



TECHNICAL UNIVERSITY OF CLUJ-NAPOCA

ACTA TECHNICA NAPOCENSIS

Series: Applied Mathematics and Mechanics

Vol. 55, Issue I, 2012

CAM IN THE CNC MACHINING PROCESS

Corina-Mihaela MATEI, Andrei STEOPAN, Ioan BLEBEA

Abstract: The paper discussed the importance of using CAM software's for the toolpath generation when machining complex decorative models using 3-axis CNC milling. Special attention is place upon cam simulation as an important aid in optimization the machining process. The aim of this paper is to describe and reveal the facilities of CAM strategies of ArtCAM Pro software for decorative object production.

Key words: decorative model, ArtCAM Pro, CAM strategies, toolpath simulation, CNC machining

1. INTRODUCTION

In modern manufacturing practice, computerized numerical control (CNC) machine tools are commonly used to produce a variety of objects for decorative purposes. CAD/CAM software tools are essential for generating creative shapes and toolpaths for the desired applications e.g. milling, engraving. Based on their exploitation in art, they are called Artistic CAD/CAM software. Their purpose is to combine the creativity of art and the precision of technology.

When setting up a project for CNC milling it is of crucial importance to relate setting in the digital file to the psychical setup of the CNC machine. Aspect such as the dimension of the material, position and tools properties need to correspond to the setting in CAM stage [1].

The generated toolpaths from a CAD model do not necessarily produce a part which exactly matches the virtual model. To ensure that the toolpaths generated produces an accurate physical model, prior to actual machining, it's important to inspect them through a digital simulation. The simulation makes it easier to visualize the exact toolpath and surface finish, allowing a fast and easy problem identification and change opportunity.

A series of selective guidelines are available within this paper so interested users can use this

information in order to obtain fast and reliable results.

2. PROGRAMING USING ARTCAM PRO SOFTWARE

ArtCAM Pro is a powerful software able to both design models and prepare files for machining. This paper will focus exclusively on the latter capabilities. More specifically, this paper will give guidelines for creating a machine-ready file (for example, a .nc file) from an imported 3D model.

The general workflow logic applied to this job is explained in the diagram represented in figure 2.



Figure 1. Theater Comedy and Tragedy Masks Model

Suggested toolpaths sequence therefore typically follow the template: roughing, semi-finishing and finishing. The idea [2] is to first

use a relatively large roughing tool (End Mill) with the Z Level Roughing strategy to remove as much material as quickly as possible to reveal a stepped, oversize component form. A suitable tool (Ball Nosed) will then be applied with the Machine Relief strategy to track over the component contours as an initial semi-finishing operation. This tool will be too large to pick out any fine detail or internal corners but will quickly remove most of the remaining stock from the component form. A further Machine Relief strategy using a Conical Flat tool will then be applied to machine to finish size and pick out the fine detail.

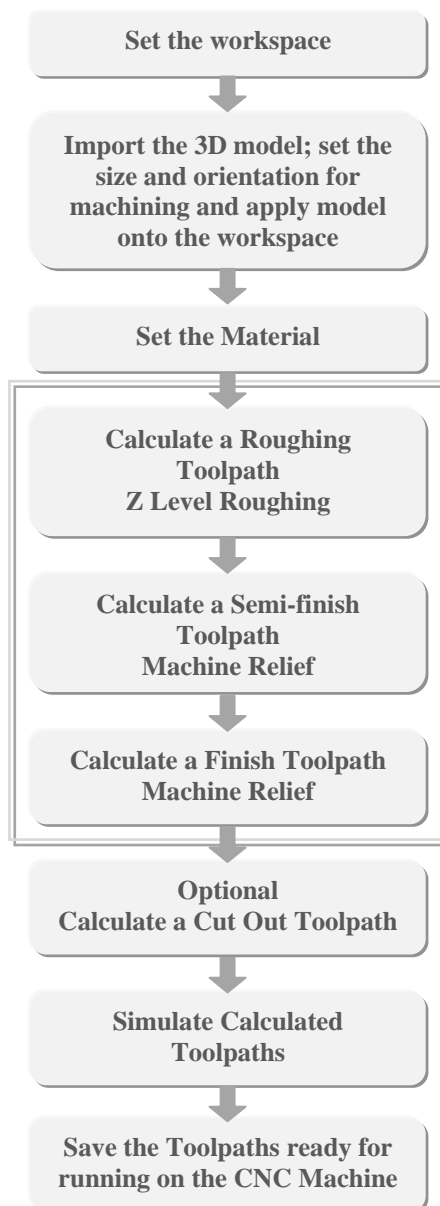


Figure 2. CAM workflow for theatre masks

2.1 Machining strategies

The strategy control the way in which the model or certain areas are machined.

For Z Level Roughing there are two types of machining strategy options: Raster, where the tool performs parallel linear moves or Offset, where the tool follows concentric contours. The machining is done in slices. In this case, the slices are machined using the raster strategy without additional profile pass.

The Machine Relief strategy covers a variety of semi-finish and finishing options. If required it is possible to restrict the machining area.

For the proposed model only certain areas would benefit from a further semi-finishing and finishing strategy.

Machining under selected areas is very useful to achieve high quality surface finish and efficiency by minimize the machining time. These options give the choice of: raster, offset, 3D offset and 3D offset spiral clearance strategies within these regions.

