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TYING TOGETHER DESIGN, PROCESS PLANNING AND MACHINING WITH STEP-NC TECHNOLOGY*

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ABSTRACT

While simulation has been successful in tying design and process planning into an iterative loop, machining has traditionally been a downstream terminus of the manufacturing cycle. Simulation of machining has proven difficult due to the highly dynamic and nonlinear nature of the material removal process. Consequently, actual machining must be performed to determine the suitability of designs and process plans. The nature of the programming interface to machine tools has been an obstacle to information flow in both directions. Toward the machine tool, useful design and process information is reduced to primitive tool path motion statements. From the machine tool, no automatic data paths exist, so machinists must translate results into recommendations for design or process changes manually. The advent of STEP (ISO 10303) data for numerical control, or STEP-NC, promises to rectify these problems by providing full product and process data to the machine tool controller at run time and an automatic path back to these upstream activities using technologies such as XML. This paper describes STEP-NC and early experiences that show how these benefits can be achieved.

KEYWORDS: manufacturing, design, process planning, machining, numerical control, STEP, ISO 10303, ISO 14649

INTRODUCTION

Discrete parts manufacturing typically follows a sequence from design through process planning to machining and inspection. At each step, experts use software programs with which they are familiar and which are tailored to the particular domain. In design, tools

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include computer-aided design (CAD) and finite-element analysis (FEA) systems. Design is a labor-intensive process, often beginning with conceptual drawings, that results in electronic data files. In the 1980s, design files were exchanged according to the Interim Graphics Exchange Standard (IGES) [1]. IGES has been superseded by the Standard for the Exchange of Product Model Data (STEP), International Organization for Standardization ISO 10303 [2], particularly STEP Application Protocol (AP) 203 [3]. AP 203 defines solid model geometry using boundary representations.

In process planning, computer-aided manufacturing (CAM) software takes AP 203 design geometry, and uses manufacturing rules to generate process plans suitable for automatic machining by computer-numerical control (CNC) machine tools. Process plans are typically tool paths, expressed in Automatically Programmed Tools (APT) format [4]. APT is not typically executed directly by machine tools, but is converted into another format, ISO 6983 [6], the "G codes" familiar to machinists. This conversion is known as numerical control (NC) post-processing, and it is complicated by a proliferation of vendor-specific extensions to the ISO 6983 standard. The combination of APT data generation and NC post-processing strips out almost all of the product and process data contained in the CAM system, resulting in a primitive sequence of straight-line or circular arc motions. Some CNCs extend their ISO 6983 dialect to include more complex tool paths, such as non-uniform rational B-splines (NURBS), but these extensions are non-portable. Even if ISO 6983 were to be updated with more complex tool path types, NC code would still be lacking any information on material type, tolerances, surface finish, expected forces, and other product or process data that could be used to support intelligent control.

Once a part is machined, it may be inspected either on the machine tool itself, or on separate metrology equipment such as a coordinate measuring machine (CMM). If on-machine inspection is done, the inspection routines are typically coded as macro programs in the vendor's extension to ISO 6983. Inspection macros may be as simple as logging points to a data file for later analysis, or as complex as determining systematic errors and adjusting offsets so that successive runs are improved. However, the parameters referenced by these macros, typically tolerances, are not conveyed with the NC code in any standard way. As a result, changes to the design- or manufacturing tolerances made upstream of the CNC may be lost, requiring manual re-work of the inspection macros. Ad hoc methods are used in practice to streamline the process, but it remains tedious, error-prone, and non-portable. The situation is similar for post-process inspection on CMMs. In this situation, inspection programs are written during the process planning phase, using inspection planning software that generates either Dimensional Measuring Interface Standard programs (DMIS) [5], or programs in the CMM vendor's proprietary format. Changes to the design- or manufacturing tolerances necessitate a repeat of the inspection planning phase.

The loss of product and process data that occurs when process plans are converted into much simpler formats has been the primary barrier to the introduction of intelligent control for machining. Recognition of this led to the introduction in the mid-1990s of STEP-NC (ISO 14649, and ISO 10303 AP 238). STEP-NC is intended to be a replacement for G codes that provides much richer data and allows feedback of process information from the machine tool back upstream to design and process planning. STEP-NC is currently being validated in quasi-production pilot projects, and is showing promise as the next generation data interface to machine tools.

STEP-NC

STEP-NC is a draft ISO specification for a data model for machining. STEP-NC differs from conventional tool path representations in its use of machining features as the basic element. Working steps link machining features with tools and technology parameters such as feed rate and spindle speed. The collection of previously flat sequences of tool paths into working steps is analogous to object-oriented programming, where working steps are the objects. At the very least, this provides a more intuitive view into the intent of a machine tool program for the operator, assisting with setup and supervising. More importantly, it provides the machine tool controller with information that can be used for just-in-time tool selection, tool path generation, intelligent error recovery, and other capabilities for intelligent control.

