Challenges and Choices in the Specification and Implementation of the STEP-NC AP-238 Standard

Martin Hardwick
Department of Computer Science,
RPI,
Troy, NY 12180

David Loffredo
STEP Tools, Inc.,
14 First Street,
Troy, NY 12180

STEP-NC AP-238 is the result of a ten-year effort to replace the RS274D (ISO 6983) M and G code standard with a modern associative language that connects CAD design data to CAM process data. STEP-NC builds on previous work to develop a neutral data standard for CAD data and uses the modern geometric constructs of that standard to define device-independent tool paths, and CAM-independent volume removal features. This paper describes four key factors, or challenges, that had to be addressed to extend STEP-NC AP-238 functionality beyond that offered by other CNC standards. The four factors are difficulties inherent in all CAD/CAM integration projects and can be summarized as follows: the information complexity of geometry, the volume of attributes defined for manufacturing processes, the difficulties of integrating models that span CAD and CAM, and the requirement for easy implementation. We describe an advanced CNC application that uses these four factors and show how STEP-NC AP-238 enhances CNC machining and measurement. [DOI: 10.1115/1.2768090]

Introduction

STEP-NC has been developed to make computerized numerical control (CNC) machine tools easier to program and safer to operate by combining computer-aided design (CAD) information with computer-aided manufacturing (CAM) information. The CAD information allows the operation of CNC controls to be made more visual and enables new quality control and measurement applications on the control. The CAM information allows for intelligent reprograming when some part of the process must change.

STEP-NC was published as an international standard in the ISO 10303 series as Application Protocol 238 (AP-238) by the International Organization for Standardization in 2007 [1]. It is the result of a ten-year effort to replace the RS274D interface for machine tools with a more modern associative language. There have been previous attempts to replace RS274D, such as the EIA 494 Basic Control Language (BCL) [2] and the ISO 14649 object model. Each of these previous attempts was technically sound and achieved sufficient consensus for standardization but, to date, has not been adopted on a sufficient scale to challenge the status quo.

In this paper, we describe four key factors that we believe will allow STEP-NC AP-238 to replace RS274D as the main standard for transmitting programing data to CNC controls. The first two factors were put in place by others, and the last two by the authors. Together, they make AP-238 usable by today's systems, provide significant new benefits, and should enable wide deployment.

The next four sections describe each factor: (1) how CAD geometry was modeled and why STEP-NC uses EXPRESS as its information modeling language, (2) the machining capabilities put into the standard by the ISO 14649 experts, (3) how meeting the requirements of design and manufacturing required the standard to be integrated, and (4) how new implementation methods make deployment by vendors and users easier. The discussion is followed by a description of an example application that uses the four factors. Finally, we conclude by summarizing the results and discussing other new applications that can be enabled by STEP-NC AP-238.

Factor 1: Modeling CAD Geometry

Figure 1 shows an example of the RS274D (ISO 6983) language and describes some of its limitations. RS274D is similar to computer assembly languages. Low-level codes like this were once used to control printers and many other kinds of devices but, in recent years, have been replaced by high-level languages such as Postscript and XML.

Replacing RS274D with something more semantic has been a goal of the industry since about 1980, but engineering the replacement has proved difficult. One issue was the difficulty of interpolating five-axis 3D geometry in real time for expensive, potentially dangerous machinery. This has largely disappeared because of the increasing power of microprocessors. Another barrier was the difficulty of replacing vector codes, which are easy to understand, with 3D geometry data, which is easy to view but much harder to understand when written as words in a text document.

The first ISO STEP standards were developed in the 1980s and 1990s to enable data exchange between CAD systems [3]. After extensive study of the available options, the STEP community concluded that they needed a new information modeling language to describe 3D geometry and other product model data. A key requirement for this language, called EXPRESS [4] was an advanced inheritance model called AND/OR inheritance that goes beyond multiple inheritance and allows types to be combined using logical operators.

Figure 2 shows some STEP usage of EXPRESS. In the "point" definition, all attributes of the point are defined elsewhere in the inheritance tree. In the "b_spline_curve" definition, the EXPRESS SUPERTYPE clause describes the allowed combinations in the inheritance tree, and the WHERE clause describes additional instance constraints.

Because of its complexity, the use of EXPRESS is not widespread. Instead, the web community invented XML Schema for defining information exchange. XML Schema does not model inheritance, though it does allow a form of refinement that can be used to simulate single inheritance. XML Schema has become very popular, but trying to understand the EXPRESS models of STEP in XML Schema is almost impossible because the inheritance has been removed. Therefore, the STEP community continues to use EXPRESS.

The STEP geometry definitions were completed and published in 1994 and have been adopted by all the STEP application protocols (see Appendix). An application protocol is an implementable specification within the STEP ISO 10303 series of standards. As shown in the Appendix, there are many application protocols, but there are also other specifications, such as the EXPRESS language, XML and ASCII file formats, and shared resource schemas for geometry, part information, materials, features, tolerances, and more. Most mechanical CAD stations now have an AP-203 or AP-214 translator. Debugging and resolving the conflicts between the vendors so that STEP geometry became widely available was a major exercise that might not have been possible without EXPRESS and its precise mathematical definitions of CAD geometry.

