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Research Paper

Integrated Interface Development Environment using STEP Universal Data Structure

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Abstract

Nowadays modern manufacturing demands advanced computer controller, having higher input language and less proprietary vendor dependencies. STEP stands for Standard for the Exchange of Product model data is the next generation of data model between CAD/CAM and CNC system. STEP is still under research and development all around the world. This paper describes the design, development and testing of an integrated Interface development environment for STEP file using Universal Data Structure, which aims to provide support for machining operation. The system also aims to provide function of reading and extracting the relevant information associated with the machining data and to write the G-Code file. The sample of machined block is designed from 3D CAD modeler which consisted of features need to be machined from a blank workpiece and saved in the STEP file format. The validation process will be done using the simulation in the Mach3 software.

Keywords: *STEP; STEP-NC; CNC; G-code; Graphic User Interface.*

1. Introduction

Computer numerical control (CNC) is the main brain of the machine tools in the manufacturing industry which it uses the low level G-code language in order to control the machine. The G-code language is design in the era when paper tape is the medium for moving data between computers and CNC systems. Nowadays the G-code programs are generated using the computer aided manufacturing (CAM) tools by using the geometrical design data from computer aided design (CAD) tools as the input. However different vendors apply different version of G-code which leads to proprietary nature of CAD-CAM-CNC chains. In order for the CAM to generate the G-code for different machine, CAM need to know everything regarding the machine such as the model, detail description of the machine tool and their peripherals such as cutting tools and others components. These components will be handled by a unit within CAM tool called “post-processor” which operates about the G-code, machine tools and libraries of cutting tools. For CNCs, it can only accept programs within their proprietary versions of G-code. Despite that current CNC framework have become more advanced, incompatibility between their proprietary data format restricts further enhancement of CNC-based machining which can also lead to data redundancy and possible errors [1]. Other than that, the shortcoming for the conventional NC programming are limited NC-programming interfaces, no link to CAD data and technology, missing information in shop-floor level, insufficient for controls as well as machine tools and the need of post-processor [2]
In order to overcome the proprietary data restriction among the CNCs machine, the STEP or known as ISO 10303 has been created which provides a new data exchange interface between CAD/CAM and CNC system. STEP-NC has been considered a new programming language for the CNC machine. Part 21 data file is an interpretation from the solid model that contains relevant information and represent the information in an internal format that enables easy modification and viewing from the part 21 file itself [3]. STEP file is an international standard which is consists of design features for the CAD model which is provides general resources for the exchange of data model in a standard CAD form between CAD system or software [4]. STEP is a universal data structure which only in a text format and can be open, edited and saves in any open source software. By using the STEP data structure, the post processor will not be use anymore because the process will bypass the post processor. It is because the information in the data structure file is enough to perform machining without the need of post processor. The most breakthroughs finding in the CNC field is that this STEP file which bypass the post-processor and save more time for the manufacturing process.

2. STEP Programming Standard

ISO 10303, also known as Standard for the Exchange of Product Model Data (STEP) is a high-level programming language, which is, includes design features and manufacturing information regarding the product. This new programming language can standardize information from design to NC controller. It allows bi-directional data flow between CAD/CAM and CNC without any redundancy and loss of information [5]. In the bi-directional data flow, the exchange data should be defined using the object-oriented modeling method and no data leakages are allowed on CAD-CAM-CNC data chain (Fig. 1). A new generation of intelligent controllers can interpret STEP file information to generate, simulate and optimize machining tool-paths [6]. STEP file does not describe the tool movements for specific CNC machine tool as G code does but provides a feature based data model. A STEP-NC data file is a universal data structure which means it is an open source data structure and not proprietary to any specific machine or brand and can be used on various open source machine tool controllers in the market. It means that just by using STEP data file only, it can run on any open source machine controller without the need of post-processor for that machine.