STEP-NC is divided into two standards, ISO 14649 and ISO 10303 AP 238. ISO 14649 is the responsibility of ISO Technical Committee 184, Subcommittee 1 (TC 184 SC1). It defines the data requirements for machining. ISO 14649 has several parts, covering generic machining data, and process-specific data for milling, turning, wire- and sink electrical discharge machining (EDM), and tool requirements appropriate to each process. ISO 14649 is defined using the EXPRESS data modeling language from STEP (ISO 10303 Part 11) [7], and is an Application Reference Model (ARM) in STEP parlance. That is, it models the data in a standalone way, not necessarily with any association to other data models.

ISO 10303 AP 238 is an Application Interpreted Model (AIM) in STEP parlance, and maps the data modeled in the STEP-NC ARM onto STEP's integrated resources. The integrated resources include common data models used throughout STEP, including tolerances, geometry and topology representations, and material. AP 238 is harmonized with existing STEP APs, enabling a much tighter integration with applications that use STEP. AP 238 is the responsibility of ISO TC 184 SC1, the same subcommittee responsible for STEP itself. The integration accomplished through AP 238 is expected to pave the way for full integration of CAD, CAM, and CNC, the first two of which already use AP 203 to transfer design geometry.

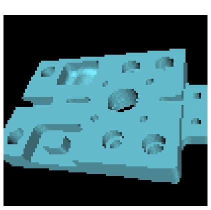
STEP-NC EXPERIENCES

STEP-NC is being validated in quasi-production pilot tests in the U.S. and internationally. Most of the work has focused on milling, the first of the processes to be represented in both ARM and AIM formats. Full deployment of STEP-NC requires support through the entire manufacturing process: design (in CAD systems), process planning (in CAM systems), and machining and inspection (in CNC machine tools). CAD systems currently use AP 203 to exchange design geometry with enough success to claim that AP 203 has replaced IGES as the format preferred in the automotive and aerospace industries. The mature use of AP 203 in CAD systems means that for the most part, the CAD phase already supports STEP-NC. What is lacking in CAD systems is a standard way to output design tolerances. While this is not an impediment to demonstrating initial STEP-NC results, in the long term tolerances need to be conveyed both to CAM systems and CNCs. For CAM, tolerances can be used to automatically select machining processes, such as rough milling and finish milling. Currently these processes are selected with some degree of human interaction, typically a trained machinist looking at tolerance annotations on printed drawings and picking which operations are suitable. For CNC, tolerances can be used to dynamically change motion parameters and effect a speedup in machining for those regions where tolerances are large enough to allow it. These parameters include such values as the maximum allowable radius through a corner, or the degree of blending along a path approximated by line segments. Tests conducted by a major supplier of CNCs in an aerospace application showed that when an NC program is supplemented with tolerance information, dynamic adjustment of parameters can result in a 30% reduction in machining time.

CAM systems currently use AP 203 as an input format, tying them together with CAD systems, but process plan output remains the rudimentary APT and ultimately ISO 6983 tool path. Thus, aside from adding support for tolerance inputs, CAM systems must be changed to support AP 238 output. Likewise, CNCs must be changed to support AP 238 as input. This is not to say that AP 238-enabled CNCs will no longer be programmable using legacy G codes. Rather, it is expected that new CNCs will be able to handle both forms, with G code support for existing NC programs (many of which may be years or even decades old) and for manual data input by machinists who prefer this method for job setup.

An incremental advance toward STEP-NC is to convert ISO 14649 or AP 238 process plans into ISO 6983 G codes, and run the G code programs as usual. This of course assumes that one can generate ISO 14649 or AP 238 programs to begin with. Assuming one can (as will be discussed shortly), this demonstrates that STEP-NC is at least equivalent to G codes. If the conversion of STEP-NC to G codes is automatic, then it can be done just in time on the CNC itself (or on a networked computer nearby), and for all intents and purposes is part of a STEP-NC-compliant machine tool. This approach does nothing to further the cause of intelligent control, but does demonstrate an early path to STEP-NC adoption. The participants in ISO TC184 SC1 used this technique extensively during the early testing phases [8].

STEP-NC programs were initially generated using research software tools. As shown in Figure 1, the data format is far too complex to hand edit. The analogy often made is that STEP-NC is like the PostScript printer data format, with G codes similar to simple plotter statements. Plotter statements are short and easy to understand, but are rudimentary. PostScript is a far richer language, but also much more complicated.



```
DATA;
...
#10=PLANE_ANGLE_MEASURE_WITH_UNIT(PLANE_ANGLE_M
EASURE(0.01745329252),#9804);
...
#3962=PATH('General Closed Path Profile''s
path.',(#13311,#13312,#13313,#13314,#13315,#133
16,#13317,#13318,#13319,#13320,#13321,#13322,#1
3323,#13324,#13325,#13326,#13327,#13328,#13329,
#13330,#13331,#13332));
...
#4705=PATH_FEATURE_COMPONENT('boss
height','linear',#9363,.T.)
...
ENDSEC;
```

Figure 1. Excerpts from ISO 10303 AP 238 data describing milling operations for the STEP-NC test part shown in the accompanying picture.