STEP-NC eliminates the barrier to describing 3D geometry data by harvesting the work done for the STEP standards. By adopting the EXPRESS definitions of 3D geometry and corresponding STEP CAD data, STEP-NC can describe models for workpieces, stock,

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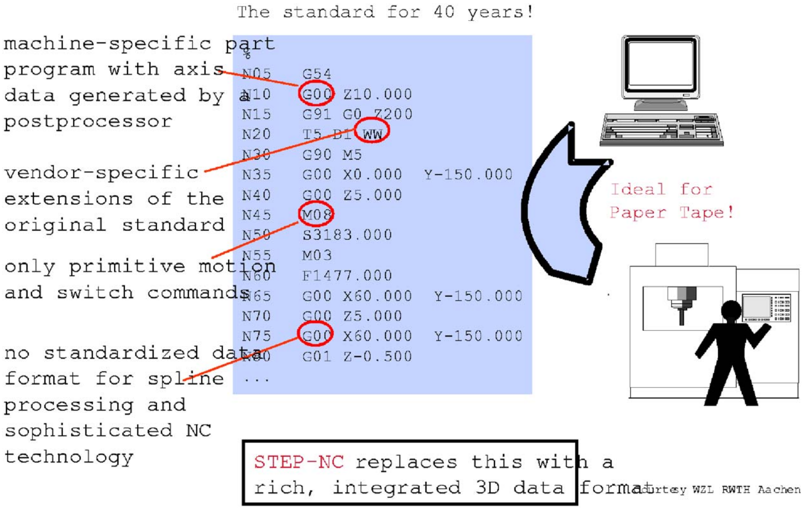


```
%
N05   G54
N10   G00 Z10.000
N15   G91 G0 Z200
N20   T5 B1 WW
N30   G90 M5
N35   G00 X0.000   Y-150.000
N40   G00 Z5.000
N45   M08
N50   S3183.000
N55   M03
N60   F1477.000
N65   G00 X60.000  Y-150.000
N70   G00 Z5.000
N75   G00 X60.000  Y-150.000
N80   G01 Z-0.500
...
```

Fig. 1 Some limitations of the RS274D Standard

fixtures, cutting tool shapes, and sophisticated tool-path curves.

Factor 2: A Linked Model for CNC-Controlled Processes

CAM and CNC processes are controlled by a large number of interlinked operations and parameters. In RS274D, these operations and parameters are specified by a language but never explicitly linked to one another. As a result, simulation systems must execute entire programs to understand the state at a given moment. Likewise, machine tool controls cannot simply jump to an arbitrary location in a program, but must be run block by block to ensure that all necessary context and state is correct.

The core of the STEP-NC standard is a new model for CNC control. This process model was originally developed as a standard called ISO 14649 [5] by an IMS project led by Siemens with contributions from the University of Aachen and the University of Stuttgart in Germany, Komatsu and Fanuc in Japan, Heidenhain in Switzerland, and the Pohang University of Science and Technology in Korea [6]. The first model was for the control of CNC milling machines, but subsequent models have been developed for control of turning and electrical discharge machining (EDM).

In STEP-NC, each operation is linked to all of the other information in the part program by the relationships shown in Fig. 3. Given any operation, an application can find its tooling requirements, the parameters, geometry, and tolerances of its feature, as well as any necessary strategy and technology information. Therefore, algorithms can be written on a STEP-NC controller to make the machine tool faster and more intelligent.

Figure 3 gives an outline of the model as an EXPRESS-G diagram [3]. In the top left corner, the workpiece geometry defined by STEP is shown as a single box labeled geometry. Below this a series of machining features are available. These features are the same as those defined in the STEP AP-224 protocol. The rest of the diagram outlines the new information defined by the STEP-NC model.

A STEP-NC part program is a work-plan executable that contains a series of working steps. Each working step applies a machining operation to a manufacturing feature, for example, a rough milling operation to a pocket. A machining operation may

```
ENTITY point
  SUPERTYPE OF (ONEOF (cartesian_point, point_on_curve, point_on_surface,
    point_replica, degenerate_pcurve))
  SUBTYPE OF (geometric_representation_item);
END_ENTITY; -- point

ENTITY b_spline_curve
  SUPERTYPE OF (ONEOF (uniform_curve, b_spline_curve_with_knots,
    quasi_uniform_curve, bezier_curve) ANDOR rational_b_spline_curve)
  SUBTYPE OF (bounded_curve);
  degree              : INTEGER;
  control_points_list : LIST [2:?] OF cartesian_point;
  curve_form          : b_spline_curve_form;
  closed_curve        : LOGICAL;
  self_intersect      : LOGICAL;
DERIVE
  upper_index_on_control_points : INTEGER := SIZEOF(
                                   control_points_list) - 1;
  control_points               : ARRAY [0:
                                   upper_index_on_control_points] OF
                                   cartesian_point := list_to_array(
                                   control_points_list,0,
                                   upper_index_on_control_points);
WHERE
  WR1: ('CONFIG_CONTROL_DESIGN.UNIFORM_CURVE' IN TYPEOF(SELF)) OR
       ('CONFIG_CONTROL_DESIGN.QUASI_UNIFORM_CURVE' IN TYPEOF(SELF)) OR
       ('CONFIG_CONTROL_DESIGN.BEZIER_CURVE' IN TYPEOF(SELF)) OR
       ('CONFIG_CONTROL_DESIGN.B_SPLINE_CURVE_WITH_KNOTS' IN TYPEOF(SELF));
END_ENTITY; -- b_spline_curve
```

