STEP-Compliant CAD/CNC System for Feature-Oriented Machining

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ABSTRACT

STEP-NC is proposed for the next generation of data exchange between CAD/CAM and CNC systems. However, the STEP-compliant CAD/CAM systems and STEP-compliant CNC systems are researched individually in the last decade, which hinders the practical applications of STEP-NC. In the paper, STEP-compliant CAD/CNC system is proposed that consists of STEP-CAD and STEP-CNC, and feature-oriented machining is introduced for 2.5D feature and freeform surface machining, wherein T-spline is introduced for the representation and data exchange of freeform surface. A practical machine tool is constructed under the proposed STEP-compliant CAD/CNC system. Some workpieces are tested to verify the feasibility of the proposed STEP-compliant CAD/CNC system for the feature-oriented machining.

Keywords: STEP-NC, CAD, CNC, Feature-oriented machining, T-spline.
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1 INTRODUCTION

Since numerical control technology has been developed in 1950s, G/M code was used as the data exchange standard between CAD/CAM and CNC machines [12]. However, it is a low-level language that delivers only partial information (axis motion) to CNC that makes CNC isolated without understanding high-level information. It also breaks the CAD-CAM-CNC digital manufacturing chain, and gathering all the information is hardly possible during manufacturing. The information barrier between upstream and downstream of manufacturing hinders the development of next generation CNC system.

STEP-NC is the extension of STEP in the field of numerical control that consists of rich information including 'what-to-make' and 'how-to-make'. There are two versions of STEP-NC are being developed by ISO. The first is Application Interpreted Model (AIM, ISO 10303 AP238), and the other is Application Reference Model (ARM, ISO 14649). AP238 is implemented with the help of exist APs of ISO 10303, such as AP203 for geometric modeling, AP224 for feature based product modeling, etc. However, ISO 14649 created its own machining features description rather than

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using available in AP224 of ISO 10303. In the paper, we focus on the implementation of ISO 14649 for STEP-NC technologies.

With the help of object-oriented programming and neutral description, STEP-NC can exchange the whole product data information throughout the life cycle of manufacturing encompassing CAD, CAM, and CNC. STEP-NC is a perfect technology that can make the designing and manufacturing connected with lossless information. Hence, the traditional CAx systems need to be improved to make them suitable for STEP-NC.

However, STEP-compliant CAx systems are researched individually in the last decade, such as STEP-compliant CAD/CAM systems [7,19] and STEP-compliant CNC systems [9,11]. STEP-compliant CAD/CAM systems focus on the generation of STEP-NC physical files including geometric information, process information, tool information, etc. STEP-compliant CNC systems focus on the CNC controller which can read the STEP-NC file directly and control the machine tool. Therefore, the research combines STEP-compliant CAD/CAM and STEP-compliant CNC is rare due to the complexity of STEP-NC implementation and integration.

STEP-NC is described using object-oriented language named EXPRESS. All manufacturing features are defined using entity *manufacturing_feature* that is the supertype of *two5D_manufacturing_feature* (2.5D features) and *region* (freeform surface), and feature-oriented machining can be introduced. In the current research work, most researchers consider 2.5D features such as holes, pockets, slots, etc. Freeform surface is widely used in industry for manufacturing of the aerospace, automotive, and dies/moulds. NURBS is the only mathematic representation for freeform surface in the ISO standard. However, NURBS has the inherent drawbacks that makes it difficult for the STEP-compliant freeform surface machining.

In the paper, we proposed a STEP-compliant CAD/CNC system that consists of STEP-CAD and STEP-CNC. STEP-CAD can be used to generate the typical STEP-NC files for 2.5D features and freeform surface. STEP-CNC can read the STEP-NC files directly, and have the capability of online toolpath generation for 2.5D features and freeform surface. Beyond that, T-spline is introduced for the representation of freeform surface. A practical machine tool is constructed under the proposed STEP-compliant CAD/CNC system, and some workpieces are tested to verify the feasibility of the proposed system.

The paper is organized as follows. Section 2 presents a literature review about STEP-compliant CAx systems and T-spline. STEP-compliant CAD/CNC system is proposed in Section 3. In Section 4, a practical machine tool is constructed and some workpieces are tested. We end with the conclusion in Section 5.

