➤ BY SUSAN WOODS, FEATURES EDITOR

STEPin' Out

STEP-NC is almost an official ISO standard, but when will it become part of everyday use?

According to Martin Hardwick, president and CEO of STEP Tools Inc., Troy, N.Y., the STEP-NC standard has been approved and is scheduled to be published by the International Organization for Standardization in June. That will be a major hurdle for the data standard, and brings it one step closer to being put into everyday use.

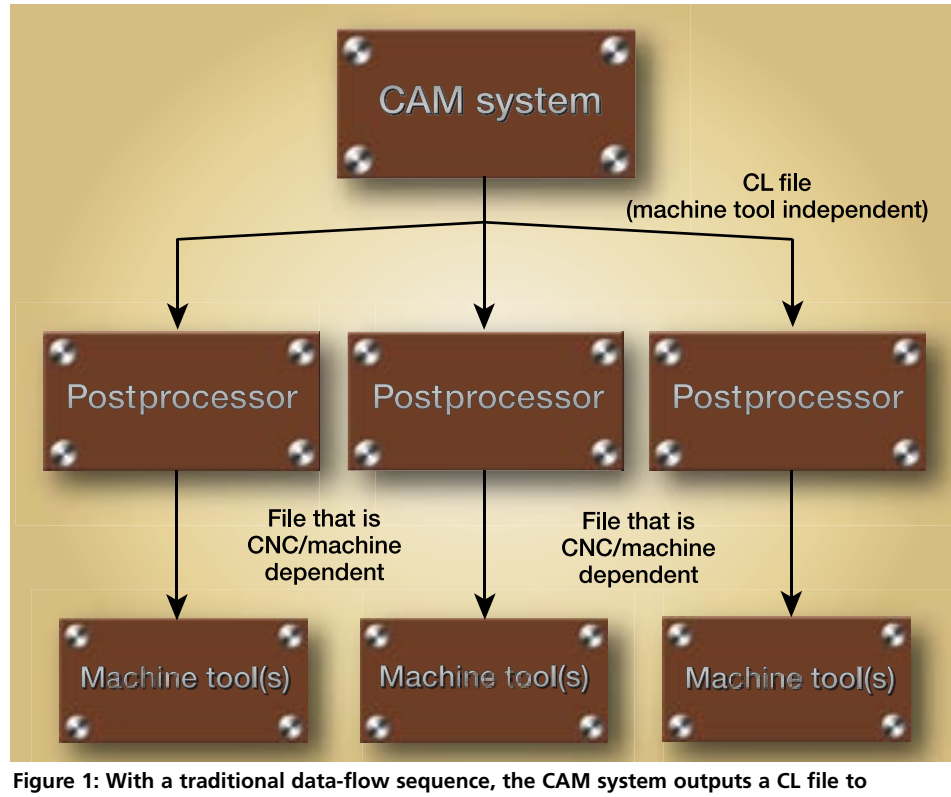

Figure 1: With a traditional data-flow sequence, the CAM system outputs a CL file to individual postprocessors that output M and G code to the CNC/machine tool that is CNC/machine tool-dependant.

Limitations of Code

RS274D is the data standard currently used to tell CNC systems how to make a part. An RS274D file contains a list of M and G code that tells the machine tool what moves to make to cut the part and to control various functions, like coolant flow.

In conventional CNC machining, a part is designed using a CAD system. Then the design model is output in a variety of different file formats, such as DXF/DWG, IGES, ACIS, Parasolid or native CAD, and that is read into a CAM system. The CAM system then develops a process to machine the workpiece to the finished part geometry. This process is converted by a postprocessor into G and M code, which is then sent to the machine tool's CNC.

It can take hundreds of thousands of lines of code to make a part. G and M code only tells the machine tool where to move the cutting tool. It does not provide any information about the part being machined.

Over the years, CNC vendors have added customized code to the RS274D language. The code is not standardized, thus, a postprocessor configured for one CNC/machine tool may not be suitable for another. A new postprocessor must be developed for each specific CNC/machine tool. It is not uncommon to have to maintain a vast library of postprocessors to support the machine tools within a manufacturing facility.

