufacturing firms who have successfully implemented STEP in their Product Data Management system and e-manufacturing/e-business, can easily extend their competitive edge in today's much globalized world economy. For the machine tool manufacturers, opportunities also exist. Those who venture into investing and supplying machine tools with STEP-NC enabled CNC controllers will certainly put them in an advantageous position in the market. In a similar manner, this applies to CNC controller manufacturers. CAM and even

CAD/CAM vendors can increase their competitiveness by adding STEP-NC compliant software to their product. This is believed to be the forerunner of the STEP-NC's commercialization mission. Software vendors can invest in and develop STEPNC translators, or otherwise known as ''plug-ins'', for the major CAD/CAM systems. The primary function of these types of software tools is that of feature recognition, i.e. embodiment of AP203 files to achieve AP224 and eventually AP238 files.

VIII. CONCLUSION

Current NC programming is based on ISO 6983, called G-code, where the cutter motion is mainly specified in terms of position and the feed rate of axes. Even though G/M codes are a well-accepted standard world-wide it is in fact a bottleneck for today's CNC production chain. Programming with G/M codes results huge programmes which are difficult to handle; last-minute changes or correction of machining problems on the shop floor are hardly possible and control of programme execution at the machine is severely limited. Even worse, due to many different "dialects" and vendor-specific additions to the programming language, part programmes are not interchangeable between different controls. As a result, porting programs between machines is difficult.

STEP-NC provides not only a full description of the part, but the manufacturing process as well-annotating CAD design data with manufacturing information about the stock, its cutting characteristics, and tool requirements. STEP-NC defines data representing working steps, a library of specific machining operations performed at the CNC, so that any controller will be able to calculate the tool path based on definitions contained in formatted routines integrated within the controller itself.

STEP-NC is a new model of data transfer between CAD/CAM systems and CNC machines, which replaces G-code. It remedies the shortcomings of G/M codes by specifying machining processes rather than machine tool motion. Working-steps correspond to high-level machining features and associated process parameters. CNCs are responsible for translating working-steps to axis motion and tool operation. Basically, the standard is the smooth and seamless exchange of part information between CAD, CAM, and NC programming.

At the moment STEP-NC standardization is in ISO/DIS phase (Draft International Standard) and international development work continues in many countries. Many end-users have started their pilot-projects concerning utilization of STEP-NC and also CAD/CAM software vendors have made implementations for research projects.

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