2 LITERATURE REVIEW

STEP-NC provides a mechanism for a bidirectional data flow for the CAD-CAM-CNC digital manufacturing chain. It can transfer the process planning tasks from CAM to CAD, and it can also transfer toolpath generation and decision-making jobs from CAM to CNC. Hence, the traditional CAx systems need to be improved to make it compatible for the STEP-NC technology. However, STEP-compliant CAx systems are researched individually, such as STEP-compliant CAD/CAM systems and STEP-compliant CNC systems.

STEP-compliant CAD/CAM systems work on the model designing, micro process planning and macro process planning. STEP-NC file is generated by STEP-compliant CAD/CAM systems that consist of geometric, tool, process, technology, etc. Newman et al. proposed STEP-compliant CAD/CAM system firstly, and three frameworks haven been defined based on the complexity of integration, and agent technology is introduced for process planning based on different features such as pockets and holes [7,8]. Yusof et al proposed a STEP-compliant CAD/CAPP/CAM system for turning operation, and the process information can be specified through GUI [20]. Li introduced ontology into the STEP-compliant CAM system, which can generate the optimized toolpath at the shop floor level [5].

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STEP-compliant CNC systems consist of MMI (Man Machine Interface), PLC (Programmable Logical Control), and NCK (Numeric Control Kernel), which can use STEP-NC file as input to control machine tools. Suh et al proposed a STEP-compliant CNC system for turning that is composed of three subsystems: code generating system, code editing system, and autonomous control system [10]. Calabrese et al presented a STEP-NC compliant CNC embedded controller that is designed and realized using a low cost microcontroller [1]. Wang et al. proposed a STEP-compliant adaptable CNC system incorporating the function block technology, which can be used to realize the intelligent, interoperable and adaptive CNC machining [16]. STEP-compliant CNC systems are categorized into three classes: conventional control, new control, and new intelligent control [10]. Until recently, most of the STEP-compliant CNC systems fall into the first and second categories.

STEP-NC gives the details about 2.5D feature machining, and 11 types of 2.5D features are defined in the standard [3], and three examples are provided that only contain 2.5D features [4]. In the current research, most researchers focus on the 2.5D machining, and only 2.5D features are tested [8,10,18]. However, freeform surfaces are widely used to represent complex models in the field of automotive and aeronautics to meet the functional requirements. Freeform surfaces are typically constructed using NURBS that is the only standard for the representation of freeform surfaces [2]. Nevertheless, NURBS has two drawbacks to represent the freeform surfaces, especially the complex surfaces. When using a single surface to represent the complex model, a lot of redundant control points are introduced that only to satisfy the topology of the surface. Trimming and joining surface could represent the complex model using less control points. However, there will be some gaps between two adjacent surfaces. The limitations become the roadblock for the complex surface manufacturing using STEP-NC technology.

Not until the advent of T-spline [13,14], these problems are lack of perfect solutions for STEP-compliant CAx systems to represent and machine freeform surface. T-spline has charming characteristics of less control points, local refinement, and watertight representation. Comparing to NURBS, T-spline can use a single surface to represent model with arbitrary topology without introducing redundant control points, which is convenient for the STEP-compliant freeform surface machining. However, the official STEP-compliant data models of T-spline are not released. Hence, we proposed STEP-compliant data models for T-spline in our previous work [17]. Thus T-spline can be introduced into the CAx systems using STEP, and STEP-NC based T-spline surface machining has been verified using UV strategy [21].

3 STEP-COMPLIANT CAD/CNC SYSTEM

3.1 Architecture Design

In the last decade, STEP-compliant CAD/CAM systems and STEP-compliant CNC systems are studied individually. However, STEP-NC can transfer partial functionality from CAM to CAD, and transfer others from CAM to CNC. Thus, STEP-compliant CAD/CAM systems and STEP-compliant CNC systems are not proper to describe the whole digital manufacturing chain. In the paper, STEP-compliant CAD/CNC system is proposed that is more reasonable and intuitional to represent the bidirectional data flow through digital manufacturing chain.