Fig. 2 STEP definitions for point and B-spline curve written in EXPRESS

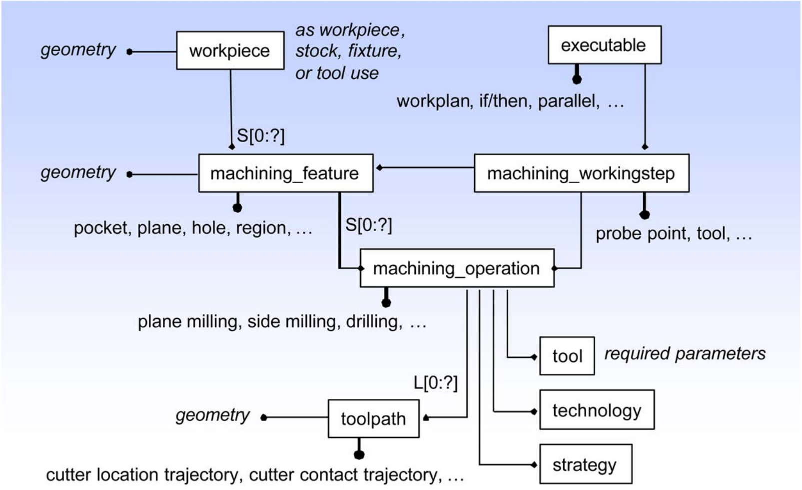


Fig. 3 EXPRESS-G **description of the STEP-NC process model**

be used in several working steps, and several working steps may be necessary to complete a feature. For example, completion of a pocket may require a roughing operation to be followed by a finishing operation. Work plans may be nested, and other executables types can be conditional on the result of probing operations, can run concurrently on machines with multiple cutting heads, and make use of other types of control flow.

Each working step in a STEP-NC part program is a complete description of an operation that does not require context from previous operations. As shown in Fig. 3, each operation includes a reference to a cutting tool, technology parameters, and any necessary strategy information. An operation may also include an explicit tool-path description. A CAM system may choose to send just the operational parameters, just the tool path, or both to the CNC. STEP-NC operations can be simulated individually using in-process models of the stock and workpiece, and the order of a programing sequence can be changed without invalidating the rest of the program.

Factor 3: Integrating the CAD, CAM, and CNC Models

The developers of the ISO 14649 process model used EXPRESS because they needed to make extensive use of inheritance and wanted to reuse the STEP definitions for geometry, but did not develop a STEP application protocol like AP-203.

A STEP application protocol is created in two stages. In the first stage, the requirements of the application are captured as an object model. In the second stage, these requirements are mapped into the STEP integrated resources. This second stage has always been controversial because of its complexity, but it is necessary for harmonizing data across APs, eliminating redundancy, and allowing models to be improved without disrupting existing implementations.

The integration of STEP-NC began in the USA in 1999 with the model-driven intelligent control of manufacturing (MDICM) project funded by the National Institute of Standards, Advanced Technology Program (NIST ATP). The work was performed by STEP Tools, Inc., in conjunction with an industrial review board [7]. The new protocol was assigned the number AP-238 and submitted for initial technical review in 2001. In 2004, the draft standard was submitted for business case review by all the member countries of the International Organization for Standardization (ISO) and the full international standard was published by ISO in April 2007.

STEP-NC was integrated by AP-238 for two reasons. The first was to enable more data sharing between CAD, CAM, and CNC applications for design to manufacturing pipelines. As shown in Fig. 4 in a typical pipeline, Design creates the specification for a product as a 3D model, usually with a CAD system. Path planning generates tools paths for the CNC machines, usually with a CAM system. Manufacturing controls production using a CNC system. In many enterprises, the CAD and CAM functions are combined into one integrated CAD/CAM system, but in nearly all cases the CNC function is performed by a separate system.

In Fig. 4, integration allows data represented by AP-203 second edition (AP-203 e2), AP-214, or AP-224 to be reused in AP-238. Integration is important because, for most industries, the Fig. 4 pipeline is a simplification that only shows the main data flow for parts that are going to be machined. In the more general case, other functions, such as subcontracting, quality control, and downstream maintenance, must also be considered. For example, if a decision is made to manufacture the part using castings or forgings, then use of the STEP protocols AP-223 and AP-229 becomes appropriate and the AP-203, AP-214, or AP-224 data must first be diverted to one of these protocols before being applied to AP-238 for machining.

Figure 4 uses AP-203 e2, AP-214, or AP-224 to transmit information so that tolerances can be sent from CAD to CAM and then to the CNC. AP-203 edition 1 only enabled the exchange of nominal solid model information; thus, it is an improvement over IGES (Initial Graphics Exchange Specification), which only enabled the exchange of drawing information but does not enable the transmission of tolerances. The newer APs all share a harmonized tolerance model. Figure 4 shows AP-203 edition 2 being used because it is the most popular solution within the aerospace industry, but will not be an ISO standard until the end of 2008. AP-214 would be the most popular, but the CAD vendors are waiting to implement its tolerance definitions until AP-203 e2 is finished. Therefore, early implementations of STEP-NC use AP-224 to get tolerance and feature data because AP-224 translators are available from specialty vendors.