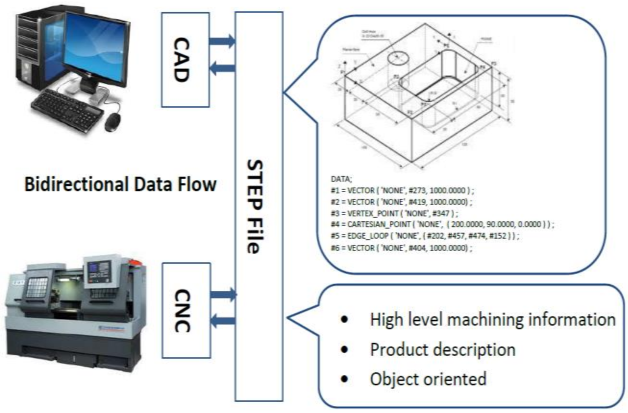


Fig. 1: STEP high level program

2.1. STEP Data Structure

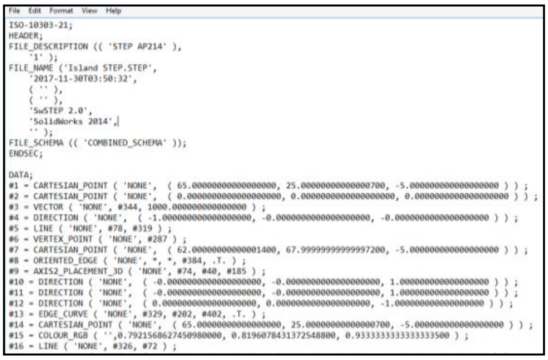
To enable bi-directional interoperability, STEP has its own ISO standard which being used and understood by all which is ISO 10303. File extension for STEP file is STEP or STP file is an open source file and can be opened using any standard software such as Microsoft Word, Notepad, etc. STEP data structure consists of HEADER and DATA section. General information is described in HEADER section. This includes file title, author and schema information. In DATA section, the instances consist of design feature of the solid model which are the Cartesian coordinate, Cartesian point, lines, axis, units, etc. By having all information in the same file all changes are effective thus eliminating the need of post-processor. This is why STEP standards appear as the relevant solution to extend the interoperability of CAx chain to the CNC level.

2.2. STEP File and Solid Model

STEP file consists only design features of the solid modelling (Fig. 2). In addition, it only has Cartesian coordinate and other basic features of the solid model. In this paper, the project is only using Part 21 STEP file instead of total STEP NC file due to exploiting the Cartesian coordinates in order to generate the G-code. Solid model has been created using the Solid work software in order to use it as the main product feature in this paper (Fig 3). A simple solid model has been designed and properly given dimension in order to get the fully defined solid design. Then the solid model is saved in a STEP file using the same software which is the solid work. The solid model which is in SLDPRT file will be opened in CAM in order to simulate and generate the G-code. The x-y-z coordinate from the auto-generated G-code will be compared with the Cartesian coordinate in the STEP file which has been saved earlier.

2.2. Extract the Cartesian Coordinate

The STEP file consists of design features for that particular solid model. It consists of Cartesian coordinate in that solid model for every angle and edge. By checking the entire coordinate and compare with the design and STEP file in order to make sure it is correct and accurate. It will label all the Cartesian coordinate especially the one that is involved with the machining. The labelling process is very important in order to generate the G-code. As we all know that G-code is created based on coordinate for each corresponding axis [7]. All the coordinate will be combined together in order to create the tool path for the machining purpose. The coordinate actually the movement of the tool and thus generate the tool path based on the required geometry. In this step, only the Cartesian coordinate in the STEP file data will be selected and labelled according to the corresponding solid model. After the extraction, the Cartesian coordinate will be rewritten again in the different file which consists only the Cartesian coordinate. Then based on the Cartesian coordinate from the STEP file, the G-code is generated (Fig. 4). The Cartesian coordinate that is taken only from the affected machined surface which is the surface involved with the milling process.



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File  Edit  Format  View  Help
ISO-10303-21;
HEADER;
FILE_DESCRIPTION (( 'STEP AP214' ),
    '1' );
FILE_NAME ('Island STEP.STEP',
    '2017-11-30T03:50:32',
    ( '' ),
    ( '' ),
    'SwSTEP 2.0',
    'SolidWorks 2014',
    '' );
FILE_SCHEMA (( 'COMBINED_SCHEMA' ));
ENDSEC;

DATA;
#1 = CARTESIAN_POINT ( 'NONE',  ( 65.00000000000000000, 25.00000000000000700, -5.000000000000000000 ) ) ;
#2 = CARTESIAN_POINT ( 'NONE',  ( 0.0000000000000000000, 0.0000000000000000000, 0.0000000000000000000 ) ) ;
#3 = VECTOR ( 'NONE', #344, 1000.000000000000000 ) ;
#4 = DIRECTION ( 'NONE',  ( -1.000000000000000000, -0.0000000000000000000, -0.0000000000000000000 ) ) ;
#5 = LINE ( 'NONE', #78, #319 ) ;
#6 = VERTEX_POINT ( 'NONE', #287 ) ;
#7 = CARTESIAN_POINT ( 'NONE',  ( 62.00000000000001400, 67.99999999999997200, -5.000000000000000000 ) ) ;
#8 = ORIENTED_EDGE ( 'NONE', *, *, #384, .T. ) ;
#9 = AXIS2_PLACEMENT_3D ( 'NONE', #74, #40, #185 ) ;
#10 = DIRECTION ( 'NONE',  ( -0.0000000000000000000, -0.0000000000000000000, 1.000000000000000000 ) ) ;
#11 = DIRECTION ( 'NONE',  ( -0.0000000000000000000, -0.0000000000000000000, 1.000000000000000000 ) ) ;
#12 = DIRECTION ( 'NONE',  ( 0.0000000000000000000, 0.0000000000000000000, -1.000000000000000000 ) ) ;
#13 = EDGE_CURVE ( 'NONE', #329, #202, #402, .T. ) ;
#14 = CARTESIAN_POINT ( 'NONE',  ( 65.00000000000000000, 25.00000000000000700, -5.000000000000000000 ) ) ;
#15 = COLOUR_RGB ( '',0.7921568627450980000, 0.8196078431372548800, 0.9333333333333333500 ) ;
#16 = LINE ( 'NONE', #326, #72 ) ;
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Fig. 2. STEP File