STEP-compliant CAD/CNC system has two subsystems: STEP-compliant CAD system (STEP-CAD) and STEP-compliant CNC system (STEP-CNC). STEP-CAD is proposed to design the model and generate STEP-NC files that include machining information such as machining features, working step, tool definition, etc. STEP-CNC consists of MMI, NCK, and PLC that can read STEP-NC file directly. MMI is the major difference between conventional CNC system and STEP-compliant CNC system that should have the capability of online toolpath generation, and simulator to validate the toolpath. The architecture of STEP-compliant CAD/CNC system is shown in Fig 1.

The STEP-CAD has four main function blocks that serve different purpose: 3D model design, manufacturing feature definition, macro process planning and micro process planning.

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- 3D model design: The block is used for shape modeling, and feature-oriented design should be introduced to make it suitable for STEP-NC definition. STEP-NC uses AP203 as the representation of geometry that is widely used for CAD data exchange between different CAx systems, and many commercial software support AP203 such as Solidworks, UG, Rhino, etc.
- Manufacturing feature definition: The block is in charge of feature definition for manufacturing. Manufacturing feature is the basic element for STEP-NC machining that contains feature related parameters. Manufacturing feature can be defined manually or automatically.
- Macro process planning: The block provides a STEP-NC mechanism to plan high-level manufacturing process, such as work plan, working step, sequence, etc. In addition, it provides some functions to regulate the interrelationships, for example optimizing the execution order of working steps.
- Micro process planning: The block defines the rest necessary data models involved in the process planning include operations and resources. Operation indicates how a workpiece is made, such as strategy, technology, toolpath, etc. Resource provides the machining related information, such as cutting tool, probing tool, cooling, etc.

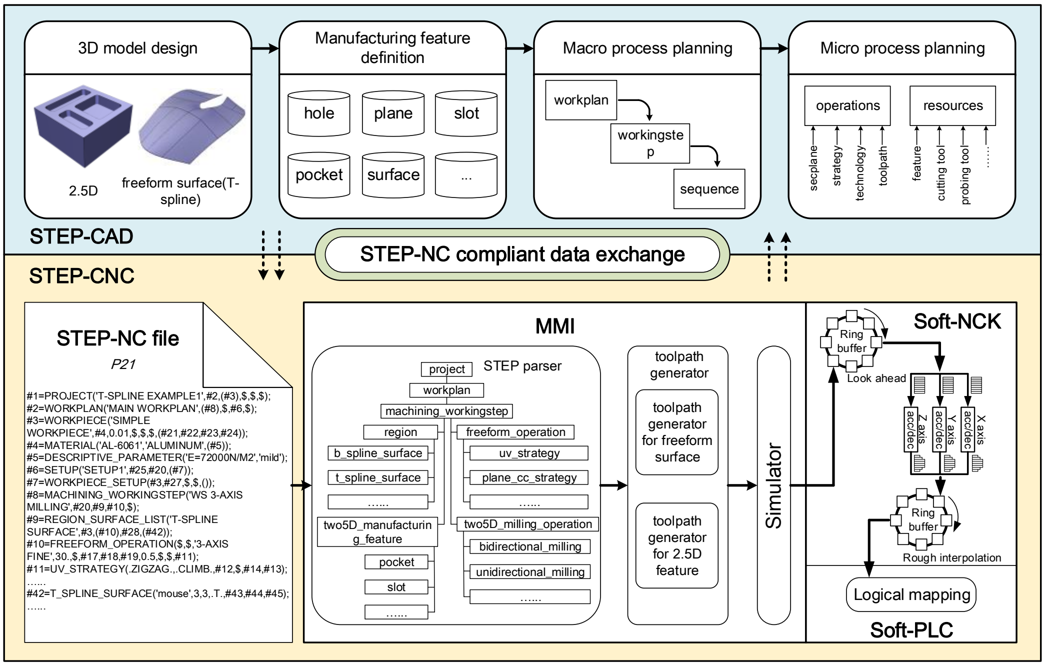

Figure 1: A STEP-compliant CAD/CNC system.

The STEP-CNC has three key components: MMI, NCK, and PLC.