The First Step

The Standard for the Exchange of Product Model Data (STEP) is a data standard developed based on other product data-exchange standards, such as IGES, that gives an explicit and complete representation of product data throughout its entire life cycle (design, engineering analysis, manufacture, support, and maintenance and disposal). STEP is intended for neutral product model data exchange and archival storage.

The core of the STEP standard is a catalog of application protocols. For example, STEP AP-203 defines a standard for CAD design data. It is used for the exchange of CAD model data, regardless of what system the data originated in. STEP AP-203 became a full ISO standard in 1994, and since then CAD vendors have implemented STEP AP-203 data translation.

Making Steps

STEP AP-238 is the official STEP

application protocol for the STEP-NC standard. STEP-NC defines a data standard for CNC systems. STEP-NC replaces the RS274D standard as the primary interface between CAM and CNC systems.

STEP-NC is an extension of STEP that enhances the design information with manufacturing information, including information about the workpiece material, its manufacturing features, tolerances, tool requirements, process sequence, speeds, feeds and other process parameters.

STEP-NC defines data in "working steps"—a group of specific operations to be performed on a CNC machine tool. It breaks down every machining operation into the steps required to perform the operation.

STEP-NC allows the process information that already exists in CAM systems to be output to CNC machine tools. This expanded process information offers the potential of expanded capabilities on CNC machine tools.

The original concept of STEP-NC envisioned capturing all the data and process information required to create a part, and the STEP-NC-enabled CNC was expected to generate the toolpath. The explicit toolpath was a byproduct used only to statically document the toolpath generated by the CNC. In this way, the STEP-NC file was transferable between various machine tools because each machine tool was responsible for generating its own toolpath, given the information contained in the STEP-NC file.

"When STEP-NC was first conceptualized, [roughly] 5 years ago, the thought was that you would describe to a machine tool the sequence of processes that you wanted to have occur but it would do the low-level detailing—things like explicit toolpath generation and selection of tooling and the determination of speeds and feeds," said John Callen, vice president of marketing at Gibbs and Associates, Moorpark, Calif. "I would tell the machine tool to cut a pocket, and I would give it the pocket's parameters. The machine tool, with its intelligence, would determine what tools to apply, what toolpaths would need to be followed to remove the material and what cutting parameters would be necessary."

That was a fairly ambitious approach. There was a realization within the STEP-NC consortium* that to move that level of intelligence down to the machine tool would be a fairly significant hurdle.

So, the consortium embraced the notion of explicit toolpath as a fundamental data structure. Based on this new orientation using explicit toolpath, a series of conformance classes, which allow support of the standard in minimal subsets, was defined for STEP-NC. The conformance classes are:
CC1—explicit toolpath geometry;
CC2—part geometry;
CC3—feature and operation data;
CC4—design specification data (dimensions and tolerances).

The first two conformance classes are used to manufacture the part from the toolpath data. The latter conformance classes can take advantage of feature information to do some toolpath planning on the CNC.

"Today, in the STEP-NC data structure, the lowest-level conformance class, CC1, literally is an ISO standard representation of explicit toolpath with some structure that directly replaces standard M and G code," explained Callen. "The notion that the machine tool itself generates the toolpath that it uses to realize the process—generative toolpath—is relegated to the highest conformance class, CC4."

The following companies contributed to this report:

Gibbs and Associates
(800) 654-9399
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National Institute of Standards and Technology
(301) 975-6478
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STEP Tools Inc.
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**The STEP-NC consortium is made up of members of the Open Modular Control Systems (OMAC) Users Group's Machine Tools Working Group. The consortium participants include people from Boeing, Gibbs and Associates, STEP Tools, Unigraphics, National Institute of Standards and Technology and General Dynamics. The consortium is working with other international groups to pursue an international STEP-NC standard.*

In the Meantime

Until there are native STEP-NC CAM systems and CNCs, an interim solution is available to allow people to take advantage of STEP-NC today.

With the interim step, the CAM system outputs AP-238 data via an AP-238 plug-in interface. That STEP-NC file is sent to a STEP-NC converter on an existing CNC that reformats the AP-238 toolpath data into M and G code so the CNC can read it.

