OntoSTEP-NC for Information Feedbacks from CNC to CAD/CAM Systems

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Abstract. This paper exposes a proposal to ensure manufacturing feedbacks from the CNC machine to CAD/CAM systems by using an ontology through information systems. Centered on the Manufacturing Process Management platform, the trades between Product Data Management and Enterprise Resource Planning platforms are based on the new semantic model OntoSTEP-NC – an ontology based on STEP-NC standard. This defines the Closed-Loop Manufacturing which provides and allows a data extraction from the CNC machine and reinjection of relevant information to the CAM systems. This would help CAM programmers making choices based on company good practices stored in the database.

Keywords: STEP-NC; OntoSTEP-NC; Closed-Loop Manufacturing, Manufacturing Process Management.

1 Introduction

To be more competitive and due to the globalization context, aeronautical manufacturers tend to improve the triptych: Cost, Time, and Quality.

This is why French FUI project called ANGEL (Atelier Numérique coGnitif intEropérable et agiLe) focuses on the capitalization of cuts know-how in order to improve the competitiveness of companies developing tools and methods to retrieve information from the CNC machine. To achieve the information flow bi-directionality from CNC machines to CAD, the systems must be able to exchange information and to use this information.

Defined as “The ability of two systems (or more) to communicate, cooperate and exchange services and data, thus despite the differences in languages, implementations, executive environments and abstraction models” [1], interoperability is described by 3 levels: Semantical, Technical and organizational [2]. The use of standard format which allows having a unified approach can be one of the various solutions for

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interoperability. To achieve the interoperability between the CAX software the STEP-NC standard seems to be one of the most promising [3].

Beyond interoperability aspect this paper focuses on the feedback from the machining to the design and industrialization phases. The main question the paper would answer is: "How to integrate manufacturing expertise from manufacturing in the design / industrialization process for mechanical parts?". Indeed incrementing the cutting knowledge would help to manufacture the right part-the first time thus despite the files formats differences.

To answer this question, the next section presents a state of the art on STEP-NC standard as a basis for CAX software trades. The third section will explain the proposal to achieve the integration of manufacturing expertise and knowledge extracted from the CNC machining. Section 4 concludes this paper presenting future works.

2 State of the art

2.1 STEP-NC a solution for data exchange

Designing and manufacturing systems trades (CAD, CAM, Post-Processor, CNC machine, etc.) are led by specific file. Indeed, software has its own language and makes interoperability more difficult. For example, CATPart and CATProcess will be used respectively for Dassault Systems CAD and CAM systems, Top'CAM for Top-Solid CAM system, G-Code and M-Code will be used as specific inputs according the CNC controller, etc.

Therefore to ensure transaction between software, one solution consists in using standard format. Indeed this unifying approach allows having the same language for all the manufacturing technologies. To first achieve the data exchange between CAD and simulation, STEP standard has been developed (Lee, S.-H. et al., 2006). The STEP standard is an open and normalized standard that aims to promote the data exchange in a format which is understandable and shared by all. According to [5], the STEP standard provides a neutral, sustainable and scalable data exchange format.

In the last years, STEP-NC, a new standard format with enriched data has been developed in order to improve the systems interoperability [6] by integrating processing data. Indeed, the STEP-NC standard encompasses machining process, cutting tools description, and CAD features and requirements. This enriched standard format allows having in the same file all the information required for the whole development stage from the early design to the machining.

STEP-NC is led by two standards which are interested in two different levels – AIM (Application Interpreted Model) and ARM (Application Reference Model):

- ISO 10303 AP238 which concerns the AIM level. This level is based on the ISO 10303 standard which defines the STEP standard. The Application Protocol (AP) 238 untitled "Computer Numerical Controllers" mainly allows adding information for CNC machining. In this way, the STEP standard is enriched with the manufacturing feature.

- ISO 10649 deals with the ARM level. This level is higher than the previous one and it also defines the machining strategy.

