A method of verifying reduction in machining time for three axis CNC lathe mill

Atul Khosla*

School of Mechanical Engineering, Lovely Professional University, Phagwara, Punjab 144411, India

ABSTRACT

Intricate features in manufactured products are being incorporated to attract customers and to provide component of any shape, there is an urgent requirement of not only machining but also finishing scallops using identification of complex and complicated shapes so as to ultimately reduce the machining time. In the present work, a primary input is given in the form of STL file format depicting part geometry and the output data in the form of cutter location (CL) points is obtained through the machining algorithm considering various machining parameters and important checks. The main objective of the present work is to save machining time.

Keywords: CNC machine, STL file, lathe mill, tool path generation algorithm, feature identification.

1. INTRODUCTION

With the inception of different CAD/CAM softwares, machining of intricate components can be done on Computer Numeric Controlled (CNC) machines with highest accuracy in terms of dimensions. These softwares help in simplifying CNC programming. The complete CNC machining process is divided into three stages:

- Design phase
- Tool Path planning stage
- Verification and manufacturing stage

3D Computer Aided Design (CAD) model is prepared using any CAD software such as CREO, CATIA etc. in the first stage.

In the second stage, the cutter path is planned to produce cutter location data that can be translated into G-Codes which is fed to the CNC machines.

In the last stage, firstly generated tool path is verified and checked in a simulator for machining errors or flaws and then the actual component is loaded on the machine wherein the machine tool works as per the G & M codes generated to provide final product.

1.1 TOOL PATH PLANNING AND MACHINING APPROACHES

Tool path planning is the motion planning of machine tool and provides the exact trajectory followed by machine tool. Tool path gives information about surface topology because of scallops developed during machining.

Basically intricate surfaces are machined in three sub-stages: Rough machining, finished machining and manual finishing using ball nose end milling cutter on 3-axis milling machines. CNC milling machines direct the tool via a computerized controller.

Gray et al. [1] proposed a technique in which the machine operates from simple PC. The work done by Bedi et al. [2] is an elaboration of wherein the information of CAD model is taken as STL (Stereo-lithography) file and tool path is directly generated from STL file, thus eliminating the need of postprocessor, in which is used to generate the G & M codes. H. T. Yau et al. [3] proposed a model on Numerical control machining of triangulated sculptured surfaces in a stereo lithography format with a generalized cutter.

1.2 PROBLEM IDENTIFICATION

A tool path generation algorithm had been already developed for the three axis CNC milling lathe machine with ball nose end mill cutter [6]. Here tool path was generated which moves in the helical path with uniform side feed and varying radius of cutter for roughing and finishing passes. By taking the STL (Stereo-lithography) file as initial input from the CAD software, the computer program generates the cutter location (CL) points in a text file. Tool path generated is capable of generating all artistic features and complex parts other than making the simple turning parts. After this generalized tool path algorithm, it was identified that there was a need of feature based tool path planning algorithm which would identify the features like cavities, pockets, grooves etc. based on the x values generated in the text files obtained from the previous algorithm. Hence, a generalized algorithm for the cylindrical and conical geometries having distinct features such as cavities, pockets, grooves has to be developed so as to save machining time.

1.3 PARAMETERS REQUIRED FOR GENERATING TOOL PATH

a) STL data for CAD model: The STL file [4] [5] is the first input to the tool path generation algorithm. The STL standard includes ASCII data format.

b) Radius of roughing and finishing tool: The radius of roughing and finishing tool is the next input given by user

c) Depth of cut for roughing: The depth of cut for roughing pass can be given by user as per requirements.

d) Finishing allowance: The another value which user inputs in the program is the finishing allowance which the roughing tool should leave for finishing tool. Generally it is kept 2mm.

e) Side feed: The value of side feed is to be entered by the user. This value is kept large for roughing pass and small for finished passes.

2. IDENTIFYING GEOMETRY AND FEATURE BASED TOOL PATH PLANNING

Based on the files generated from the generalized tool path algorithm, it can be identified by comparison of the x values that whether the geometry of work piece is cylindrical or conical. The present work is based on the identification of the geometry mainly cylindrical and conical geometry and then to identify what are the various features in these geometries like cavities, pockets, grooves etc. After identification of various features in a particular geometry, tool path planning strategy has to be adopted in such a way that firstly whole geometry is to be machined and features are skipped and saved in other files then feature based tool path planning is done. Reduction in machining time is the other objective of the present work.