- MMI (Man Machine Interface): MMI is a major difference between conventional and STEP-compliant CNC systems that has powerful ability to process STEP-NC data. MMI can read the STEP-NC file directly, and parser the file into memory under the mechanism of STEP-NC. Afterwards, toolpath is generated online according to the parsed information that need to be verified and simulated before they are sent to the NCK.
- NCK (Numerical Control Kernel): NCK is the unit for controlling the servo, which consists of a look ahead buffer to reduce the real-time requirement on MMI, an accelerator/decelerator

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to generate the gently changed TCP speeds, and a rough interpolator to smooth the axis motions.

- PLC (Programmable Logical Control): PLC is used to perform control functions, and provides the logical mapping functionality to the peripheral equipment, such as servo axes, cooling system, lubrication system, etc.

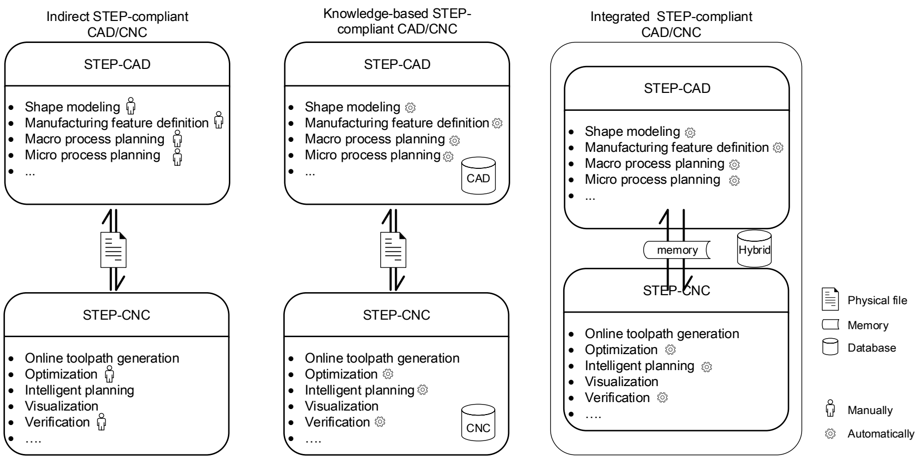

Figure 2: Three types of STEP-CAD/CNC system.

According to the complexity of implementation, three types of STEP-compliant CAD/CNC systems are proposed: Indirect STEP-compliant CAD/CNC, Knowledge-based STEP-compliant CAD/CNC, and Integrated STEP-compliant CAD/CNC. (As shown in Fig 2)

- The first type is Indirect STEP-compliant CAD/CNC. The functionalities of STEP-CAD and STEP-CNC are implemented manually, and information is exchanged through the physical STEP-NC file. This type can be realized based on the existing CAD and CNC systems using secondary development technology. In the paper, we focus on the first type.
- The second type is Knowledge-based STEP-compliant CAD/CNC. Knowledge database is introduced to realize the functionalities of STEP-CAD and STEP-CNC separately. The database could be the local database or the cloud-based database. In addition to this, information is also exchanged through the physical STEP-NC file.
- The third type is Integrated STEP-compliant CAD/CNC. The traditional CAD and CNC systems are need reinvented based on the upgraded hardware. Hence, the information can be transferred through memory, and hybrid database can be used both for the STEP-CAD and STEP-CNC. This type is the ultimate goal to achieve.

3.2 STEP-CAD system

STEP-CAD is a new type of CAD system that can append manufacturing information when designing models, and the standard STEP-NC file can be generated automatically. In the paper, STEP-CAD system is developed based on SolidWorks/UG secondary development technology for 2.5D features. Product is designed by combining STEP-NC defined manufacturing features. Meanwhile, user interface is provided to set up the process and machining tool parameters.

Three kinds of information are defined in STEP-CAD system:

- Geometric information that is used to describe the volume to be removed.
- Processing information such as security plane, feed rate, spindle speed, etc.
- Machining tool information consists of tool diameter, tool length, etc.