The STEP-NC standard structures a large number of information. Therefore, the same file can be used for all the CAX systems and then all the modifications are propagated from one to another. The use of STEP-NC also helps to archive the modifications. Indeed there is no more coherence problem from CAM and CNC program. All the modifications and optimization made by the operators in the CNC machine are translated in the STEP-NC file. This program can be then archived and re-used if necessary. The use of STEP-NC can help to save time because the post-processor can be overpassed. Indeed, the machining intelligence is moved from CAM systems to CNC machines thus the translation from computer language to machine understandable language (Post-Processors and G-Code) is no more necessary. Therefore, the following works will be based on STEP-NC standard and not on G-code defining the information flow through the CAX systems from CAD to CNC [7].

2.2 STEP-NC enabling feedback, cooperation and optimization

As seen in the previous section, the use of STEP-NC standard file makes possible the propagation of modifications to all the other manufacturing systems. Therefore changes made directly in the CNC machine will be propagated back to CAM system.

In addition to the flow bi-directionality, STEP-NC standard will simplify the industrialization phase. As a result of optimization, according to the NIST, the STEP standard can potentially save up to a billion dollars a year by reducing the costs of interoperability in sectors such as automotive, aerospace and shipbuilding. The following list summarizes the major works using STEP-NC for feedback and optimization:

- [8] achieve vertical integration with STEP-NC in order to have a standard process monitoring and traceability programming. The traceability is ensured at three different levels: Business level, Manufacturing Level, and shop floor level. This allows monitoring the capitalization.
- [9] with DIMP system based on STEP-NC provides more flexibility for cooperative manufacturing environment.
- [10] use STEP-NC as a universal programming for CNC machines. Indeed the same CAM program can be spread to many CNC machines. Thus it has been made possible with the intelligence transfer from CAM system to CNC machines to use the same CAM program into many different CNC machines.
- Through the use of STEP-NC [11] define an automatic correction of cutting parameters based on the Machine Condition Monitoring. They have developed optiSTEP-NC system which helps to perform cutting parameters optimization.
- In the same way, [12] has defined Closed-Loop Machining thanks to STEP-NC to achieve on-line inspection. To succeed this inspection they have developed a closed-loop between CAPP and CNC machine.

- Borgia [13] based on STEP-NC allows having automatic recognition of feature and generating toolpath based on machining working step. Then a mathematical optimization is conducted.

According to this short review on STEP-NC works, it appears that the standard is a promising solution for bi-directional trades and for optimization (Cutting parameters, toolpath optimization, feature recognition, etc.).

2.3 Lacks in STEP-NC

The previous study has highlighted that automatic correction/optimization and feature recognition are made reality thanks to STEP-NC use. But there is no feedback from the CNC machine to the CAD or CAM systems. This kind of feedback would help people in design or program choices. Indeed [14] formulated the following assertion: "Interoperability will enable manufacturing businesses to produce legacy components, based on the original process planning knowledge, on modern and future machine tools without the overhead of re-planning the fixturing, tooling and tool paths. This will enable future parts to be manufactured with confidence, as and when required without having to rely on the original equipment, past tooling and part programs which would be typically obsolete." Based on this assertion, it clearly appears STEP-NC is the basis for future data exchanges and to interoperability between CAX systems.

Although STEP-NC allows sharing and propagating data in both ways – CAD to CNC and CNC to CAD – STEP-NC is not yet a solution for archiving CNC machining information in order to be injected at the right time. This is why the next section will propose a solution to insure interoperability and a way to succeed in feedback from CNC machine to CAM systems. This feedback will create the Closed-Loop Manufacturing.