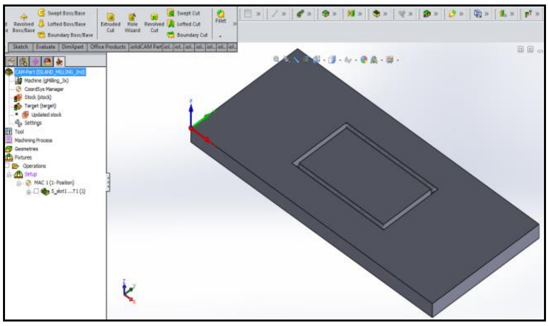
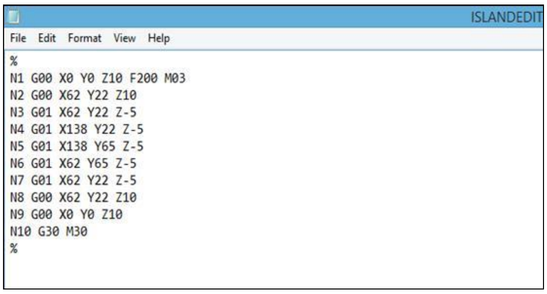


Fig. 3. Solid Model



```
ISLANDEDIT
File  Edit  Format  View  Help
%
N1 G00 X0 Y0 Z10 F200 M03
N2 G00 X62 Y22 Z10
N3 G01 X62 Y22 Z-5
N4 G01 X138 Y22 Z-5
N5 G01 X138 Y65 Z-5
N6 G01 X62 Y65 Z-5
N7 G01 X62 Y22 Z-5
N8 G00 X62 Y22 Z10
N9 G00 X0 Y0 Z10
N10 G30 M30
%
```

Fig. 4. The G-code Generated

3. Mach 3 Simulation

Mach 3 software is an open source CNC controller which controls the open source CNC machine without having dependencies on the brand and model of the CNC machine itself (Fig. 5). After the generation of the G-code from the Cartesian coordinate, the G-code must be tested in order to know it can be used or not. The validation process being done using the open source software which is the Mach 3 based on the required setting parameters (Fig. 6). By loading the G-code into the Mach 3 software, it can know whether the G-code can be used or not based on the simulation generate in the software itself. The G-code should be able to generate the toolpath based on the required CAD design solid model in order to create the final solid model using the milling operation. The toolpath generated from the movement of the tool is based on the G-code created from the Cartesian coordinate which is in the STEP file itself. This machining process involving the slot milling operation using the end mill tool with the rectangle island shapes. The milling shape is just a simple island shape with the rectangle shape with depth of 5 mm which means the z level will be -5.

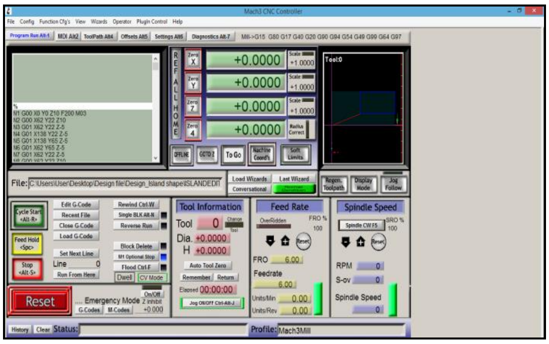


Fig. 5. Mach 3 Interface

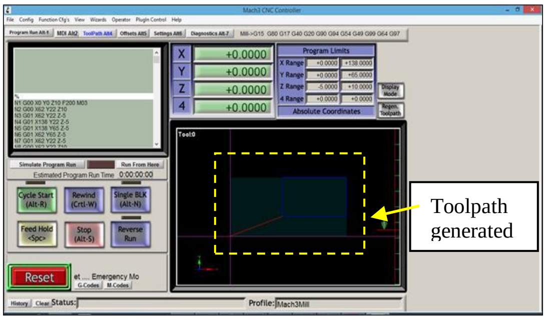


Fig. 6. Mach 3 simulation

4. Discussion

The generated G-code is based on the STEP file with the reference of the Cartesian coordinate in the file. The Cartesian coordinate is used as the reference axis in order to generate the toolpath using the G-code. Now it can successfully perform the machining process without having the post-processor. This process already successfully bypassing the post-processor and successfully generate the G-code based on the information in the STEP file. By using the Cartesian coordinate in the STEP file, it can generate the G-code based on the required solid CAD design. It is because the G-code only consists of coordinate and certain machining parameters. This process is a read-write-read-write process for the CNC machining. First of all, the read stage is when the solid model being saved as the STEP file format, the software read all the solid features and save it as the required file. Then the write process is when the information within the solid model is being summarized as a text file format which contains all the design features for that particular model. Later the read process again is when the extraction of the Cartesian coordinate in the STEP file and finally the last write process is when the generation of the G-code based on the Cartesian coordinate. This extraction of the Cartesian coordinate being done manually based on the STEP file saved earlier.

For the future recommendations, the Cartesian coordinate selection can be done by using automatic selection on a generated platform interface. The extraction can be done automatically using the generated interface which is programmed to automatically generate the G-code. read-write-read-write (RW/RW) the STEP file.

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