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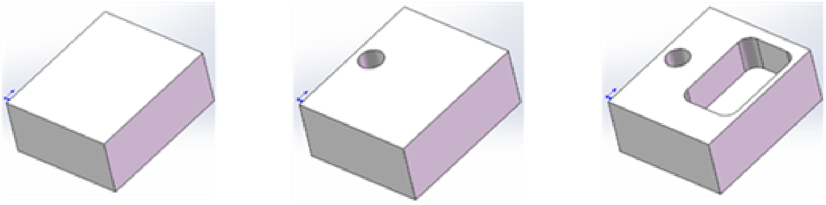
The first step of STEP-CAD system is to construct feature based on geometric information that means the user can generate the manufacturing feature directly through one interaction step. The modeling process of the example product defined in STEP-NC standard is shown in Fig 3.



a.planar_face b. round_hole c. closed_pocket

Figure 3: Modeling process of the product defined in STEP-NC standard.

Each manufacture feature is associated with one or more machining operations in STEP-NC file. In the STEP-CAD system, machining operation is defined through user interface. As shown in Fig 4, user can select a valid machining operation for a closed pocket feature from a list box, and the relevant processing parameters are displayed that can be adjusted if necessary.

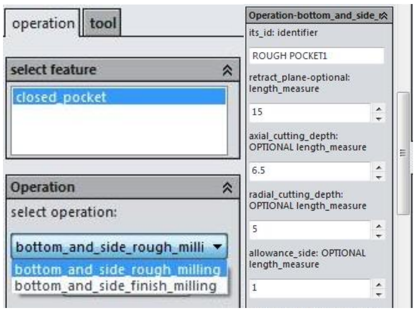


Figure 4: Processing information determination for different features.

In the proposed STEP-CAD system, machining tool information is acquired from the constructed database that is very important for machining. A user interface is provided for the user to verify and select the appropriate tool as shown in Fig 5.

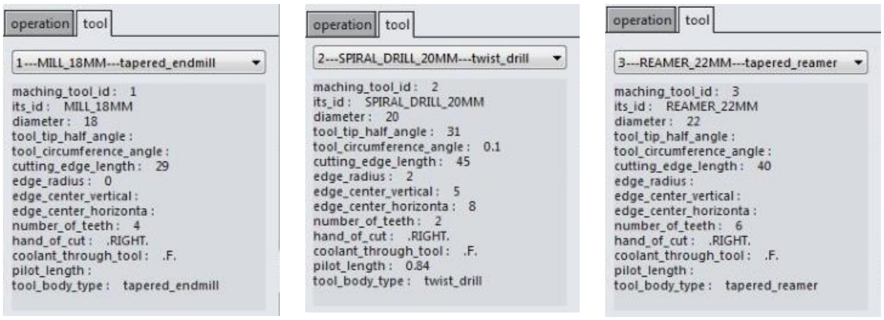


Figure 5: Machining tool user interface.

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There is little commercial software using T-spline for the freeform surface modeling, hence, a variant STEP-CAD system is proposed for freeform surface.

Rhino uses T-spline as plug-in for freeform surface modeling, and a text-based TSM (T-spline Mesh) format is released for the storage of T-spline surface. The variant STEP-CAD system can read and parse the TSM file with help of T-Spline [15] open source library, then a graphic user interface is provided for user to specify the manufacturing information, such as strategy, tool parameters, etc. The interface is shown in Fig 6.

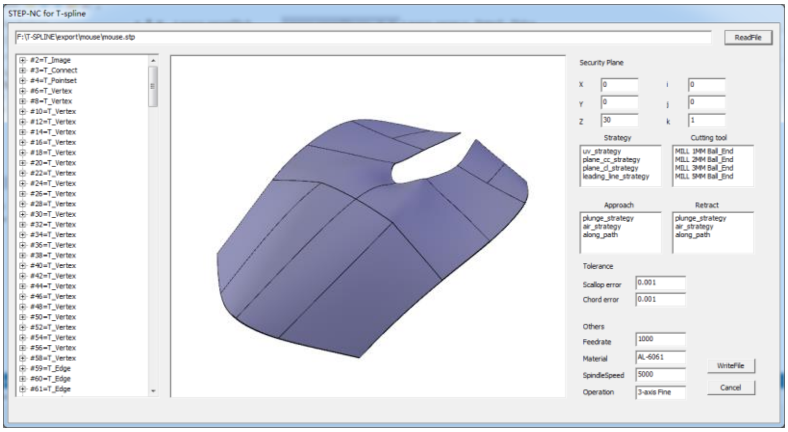

Figure 6: Variant STEP-CAD system for T-spline surface.