2.1 IDENTIFICATION OF FEATURES IN THE CYLINDRICAL AND CONICAL GEOMETRIES

If we have the combined geometry for both cone and cylinder then the tool path algorithm must verify whether first geometry is a cylinder or a cone then recognize the features in it and skip them and then go to other geometry and again recognize the features in it and skip them also and then in the last it would machine all the features separately in a single go

3. METHODOLOGY

In the present work, Ball drop method [4] was selected for developing machining algorithm. The entire work piece surface which is to be machined is mathematically represented by the triangle known as facets and surface normal called facet normal. In the Dropping Method [7], for each cutter position the tool is dropped from a height along the tool axis at any particular position in X-Y plane. The cutter drops steadily along tool axis plane and would intersect one of the triangles representing the work piece surface.

The point of contact is dependent of the topography of the surface. The tool path planning algorithm has been designed to check for the projection of the tool in a given position that the generic ball end milling cutter is touching with the part of the triangulated surface. The parameter u is set for the extreme position of the tool, and it keeps on decreasing as goes from centre to the outer of the work piece. The tool projection may be inside the triangle or may touch triangle on edge or on vertices. The tool positioning strategy must stops the tool forward motion at the first contact of the tool with either of edge, triangle surface or vertex for which triangle, edge and vertex checks have been applied to extract the information about tool position along X-axis direction of the 3-axis CNC milling lathe.

After applying all these checks, the value of point of contact (P_c) is checked whether it is in the triangle or not and the appropriate value is stored and then tool is moved to another location. These values of P_c are stored in a file to retrieve for the next step.

Then tool path is generated for both roughing and finishing operation.

In the last step, machining time is noted and compared with the time required for machining while machining whole component and not neglecting features.

3.1 WORK VALIDATION

Tool path is tested on a Custom Simulator 'Tool Sim'. The validation is done by comparison of the STL file of the three dimensional geometry with the obj file of final machined geometry. Tool path have been prepared for different models like cylinder, cone and then various features are incorporated into these geometries like cavities, pockets and protrusions and these features are then machined in the last and tool path is validated by saving the stock as obj file and then validation of tool path algorithm is done by comparing obj file with STL file

4. CONCLUSION

Tool path algorithm for the 3-axis lathe mill has been developed in the present work. Ball nose end mill cutter tool has been used in the algorithm. In the present work, tool of radius 6.35 mm has been used for roughing and 1.5875mm for finishing operations. Various checks have been applied to trace cutter contact point of contact with work piece. The accuracy of work piece depends upon the STL triangles, radius of tool for finishing and side feed for finishing pass. After generation of roughing files and finishing file, identification of the geometries was done by developing geometry based algorithm and then various features like cavities, pocket, and grooves have been separately machined in the final step.

References

[1] P.J. Gray, S. Bedi, and S. Mann, "A new paradigm for numerically Controlled machines", Virtual Concept, Biarritz, France, 2003, pp. 128-134

[2] Nick P.Manos, Sanjeev Bedi, Dan Miller and Stephen Mann, "Single Controlled Axis Lathe Mill", International Journal of Advanced Manufacturing Technology, vol. 32, pp. 55-65, 2007.

[3] H. T. Yau, C. M. Chuang, and Y C Lee, "Numerical control machining of triangulated sculptured surfaces in a stereo lithography format with a generalized cutter", International Journal of Production Research, vol. 42, no. 13, 2004 pp. 2573–98.

[4] Kandarp patel, "Web Based Automatic Tool Path Planning Strategy for Complex Sculptured Surfaces", 2010.

[5] P. F. Jacobs, and D. T. Reid, "Rapid Prototyping & Manufacturing: Fundamentals of Stereo-lithography", McGraw-Hill, New York, 1992

[6] Paras Mohan Jasra, "Generalized tool path generation algorithm for sculptured pseudo symmetric surface machining", 2009.