"ST Machine is our product for doing this," said Hardwick. "We have developed a converter that runs on Fanuc and Siemens open-architecture controls that converts the STEP-NC data into data those controls understand and can execute immediately."

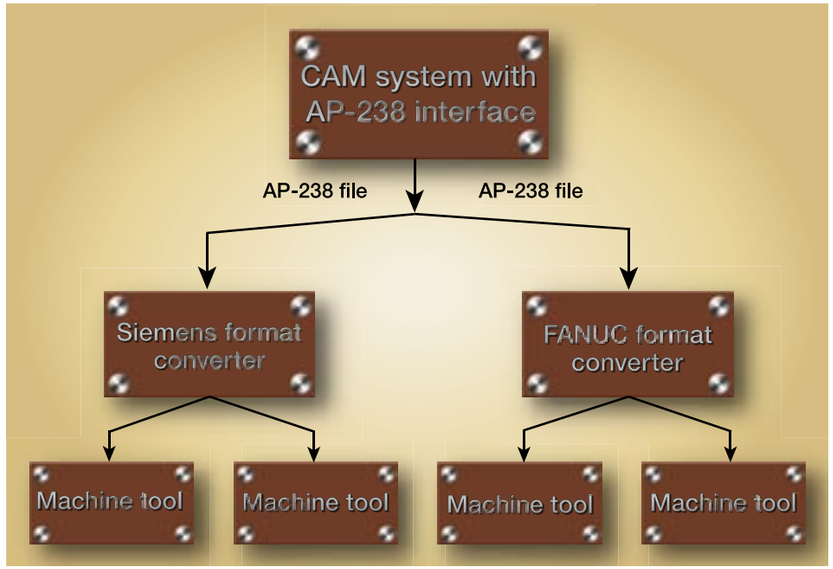

Figure 2: With the interim solution, the CAM system outputs AP-238 data via an AP-238 interface. That STEP-NC file is sent to a converter that translates it into traditional M and G code so the CNC can read it.

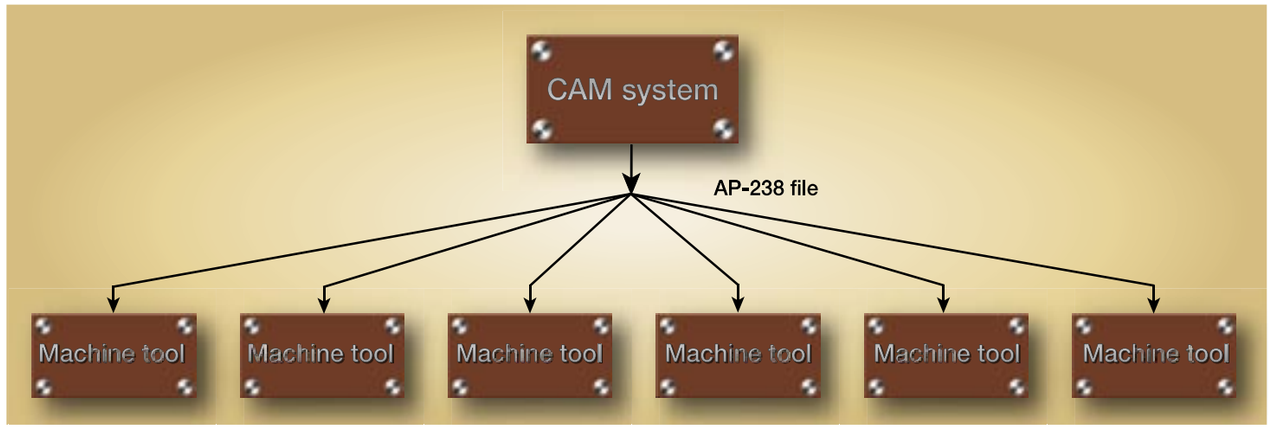

Figure 3: The final STEP-NC goal is to have the CAM system output AP-238 code directly to the CNC/machine tool.