The second reason for integration is to provide better definitions for the data. The developers of the CNC model were CNC experts, not tolerance and feature experts. The expertise required for these kinds of modeling existed elsewhere in STEP. For example, the CNC modelers knew that probing operations executed on the CNC control would need to access tolerance information. However, the tolerance model put into the ISO 14649 model was for machining parameters and does not meet the requirements of design engineers who want to define many kinds of geometric and parametric tolerances, tolerance zones, data, priorities, and more.

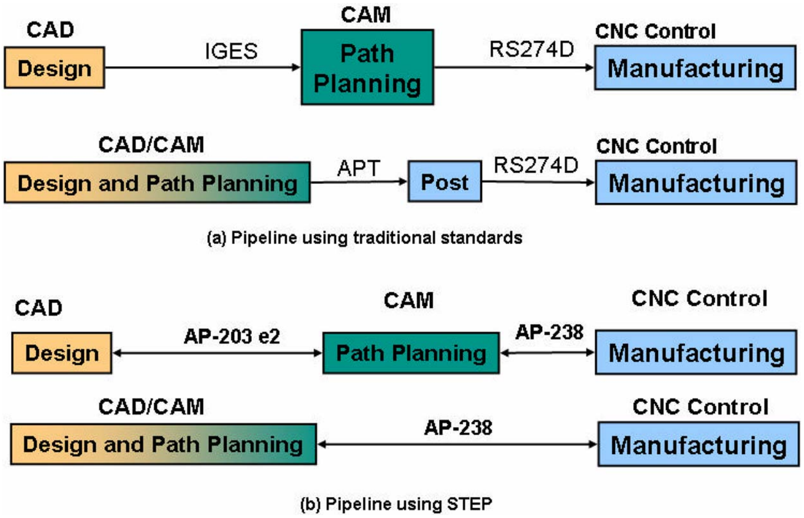


Fig. 4 Design to manufacturing pipelines

Factor 4: Simplifying Implementation Methods

The long list of application protocols shown in the Appendix proves that STEP has a development methodology that allows domain experts to create models that cover the requirements of many application domains, but a lack of implementations for many of the protocols shows that STEP does not have a methodology that allows programmers to easily implement those models.

This barrier was anticipated by the MDICM program. Most of the application protocols listed in the Appendix became complicated because they combine information from multiple application domains. For example, AP-224 combines design geometry and manufacturing feature information, AP-210 combines design geometry and printed circuit board information, and AP-218 combines design and ship structural information. AP-238 is even more ambitious because it combines design geometry, CAM process, and CNC control information. Therefore, we knew that a method to simplify implementation was needed before this protocol could be successfully deployed.

Part of the solution has been to develop an XML data format for STEP called Part 28 [8]. The APIs of STEP XML and STEP ASCII, called Part 21, are quite similar, but XML is more widely available and there are general purpose tools that can process XML. Therefore, generic tools, such as file viewers and checkers, can be made easier to use by adopting the XML format instead of Part 21.

The STEP models are integrated using mapping tables. Each mapping shows how a concept (object or property) is represented by instances of the STEP integrated resources (as defined in the Part 40 series). When two application protocols contain similar information, the information is integrated by mapping it to the same integrated resource entities. The goal of integration and the mappings is to maximize data sharing between the protocols so that end users need never have to enter the same data twice. For example, in AP-238, the integrated definitions allow a CAM programmer to define the boundary of a removal volume by identifying a loop on geometry previously defined in a CAD system.

Figure 5 shows a mapping example taken from the AP-238 specification. This mapping describes how a working step is connected to an operation using the STEP integrated resources. The mapping allows for the simple case where a new operation is created immediately for each working step, and the more complex case where a process planning application defines the requirement for an operation (using AP-240) and a CNC application assigns a working step to perform the operation at a later time.

The notation shown in Fig. 5 says that an action method entity that represents a working step should be connected to an action method entity that represents an operation (or one of its many subtypes) by an action method relationship entity. In the past, a programmer would have to study these mapping paths for each concept and write the corresponding code to assemble or interpret this graph of entity instances. AP-203 only contained 41 objects with these mappings, and most applications only implemented about ten of them, thus, it was not such an issue. The AP-224 process planning standard contains about 237 objects that require these mappings, and the AP-238 CNC machining standard contains 416 with each object on average requiring about five mappings for its five attributes.

Consequently, implementing the STEP APs has become harder because they have grown larger. The original STEP tool kits used EXPRESS compilers to automate the development of an API. These compilers generated code for the inheritance trees in the EXPRESS models and were well suited to implementing data exchange applications for CAD geometry. However, apart from generating get and put methods for the action method and action method relationship entities shown in Fig. 5, they did almost nothing to help the implementation of mappings.