STEP-NC file can be generated and exported from STEP-CAD system after process planning. The exported STEP-NC file can be formatted either in Part 21 (clear text) or Part 28 (XML). In the paper, Part 21 format is used for the storage of STEP-NC data. Each Part 21 file has a header section and a data section. The header section contains general information and comments, such as filename, author name, data, organization, etc. The data section is the main section of the STEP-NC file, containing all the manufacturing information, such as geometric, work plan, working step, technology, tool, etc. A typical STEP-NC file generated and exported by our STEP-CAD system is shown in Fig 7.

STEP-NC file is a neutral file that is independent of any CAx systems. STEP-NC file contains the rich information, such as work plan, working step, manufacturing feature, tool, etc. These information are represented using object-oriented language that are more understandable both for human and machine. With the help of STEP-NC files, any information can be exchanged between STEP-CAD system and STEP-CNC system.

3.3 STEP-CNC system

The proposed STEP-CNC system is developed based on the open-architecture CNC platform TwinCAT that consists of three key modules: MMI, NCK, and PLC. Comparing to traditional CNC system, MMI is the major difference that should have ungraded functionalities as bellow:

- Read and parse STEP-NC file directly

  A STEP-parser is developed and integrated into the STEP-CNC system based on the integrated_cnc_schema that is released by ISO TC184/SC4/WG3. In order to introduce T-spline into the STEP-CNC system to enhance the capability for freeform surface machining, we define entities for T-spline and extend the schema. The entities are defined based on the STEP-

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compliant data models [17]. The STEP-parser can read the Part 21 files directly, and parse the files into different instances that provide redundant information for MMI.

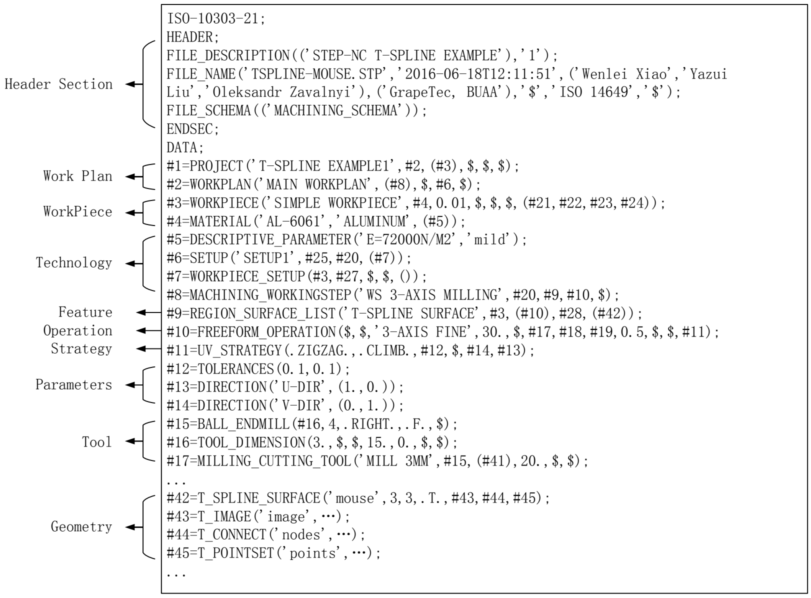

Figure 7: A typical STEP-NC file.

- Online toolpath generation

Toolpath is generated offline by CAM software in the last decade. However, STEP-NC should have the capability of online toolpath generation with high efficiency. For the 2.5D feature machining, toolpath can be generated using the typical algorithms, three models and the corresponding toolpaths are shown in Fig 8a.