There are some benefits to outputting STEP-NC data, even if it must be converted back to M and G code. "STEP-NC is richer data," said Fred Proctor, group leader of the Control Systems Group, National Institute of Standards and Technology. "With STEP-NC, you can visualize the operation. Instead of looking at a flat stream of G code scrolling by on a screen, you can bring up a window and navigate the features in a STEP-NC file. You can see the working steps that are going to cut specific areas of the part. So it is much more amenable to visualization."

Hardwick added that it is also easy to take advantage of the second conformance class, CC2, because all the end user needs to do is merge the CAD models of the part and the workpiece and the fixtures with the toolpaths to create a more complete description of the manufacturing process. STEP Tools' converter has an interface that performs this merger and gives the operator a rich visualization of the machining process that can be checked for potential collisions and to maximize efficiency.

A STEP-NC CC2 file provides a more complete representation of the machining process. This allows utilities to be built that allow the process to be viewed in context, something most CAM systems can do, but is beyond what standard M and G code supports.

Based on Demand

STEP-NC's adoption by the industry really depends on how the end users, CNC/machine tool vendors, and CAD and CAM vendors respond to it.

The standard will not take off until end users start requesting it. "Vendors operate off customer demand," said Callen. "If there is demand for functionality, it is considered against all the other customer requests for functionality and our resources are allocated based on what the anticipated return on investment is. Most of our customers are still in the process of adopting other STEP application protocols, such as AP-203 and AP-214. For them, AP-238 isn't even on their radar."

End users already have systems in place that work well enough for them. Introducing a new technology, even if it displaces a less desirable one, is not necessarily a straightforward decision. They are not willing to put their conventional production at risk. Many manufacturers, including automakers, have complete production infrastructures based on M- and G-code processes.

The Waiting Game

The STEP-NC consortium is working with CAD, CAM and CNC vendors to make the standard a reality. "There is this dance of who is going to go first," said Proctor. "The CNC vendors are saying, 'We can't get any STEP-NC data, so why should we implement it?' and the CAM guys are saying, 'Well, even if we implement it, who is going to use it?' There will probably have to be some sort of collaboration between the CAM and CNC vendors where they both agree to a timetable by which they would have products ready to generate and use STEP-NC data."

Part of the problem with getting CNC vendors on board is that they rely on being able to differentiate their products from other vendors' products.

"There is specialization that each control vendor introduces into M and G code to allow them to differentiate themselves from other vendors," said Callen. "These vendors see STEP-NC as something that homogenizes the marketplace, which means that anything they might use to differentiate their products essentially gets boiled away. "

If they have a proprietary language that only they support, and their customers have invested time into using their interface, those customers are likely to stay with that vendor. This is not necessarily a good thing from the customer's point of view. "We hear this a lot from the Tier 1s and Tier 2s," said Proctor. "They have to buy one of every CAD/CAM system in order to service all of their customers. It is expensive for them."

While they might not realize it, STEP-NC can be in the CNC vendors' best interest. If they have a unique feature in their CNC, the only way their customers are going to be able to use it is if they invest the time and effort. This can make for a hard sell.

"If a vendor introduces a new feature for their controller, it is going to have some new G code for it," said Proctor. "So in order for customers to use this new feature, they have to buy a new postprocessor or write their own. Since that is an expensive proposition, they probably aren't going to use that feature."

There is a balance between tying customers to a CNC vendor's proprietary language and making it riskier for them to use new features because they don't want to become even more tied to a vendor. STEP-NC would provide a quick avenue for the CNC vendors to get data in their CNC that they need for their new features and eliminate the risk for the end users, Proctor added.

STEP-NC essentially eliminates the need to have a unique postprocessor for each machine tool configuration. The purpose of STEP-NC is to displace all of these various postprocessors with one format, the STEP-NC format.

CAM systems also need to be re-engineered to support STEP-NC. Part of this depends on the quality of information provided to CAM systems from CAD systems.

This is another hurdle that needs to be overcome. CAD systems need to provide a product data model with geometry, surface-finish and tolerance specifications. CAD vendors are beginning to evolve their geometric modeling systems into product modeling systems, but the evolution is not complete. △