In the last four years, we have developed a new type of API for STEP-NC. This API captures the rules described by the mapping tables and converts them into high-level code that manages the set of entities used in each mapping. The API makes it possible to implement functionality at both the application requirement model (ARM) level of STEP and the application integrated model (AIM) level of STEP. Most of the coding is done at the ARM level but the AIM is important when a resource such as a geometrical tolerance must be manipulated, in detail.

```
          machining_workingstep <=
        machining_process_executable <=
              action_method <-
action_method_relationship.relating_method
          action_method_relationship
       { action_method_relationship =>
       machining_operation_relationship }
action_method_relationship.related_method ->
              action_method =>
             machining_operation
```

Fig. 5 STEP AP-238 mapping for working step to operation relationship

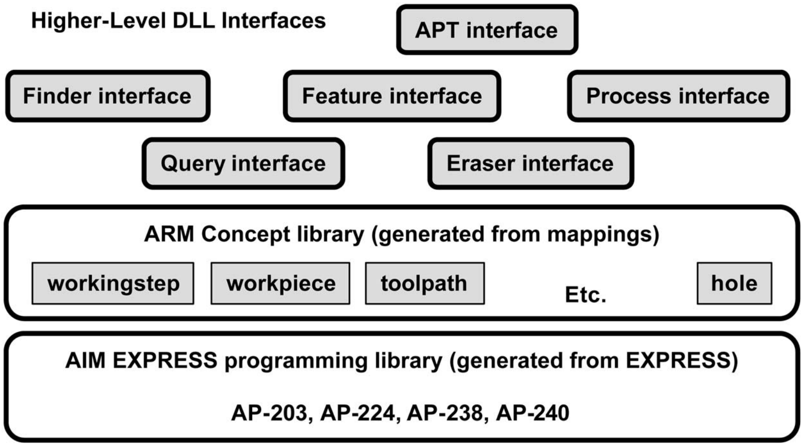


Fig. 6 New implementation architecture

The new API is illustrated in Fig. 6. An ARM object model and mapping tables are extracted from the AP specification and compiled to produce an ARM class library. This library is layered on top of the existing AIM class library. It provides services to create ARM concepts and populate the underlying AIM instances, analyze an AIM data set to identify the available ARM concepts, and access the individual AIM instances that make up a given ARM concept.

The new API is packaged as a dynamic link library (DLL) that can be included in any application. The DLL contains a large range of methods for implementing simple and complex functionality. An example of a simple method would be a function call to change the value of the speed used for a tool path. An example of a complex method would be a function call to read an AP-203 file and use the geometry defined in that file to define the workpiece, stock, or fixture of a STEP-NC program.

As demonstrated by the work described in the next section, the new technology has made it possible to implement a relatively large range of methods more quickly than was possible using the EXPRESS tool kit method. We predict that it will allow STEP vendors to implement semantic functionality for all of the STEP AP's in the next five years and enable much wider use of these standards across the industry.

Example: Measuring Accuracy on a CNC

New aerospace systems, such as the Boeing 787 or Airbus A350, require parts to be machined to higher tolerances with fewer mistakes because designs and materials continue to become more sophisticated. Consider the assembly of structural components for a wing. These components may now be made of titanium instead of aluminum. Titanium is stronger, thus the parts can be smaller and lighter, but often the features and tolerances must be tightened if the part is to deliver the same functionality.

When the titanium parts reach the assembly plant, they are assembled to create the frame for the wing. Consider an operation that adds countersinks to holes so that they can be used for fasteners in an assembly. The holes were drilled into the part by the subcontractor, and when placed on the assembly, they must have locations and dimensions that are within the tolerances defined by design. Note that these locations are relative to the assembly, not their locations on the parts drilled by the subcontractors.

Manual checking of the assembled position of the holes by the operators would require a great deal of concentration because every airframe is slightly different. The operator must look at numbers that are in the coordinates of the airframe and detect variances that can be as small as 0.001 in. This is quite difficult, thus, either a CMM or a CNC application must be programmed to detect the errors before the countersinking operation is started. Using a CMM machine would require a second setup; thus, a test application has been built to see if the job can be performed by a CNC using a probe and STEP-NC AP-238 data.

Figure 7 shows an experiment we created to explore this application. The experiment assumed the data to perform the checking were already in the CAD and CAM systems of the enterprise. It further assumed that within the CAD systems, the data are parametric and proprietary because the formulas that decide how and

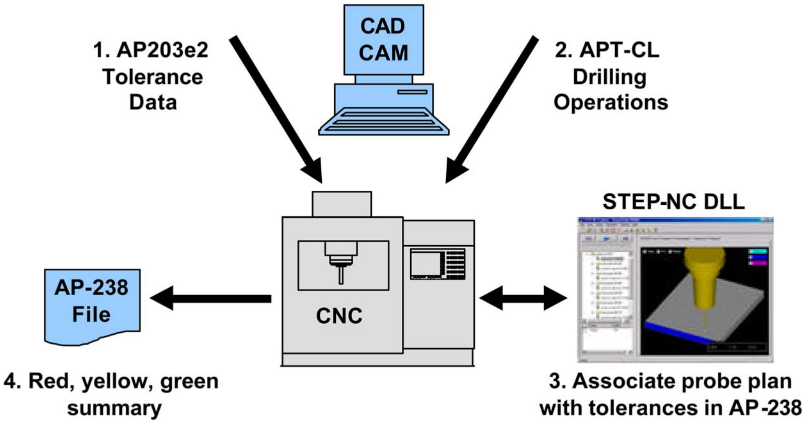


Fig. 7 Architecture of the tolerance application

where to place each component for a given aircraft configuration are extremely valuable. Hence, an airframe manufacturer does not wish to simply place the CAD/CAM system onto the CNC. STEP-NC is a better solution if it allows the necessary measurements to be made without putting these valuable formulas at risk of being copied by a subcontractor.