Due to the drawbacks of NURBS, online toolpath generation for freeform surface hinders the development of STEP-CNC. T-spline owns the advantages of less control points and watertight representation that is suitable for the online toolpath generation. STEP-NC provides four types of strategies: uv_strategy, plane_cc_strategy, plane_cl_strategy, and leading_line_strategy. In our previous research work [21], STEP-compliant toolpath generation capability for T-spline is demonstrated using uv_strategy for simplicity. In the paper, all strategies are implemented for T-spline surface using the open source library **T-SPLINE** [15]. The different strategies could be the fundamental for the intelligent STEP-CNC system as shown in Fig 8b.

- Toolpath simulation and verification

Simulation and verification of toolpath are critical for CNC machining. Hence, a 3D viewer is developed that provides an intuitive way to check the data for users, and various geometric data can be rendered such as workpiece, toolpath, cutting tool, security plane, etc. The toolpath can be simulated using the data that from calculation visually, or the position actually. The machining feasibility can be verified, and the unpredictable situation can be avoided.

- Resource acquisition

Adaptive online toolpath generation is important for STEP-CNC system to satisfy different machine tools. Hence, resource acquisition from shop floor is critical for STEP-CNC system. With the help of EtherCAT fieldbus, manufacturing resource can be accessed conveniently and efficiently, such as tool description, spindle information, etc. All the resources can be transferred into STEP-CNC systems for online toolpath generation adaptively.

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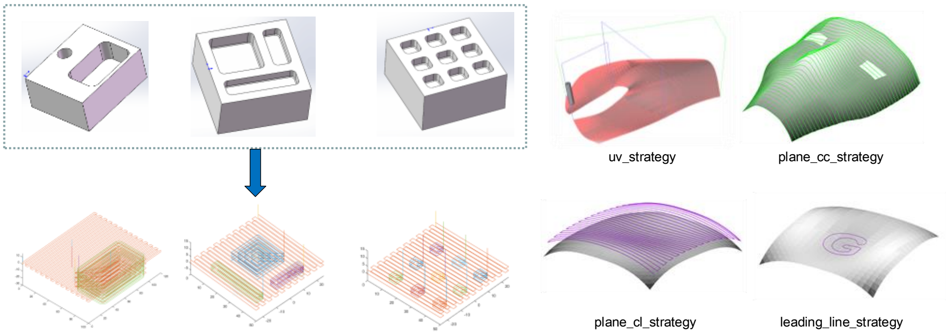

Figure 8: Toolpath generation capability: (a) Toolpath generation for 2.5D features, (b) Toolpath generation for freeform surface (T-spline) using different strategies.

4 EXPERIMENT

A practical 3-axis machine tool is constructed to verify the proposed STEP-compliant CAD/CNC system. The machine tool consists of three axes for the motion of X, Y, Z respectively, lubrication system, cooling system, controller, and other additional parts like IO, power. The architecture of the practical machine tool is shown in Fig 9. All the parts are connected via EtherCAT fieldbus that is a fast industrial Ethernet technology can support a synchronized cycle time up to 100 μs. There are four models have been tested to demonstrate the possibility of the proposed system. The diagram of the STEP-compliant CAD/CNC system and the testing models are shown in Fig 10.

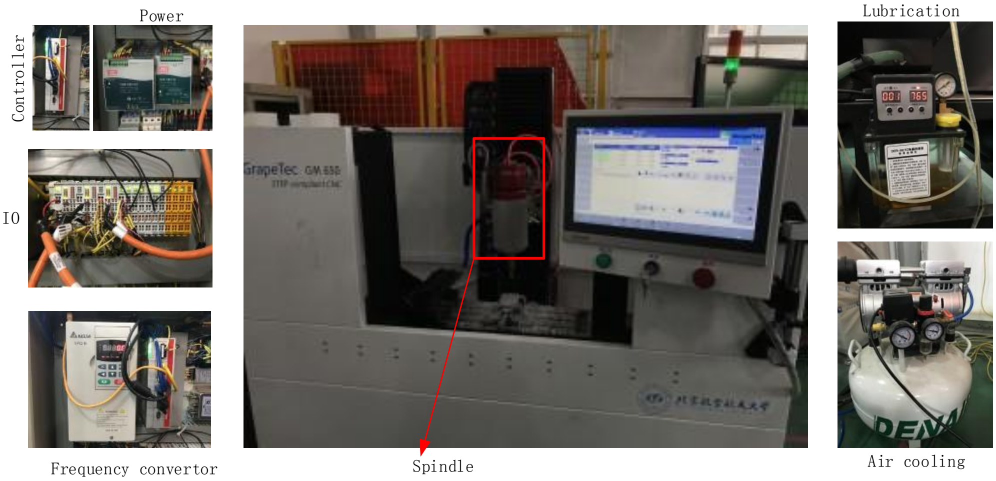

Figure 9: A practical 3-axis machine tool based on the STEP-compliant CAD/CNC system.