3 Solution for data feedback

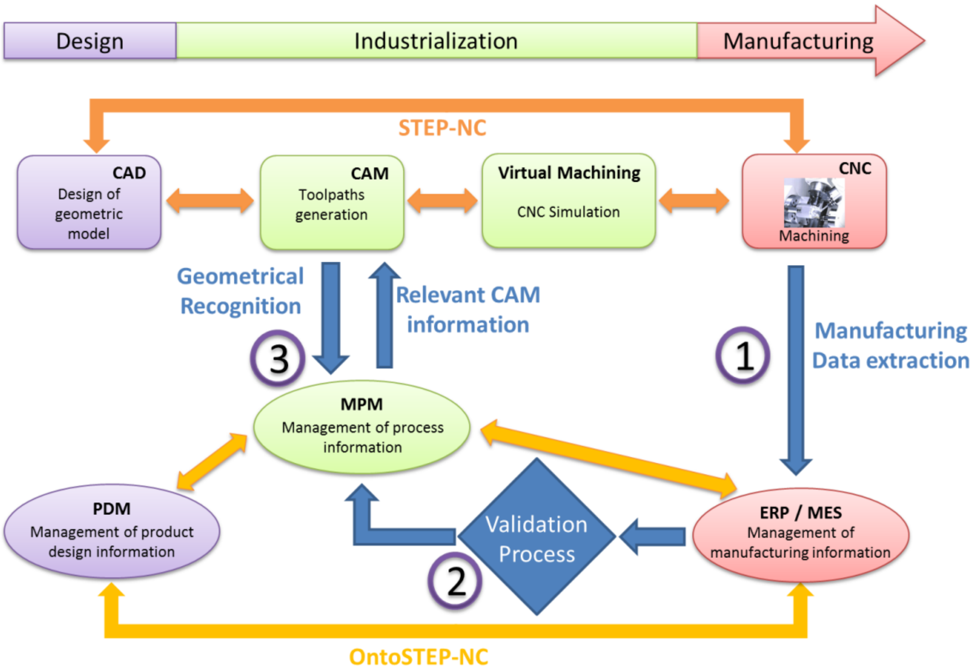
3.1 Information feedback trades

As highlighted in the previous sections, STEP-NC standard file encompasses lot of information and permits the data propagation to the whole lifecycle. But STEP-NC does not allow incrementing manufacturing loops. In contrary to the Closed-Loop Machining which concerns a real time correction of machining parameters including the simulation, the Closed-Loop Manufacturing includes the CAM systems, simulation and CNC machines. Moreover this loop doesn't provide a real time feedback but allow incrementing the knowledge from the CNC machine. In fact, the *n* previous loops would impact and infer on the loop $n+1$.

As STEP-NC file in its own format cannot support the incrementing evolution of good practices. STEP-NC requires to be connected to information systems (PDM-MPM-ERP) to ensure traceability hence to improve the industrialization phase for the future manufactured products. This is why figure 1 exposes a proposition to achieve incrementing manufacturing loops. This implementation is being made possible thanks to the interaction between information systems and the CAX systems.

The proposal contains 3 stages in the Closed-Loop Manufacturing which can be clearly identified: The first one concerns the data extraction from the CNC machine, the second is about the validation process of good practices and third one deals with the geometrical and manufacturing recognition.

Fig. 1. Closed-Loop Manufacturing for data feedbacks



3.2 OntoSTEP-NC for PDM-MPM-ERP trades

To achieve interoperability between information systems it has been chosen to use an ontology. In fact it has been chosen to use an ontology called OntoSTEP-NC based on the STEP-NC standard as support PDM-MPM-ERP trades [15]. The ISO 14649 standard has been chosen as a basis for OntoSTEP-NC model. The standard format is a meta-level model for CNC information so on OntoSTEP-NC describes a meta-level data. This model has been built using the Protégé software (edited by Stanford University) [16] defining entities as classes. Figure 2 and figure 3 describe the OntoSTEP-NC entities and its representation.

First based on ARM level the OntoSTEP-NC can be then modified to AIM model if necessary. Indeed the ontology lets a large range of possibilities to modify the model structure and to add new entities and features. Therefore the interoperability of the information systems is achieved thanks to the adaptability of the ontology model to perfectly fit with the data exchange between PDM, MPM and ERP. Moreover, the use of the recent developed OntoSTEP-NC will allow interoperability with CAX systems as explained by [14]: “Though STEP-NC has given the opportunity for machining process information to be standardized, the lack of a semantic and ontology representation makes it almost impossible to inter-relate the existing systems and languages.”