The data source for the experiment was the CATIA CAD/CAM system and two types of files were exported: an AP-203 e2 file containing geometry and tolerance data, and an APT-CL file describing the drilling operations. An application program was written using the STEP-NC DLL to match the drilling operations in the APT-CL file against the cylindrical holes in the AP-203 e2 data. APT-CL is a simplified version of the APT language that uses a binary data format to communicate tool-path information from a CAD/CAM system to a post-processor [2]. It is available for Versions 4 and 5 of CATIA and in other forms, such as ASCII from other vendors. After the matching,

1. There is visual information for the operator to check for correct matching. See Figure 8.
2. The exact location and axis of each drill are known in the coordinate system of the airframe; thus, the probing application can probe the same location and axis.
3. The location tolerance and diameter size tolerance are known for each hole because it has been found in the AP-203 e2 file and integrated into the AP-238 file.

In the test, two CNC programs were generated. The first CNC program performed probing operations on each hole. Four probes were generated for each hole to detect the north, south, east, and west boundaries. The second CNC program evaluated the results of the probing. Each hole was labeled "green" if all tolerances were satisfied. If either the position tolerance or the diameter tolerance was exceeded, the operation was labeled "yellow." If both tolerances were exceeded, then a "red" label was given.

Figure 8 shows some screen captures of the demonstration. Figure 8(*a*) shows the toleranced part and probing operations generated from the AP-203 e2 and APT-CL. Figure 8(*b*) highlights one of the operations to illustrate how the probe is moved from the center to a north, south, east, or west boundary and shows the result of the measurement with status colors assigned to individual probe results as well as complete hole features. Figure 8(*c*) shows a detail screen for one of the probing operations so that the operator can decide what to do in the case of ambiguity ("yellowness").

Summary

The four key factors we believe should enable the widespread adoption of STEP-NC AP-238 are as follows:

1. The use of STEP geometry to describe the workpiece, stock, fixtures, cutting tools, and other design components that vary with each CNC program. This geometry makes STEP-NC AP-238 a more visual and easy way to understand process for the nonexpert operator, and enables collision detection and verification on the control.
2. The robust model for CNC machining developed for ISO 14649 and integrated into AP-238 using the STEP methodology. This model allows STEP-NC AP-238 to capture the semantics of legacy CNC applications and add a new layer of functionality in which conditional and parallel operations are programmed.
3. The STEP integration process that allows the simple definition of tolerances defined in ISO 14649 to be expanded into one that can be shared with CAD systems. The expansion enables the full semantic combination of CAD and CAM data on the CNC so that parts can be measured more accurately and errors detected more efficiently.
4. New implementation methods that allow STEP and STEP-NC systems to be built quickly and simply for applications such as the one described above

The first three factors give STEP-NC AP-238 capabilities that have not been available in previous machining standards. The last factor allows us to implement them more easily. Table 1 shows how the factors apply to some key applications.

For the aerospace example in the previous section, just-in-time post-processing allows the STEP-NC program for measuring and machining the holes to be moved to a new CNC. Unlike RS274D, a STEP-NC file is neither machine specific nor fixed to a particular coordinate system. A just-in-time post-processor converts the data to CNC codes just before the data are machined and allows the same STEP-NC file to be used with many different CNC controls.

Easier shop-floor reprogramming allows minor adjustments to the CNC program to be made more easily. For example, if the diameter of a drill hole must be changed, it can be changed in one place and all of the drilling operations that use that hole will also change.

Speed and feed optimization allows the feeds and speeds of a program to be reduced or increased when a program is moved to a new machine. The AP-238 model includes data describing the depth of cut, tolerances, and required surface finish for a part, and applications are being written to use this information to dynamically optimize speeds and feeds when a STEP-NC AP-238 program is moved to a new machine.

With just-in-time simulation, a CNC program is checked as it is executed. Today, simulation and verification systems check RS274D programs, but the operator is required to enter substantial information about the geometry of the workpiece, stock, and cutting tools in order to complete the simulation. With AP-238, all of this information is included in the STEP-NC file, so, the same program can be used safely at many different locations and large enterprises can consolidate their CAM programming into one location.