5 DISCUSSION AND CONCLUSION

In the field of traditional CAD-CAM-CNC digital chain, different CAx systems work independently. Data transfer between different CAx systems costs plenty of time. STEP-NC is considered as the next generation of standard for data transfer between CAD/CAM and CNC. The STEP-NC defines not only the toolpath data model but also other related process and resource data models. STEP-compliant CAD/CAM and STEP-compliant CNC systems are investigated separately in the last decade. STEP-compliant CAD/CAM system has capability of online toolpath generation, and transfer toolpath into STEP-compliant CNC system for machining. Data transfer is unidirectional

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only from STEP-CAD/CAM to STEP-CNC, and toolpath generated commonly by STEP-CAD/CAM is not suitable for different machine tools. Combination of these two systems cannot address the problem. Hence, STEP-compliant CAD/CNC system is proposed, which has the capability of bidirectional data transfer, and adaptive online toolpath can be generated between different machine tools. However, it is difficult to develop the complete STEP-CAD/CNC system. Therefore, three types of STEP-CAD/CNC systems are proposed, and we focus on the first type for simplify. The second and third type will be researched in the future.

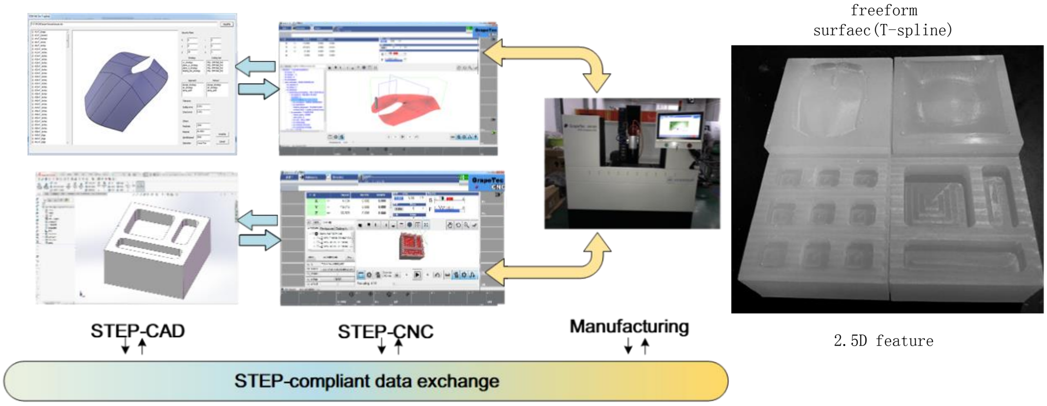

Figure 10: STEP-compliant CAD/CNC system and testing models.

In this paper, STEP-compliant CAD/CNC system is developed for the feature-oriented machining that includes 2.5D features and freeform surface. The proposed system consists of two subsystems: STEP-CAD and STEP-CNC. STEP-CAD is used to design models and generate proper STEP-NC file. A secondary development of Solidworks/UG based on STEP-NC is realized that can build the connection between CAD and CNC for 2.5D features. And variant STEP-CAD is presented to create STEP-NC file for freeform surface. STEP-CNC has the capability of online toolpath generation. Toolpath generation algorithms for 2.5D features are supported. Beyond that, four kinds of toolpath generation algorithms are realized based on the STEP-NC strategy for the T-spline surface machining. A practical machine tool is constructed under the STEP-compliant CAD/CNC system. Some workpieces are tested to verify the capability of system for the feature-oriented surface machining. In the future researches, intelligent manufacturing based on STEP-compliant CAD/CNC system will be studied, such as machining optimization for complex surface, cloud-based manufacturing, intelligent senor-based control, etc.

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