There is some commercial support for generating STEP-NC data files, through a combination of feature recognizing software, legacy CAM systems and AP-238 extensions. In this scenario, AP 203 geometry is created using a conventional CAD system. This geometry is described using solid model boundary representations, but does not contain any definition of features. In order to be useful in a STEP-NC environment, features must be recognized out of this solid model geometry. Commercial software is available for this, which automatically recognizes most if not all of the part geometry as belonging to features. A person can assign any remaining unrecognized geometry interactively. The procedure continues with macro-level process planning, in which tools, feeds and speeds are associated with each feature but actual tool paths are not computed. The STEP-NC data model for tools defines tool requirements, not actual tools to be selected, and as much or as little of the tool requirements can be specified. If little information is defined, more flexibility is left up to the micro-level process planner when selecting actual tools and generating tool paths. This process is shown in Figure 2.

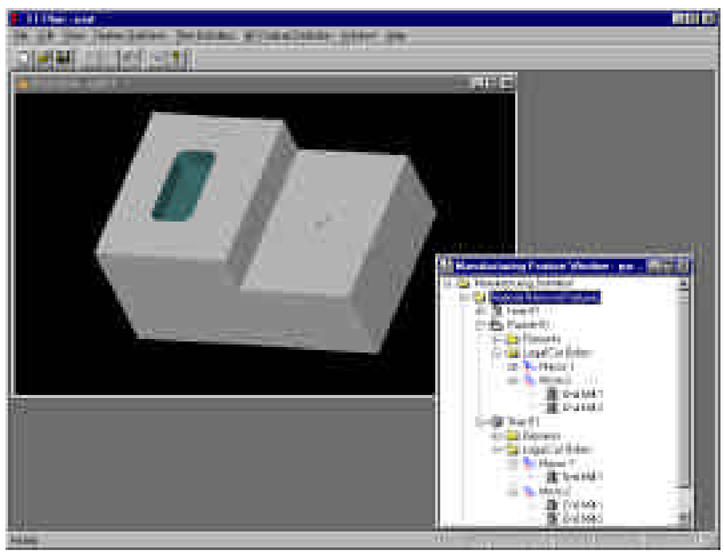


Figure 2. A simple test part with a step, pocket, and hole, and the features, working steps, and tools shown in a browseable window. Courtesy of STEP Tools, Inc.

An AP 238 file results from this macro-level process planning stage, and is then passed into commercial CAM systems through extensions that handle conversion of AP 238 into the native data formats of the CAM systems. This process is currently the least well developed, and considerable effort is being expended by CAM vendors to improve the degree of integration. Nevertheless, successful tool path generation has been accomplished in different demonstrations at the General Dynamics facility in Scranton, PA USA, the Louisiana Center for Manufacturing Sciences (LCMS) in Shreveport, LA USA, and the National Institute of Standards and Technology (NIST) in Gaithersburg, MD USA. In these demonstrations, an AP 238 macro-level process plan was converted to XML, the Extensible Markup Language, and then loaded into one of two commercial CAM systems using the extensions described earlier. At General Dynamics and NIST, the CAM software generated G code programs that were executed by the CNCs as usual. In the LCMS facility, the CAM software skipped NC code generation, and converted each tool path to a machine tool motion by calling a software application programming interface (API) provided by the controller. The tests showed a limited subset of STEP-NC functionality, for features such as faces, slots, pockets, and holes. Work is ongoing to validate more complex features, such as intersecting pockets and arbitrary contoured edges and 2-$^{1}/_{2}$ D surfaces. Future work will include eliminating G code interpretation by directly processing STEP-NC data, which will eventually lead to exploitation of product and process data not possible with legacy programming languages like G codes.

SUMMARY

STEP-NC is a pair of ISO standards, ISO 14649 and ISO 10303 AP 238, that together define machining and tooling requirements for processes such as milling, turning, and EDM, and integrate these data models into the existing STEP framework. The intent of STEP-NC is to tie together design, process planning, machining, and inspection into a seamless flow of data. STEP-NC is based on the concept of machining features rather than primitive point-to-point tool motion. In the downstream direction, design changes such as feature locations or tolerances can be conveyed through automatic process planning systems and onto machine tools, eliminating tedious hand editing of different file formats whenever changes are made. In the upstream direction, changes to machining or inspection programs made on the shop floor can be made available automatically to design and process planning systems, so that lessons learned in production tests can be relayed more efficiently to designers and process engineers. Perhaps more importantly, the availability of the rich data afforded by STEP enables true intelligent control, in which machine tools have access to material information, tolerances, and the logical flow of working steps. STEP-NC is currently being validated in pilot projects both in the U.S. and internationally. To date validation has focused on feature-based machining. Future work will include the introduction of product and process data that will enable true intelligent control.

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