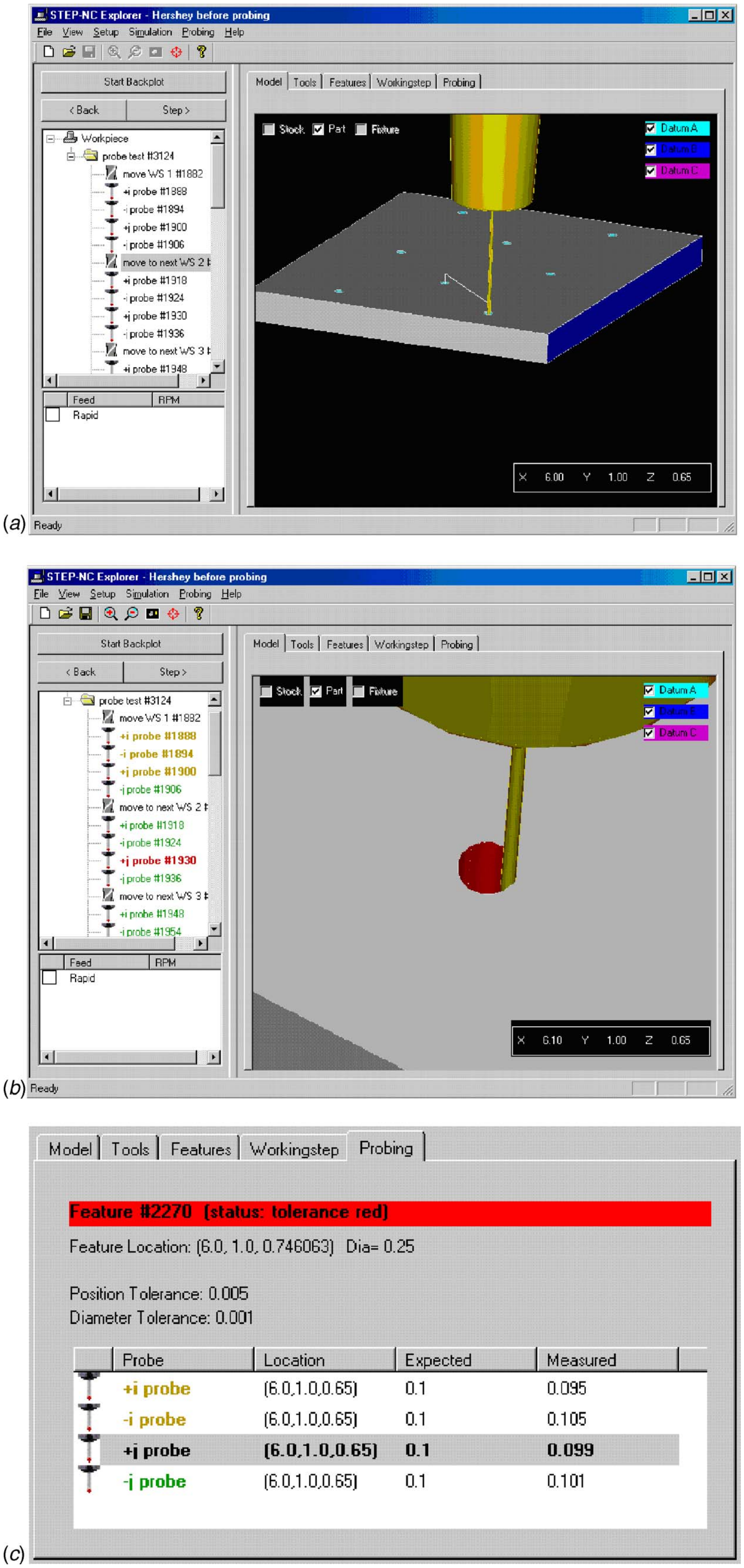
Closed-loop programming allows the quality of the machining process to be checked as it executes so errors are prevented and scrap is avoided. In the previous section, we described an application that allows an aerospace company to make sure that an expensive airframe component has been set up correctly before the part is machined. CAD tolerance data are essential for closed-loop programing because machines are never 100% accurate. Therefore, there are always slight errors and the key question that must be answered is whether or not the error is within the constraints defined by the designer.

Five-axis cutter compensation allows the tool wear to be measured and used to compute a compensating δ that is applied to the tool paths. Three-axis cutter compensation is widely used in traditional machining, but five-axis compensation is more difficult because the direction of the δ depends on the normal between the cutter and the surface. For three-axis machining, this direction is in the XY plane of the machine at a right angle to the current cutting direction of the tool.

If a model of the CAD geometry is included in the CNC program, then the surface normal of the part can be computed for each cutter location and the cross product of this normal with the direction of the tool can be used to compute a direction for the δ. STEP-NC AP-238 allows this geometry to be reused because a transformation can be defined that moves the geometry from its location within the part assembly (such as the back of a ship) to the location required for machining (typically the top center of the part).

Future Work and Concluding Remarks

Because of the initial success and promise of AP-238, new STEP-NC extensions are being investigated for traceability, shipbuilding, composite materials, and metrology. The traceability extension allows a customer to make sure that a part has been ma-



Feature #2270 (status: tolerance red)

Feature Location: (6.0, 1.0, 0.746063) Dia= 0.25

Position Tolerance: 0.005
Diameter Tolerance: 0.001

Probe	Location	Expected	Measured
+i probe	(6.0,1.0,0.65)	0.1	0.095
-i probe	(6.0,1.0,0.65)	0.1	0.105
+j probe	**(6.0,1.0,0.65)**	**0.1**	**0.099**
-j probe	(6.0,1.0,0.65)	0.1	0.101

Fig. 8 (a) Toleranced part and probing operations, (b) colors for individual probe results and complete features, and (c) probing results for a particular hole feature

Table 1 Comparison of AP-238, ISO 14649, and BCL

Application	BCL solution	14649 solution	AP-238 solution
Just in time post processing on the CNC	BCL was developed for this application.	ISO 14649 also has a solution for this application	AP-238 is a third solution.
Shopfloor reprograming	BCL does not allow for CAM reprograming.	ISO 14649 was designed for this application.	AP-238 has the same capabilities as ISO 14649.
Speed and feed optimization	BCL does not include CAD tolerances.	ISO 14649 does not include CAD tolerances.	AP-238 includes CAD tolerances.
Just in time verification on the CNC	BCL does not include CAD geometry.	ISO 14649 has a geometry model for the workpiece only.	AP-238 has geometry models for all of the required components
Measurement on the CNC	BCL does not allow for geometry and tolerances.	ISO 14649 does not allow for CAD tolerances.	AP-238 includes the CAD tolerances and geometry required for this application.
Five-axis cutter compensation	BCL does not include geometry so the normal to the surface cannot be computed.	ISO 14649 has a workpiece model but does not allow the CAD location to be transformed to the CNC location.	AP-238 allows the CNC position of the workpiece to be defined precisely so that the normal can be computed.

chined using the procedures and measurements required by its designers [9]. This is important in the aeroengine industry, for example, where many parts are made in-house or only at trusted suppliers. STEP-NC makes CNC traceability more practical because the trace can be explained using the high-level concepts in the STEP-NC model. The traceability extension should enable industries with high trust requirements to manufacture more components at a wider range of suppliers.