Fig. 2. OntoSTEP-NC graph

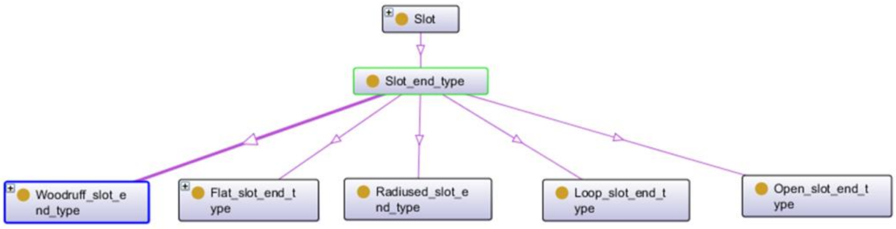


Fig. 3. STEP-NC comparison between EXPPRESS (1) and OWL language (2)

```
ENTITY slot_end_type
ABSTRACT SUPERTYPE OF (ONEOF (woodruff_slot_end_type, radiused_slot_end_type,
  flat_slot_end_type, loop_slot_end_type, open_slot_end_type));
END_ENTITY;

ENTITY woodruff_slot_end_type
  SUBTYPE OF (slot_end_type);
  radius:                   toleranced_length_measure;
END_ENTITY;
```

1

```
<SubClassOf>
    <Class IRI="#Woodruff_slot_end_type"/>
    <Class IRI="#Slot_end_type"/>
</SubClassOf>
```

2

To improve the information feedback, OntoSTEP-NC is being developed as a database in which all the extracted data could be contained. As explained in Lee’s work [17], the knowledge is not set in a relational database but organized according to the characteristics (class, properties, semantical links, attributes…) of the ontologies. This organization guaranties the unicity of the information in the database depending the process the feature belongs to.

To search information in the ontology, queries will be used and issue queries in query languages such as OWL-QL. This standard language for ontologies query offers the ability to export it to other ontologies if needed for future development by combining many ontologies to OntoSTEP-NC. For example, OntoSTEP-NC could be connected to ONTO-PDM [18] which is an ontology for ERP platforms and could also be connected to OntoSTEP [19] for the PDM and CAD platforms.

Moreover, according to [20] semantic queries enable to best capture a user’s ontology requirement and rank the resulting ontologies based on their conceptual closeness to the given query. Hence, the information could be spread to all the manufacturing technologies which require this information. The ontology is the support of manufacturing loops and data feedback from the CNC to CAM systems.

4 Conclusion

As seen in section 2 STEP-NC can support through its rich data format, much information that can be integrated in the bi-directional flow CAD-CAM-CNC. Although STEP-NC allows feedback from the CNC machine to CAD and CAM systems, STEP-NC standard file does not allow archiving and capitalizing information. Indeed, there is no incrementing process in order to capitalize good practices yet existing.

This is why the proposition of this paper uses information systems (PDM-MPM-ERP) to ensure this capitalization setting the MPM as a pivot for trades between information systems and CAX technologies. Those trades are led using OntoSTEP-NC and allow having manufacturing loops.

This proposition will be tested in our future works on three industrial use cases. Those tests will be extracted from aeronautical parts. The first part is an aluminum one which presents a large number of manufacturing features (drilling, pocket, surfacing…). This first part will test the recognition of geometrical design and the ability to compare to the manufacturing features in the database. The second part which is an aeronautical turning-milling part also in aluminum will test the data extraction due to the numerous complex form of this part. The third part equal to the second one will be in titanium. This one will allow testing the increment of the database and also to test the answers from MPM concerning the relevant CAM information trades. The extrapolation of this proposition will concern complex forms for aeronautical parts with raw material as titanium and Inconel.

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