Within the shipbuilding industry, it is desirable for subcontractors to fabricate frames, compartments, and substructures. Today, this is inefficient because essential manufacturing preparation data must be communicated using drawings, and there are too many opportunities for mistakes. The full fidelity simulation of the product and process enabled by STEP-NC will allow a subcontractor to check operations and eliminate these mistakes. Plate cutting and marking using plasma torches, lasers, and waterjets is currently being investigated as an extension to STEP-NC by the U.S. shipbuilding industry. Of course, fabrication of parts requires more than plate cutting. Robotic welding must also be considered, and a STEP-NC project for the control of robots may be initiated in the future.

Composite materials are now being used in many airframe components. The machines used to lay composite tape are large, expensive, and often interfaced to CAD/CAM systems using ad hoc methods. Applications may require simultaneous movement of multiple heads in order to lay sufficient tape and on-line quality checking to avoid error buildup. STEP-NC has parallel programing constructs for controlling multiple functions simultaneously, and it allows the geometry of the product and tooling to be transmitted to the machine so that it can be used for parallel program collision detection. Consequently, a project to investigate STEP-NC process attributes for the control of tape laying machines may also be started shortly.

One of the key features of STEP-NC AP-238 is that it enables on-machine measurement of the as-designed part. Another extension is looking at extending this capability for high-level planning of metrology operations that may be implemented on CMMs and CNCs. An early experiment has already shown that simple DMIS programs can be generated from AP-238 programs. A DMIS-like model is currently being added to the second edition of AP-238 so that goal-oriented measurements can be defined for the CNC and then implemented using a variety of technologies, such as laser tracker balls, white light scanning, and probing.

Finally, for most new standards, end users must wait for appropriate support by vendors. This is not so for STEP-NC AP-238 applications because AP-203 e1 or e2 data can be obtained from a CAD system, APT-CL data can be obtained from a CAM system, and the two can be integrated using the STEP-NC DLL. Tolerance and probing data can then be added by upgrading to AP-203 e2 when it becomes available, or by using the options of a user interface, such as the one shown in Fig. 8.

Because of all these extensions and the early deployment successes, we believe the four factors discussed in this paper will allow STEP-NC AP-238 to be deployed for high-end, high-precision, five-axis machining applications. Initially, STEP-NC will be deployed as a systems integration task that combines existing tool-path and geometry data sources. When necessary, missing data will be added via a user interface until it is directly available from the CAD and CAM vendors. As these applications become more widespread, they will spur the development of new solutions that further reduce the cost and increase the benefits of using STEP-NC, promoting a virtuous circle where continuing improvements lead to more applications and more deployment by the vendors until even the most routine three-axis machining jobs use STEP-NC.

Appendix: STEP Application Protocols

Each application protocol defines a data exchange standard. Only some of the protocols have an impact on end users. Others establish a capability that will be harvested by a subsequent standard.

AP 201 Explicit Drafting
AP 202 Associative Drafting
AP 203 Configuration Controlled Design of Mechanical Parts and Assemblies
AP 204 Mechanical Design Using Boundary Representation
AP 205 Mechanical Design Using Surface Representation
AP 206 Mechanical Design Using Wireframe Representation
AP 207 Sheet Metal Dies and Blocks
AP 208 Life Cycle Product Change Process
AP 209 Design Through Analysis of Composite and Metallic Structures
AP 210 Electronic Printed Circuit Assembly, Design and Manufacturing
AP 211 Electronics Test Diagnostics and Remanufacture
AP 212 Electrotechnical Plants
AP 213 Numerical Control Process Plans for Machined Parts

AP 214 Core Data for Automotive Mechanical Design Processes
AP 215 Ship Arrangement
AP 216 Ship Molded Forms
AP 217 Ship Piping
AP 218 Ship Structures
AP 219 Dimensional Inspection Process Planning for CMMs
AP 220 Printed Circuit Assembly Manufacturing Planning
AP 221 Functional Data and Schematic Representation for Process Plans
AP 222 Design Engineering to Manufacturing for Composite Structures
AP 223 Exchange of Design and Manufacturing DPD for Castings
AP 224 Mechanical Product Definition for Process Planning
AP 225 Structural Building Elements Using Explicit Shape Rep
AP 226 Shipbuilding Mechanical Systems
AP 227 Plant Spatial Configuration
AP 228 Building Services
AP 229 Design and Manufacturing Information for Forged Parts
AP 230 Building Structure Frame Steelwork
AP 231 Process Engineering Data
AP 232 Technical Data Packaging
AP 233 Systems Engineering Data Representation
AP 234 Ship Operational Logs, Records and Messages
AP 235 Materials Information for Products
AP 236 Furniture Product and Project
AP 237 Computational Fluid Dynamics
AP 238 Integrated CNC Machining
AP 239 Product Life Cycle Support
AP 240 Process Planning
AP 241 Life Cycle Model for AEC Facilities

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