The selected strategies for the theatre masks model are: for semi-finishing the raster strategy is applied on the boundary area to avoid any remaining material left behind by the roughing tool, and for finishing is used the classic offset strategy, specifically on the model's inside regions, because it takes less calculation and machining time requiring less tool lifts.

Setting up the toolpaths in a correct way is important to ensure there is no risk of damaging the machine or the material, and that time it takes to produce the part is optimized. Determining the right method is about finding the right combination of cutting tool, cutting speed, stepdown and stepover setting, depending on the type of material used, the characteristics of the geometry and finish quality that is appropriate [1].

2.2 Tools and assigned parameters

To successfully generate toolpaths, tooling information is required before calculation.

A comprehensive Tool database can be accessed from within the individual toolpath strategy forms. The tools are set up for a number of different materials and applications. The stored tool definitions can be modified or supplemented as required.

To accomplish the proposed toolpaths

sequences and to achieve the desired result, each operation should be performed with dedicated cutting tool type.

To remove a larger amount of material as quickly as possible an End Mill tool is generally selected for roughing as it has a flat bottom and it can be stepped over by at least 70% of the tool diameter [3]. For semi-finishing a Ball Nosed tool is chosen to remove the material left on from the roughing and for finishing, a Conical Flat tool is used to achieve a smoother, more accurate surface finish.

The motion and speed of the tool is specified through several parameters e.g. stepdown, stepover, spindle speed, feed rate and plunge rate. These parameters are often selected from machining handbooks, experience, tool tables etc. Good results can only be achieved with well-balanced settings of these parameters according to cutting tool for each operation and workpiece material.

In the following, these parameters are further discussed.

Stepover defines the distance the tool will move horizontally when making the next step. This distance is a percentage of the tool diameter and is generally in the 25-40% range [3].

Stepdown defines the distance the tool will move vertically when performing the next Z-pass. This distance must be within the total cutting and is generally 1/3 to 1/2 of the flute length at most or less depending upon the material being cut [3].

The standard approach for calculating spindle speed and feed rates is:

- select from a table the proper cutting speed v_c for the material - tool combination
- calculate the spindle speed n required to achieve v_c for a given tool diameter d

$$n = \frac{1000 \times v_c}{\pi d} \quad (1)$$

- select from a table the proper feed per tooth f_z for the type of cut
- calculate the feedrate f

$$f = f_z \times z \quad (2)$$

where z is the number of teeth on the tool.

Plunge Rate is the rate at which the tool approaches towards the material. It is generally 1/2 to 3/4 of the feed rate [3].

2.3 Toolpaths calculation

After setting all necessary, the toolpaths can be calculated at any time individually, or as a part of a batch. The result is a series of paths traversing the surface of the material, stepping over and stepping down successively to cut away the material [3].

3. TOOLPATHS SIMULATION

Prior to actual machining, it is necessary to perform a digital simulation to verify the generated toolpath/toolpaths allowing to anticipate tool movements and to evaluate finished surface quality. Any deviation from the required geometry can be visualizes and if there is any error in toolpath data then it can be corrected by running simulation again before performing the physical machining of the workpiece in real environment.



Figure 4. Simulation result

4. POSTING

Posting is the final step that translates the generated toolpaths into a correct format file understood by the CNC machine controller.

Depending upon whether the machine has a tool changer, the toolpaths for a series of cutter can be put into one large file, or in separate files for each tool.

5. THEATER MASKS MACHINIG

For machining the theater masks operations files the Fawoo 700 S CNC equipment is used.

The machine setup procedure requires a special attention. The block of material (chemiwood) is positioned on the worktable and then be adjusted manually. The tool is placed on the machine head and moved onto the top front left corner of the material block for establishing the origin point of 0,0,0 coordinates. (Material Z Zero is set to Top in the CAM – the model is in the positive x, y and the negative z area).

When changing the specific .nc file from roughing to finishing, the adequate tool must also be changed. In this case the machine does not have automatic tool changer. Thus the operations are done manually by measuring the length of the new tool and adjust the high Z origin point.

Finally, each operation program is run and the entire machining process is supervised.



Figure 5. Machining result

6. ACKNOWLEDGEMENT

This paper was supported by the project "Doctoral studies in engineering sciences for developing the knowledge based society-SIDOC" contract no. POSDRU/88/1.5/S/60078, project co-funded from European Social Fund through Sectorial Operational Program Human Resources 2007-2013.

7. CONCLUSION

Due to the complex forms that this objects for decorative purposes usually have and the high surface quality that they require, the machining automatic programming are becoming more and more indispensable. The generation of the machining process of such configurations using CAM software's in general and ArtCAM in particular, is based on several basic knowledge which is linked to technological and geometrical information, machining strategies, cutting tools and parameters.

8. REFERENCES

- [1] AA Digital Prototyping Lab, *CNC Machining Manual*, <http://www.digitalfabrication.net>
- [2] ArtCAM Training course, *Relief Machining*, <http://hobbybob.home.xs4all.nl/Artcam/>
- [3] Field, E., *Generating Machining Toolpaths in ArtCAM*, <http://www.arch.virginia.edu/arch549/handout.html>
- [4] Delcam, *ArtCAM Pro software inbuilt help and explanations guide*.

CAM ÎN PROCESUL DE PRELUCRARE CNC

Lucrarea subliniază importanța utilizării softului de tip CAM în generarea traiectoriei sculei pentru prelucrarea suprafețelor complexe decorative prin frezare CNC în 3-axe. O atenție deosebită este acordată simulării traiectoriilor cu scopul optimizării procesului de prelucrare. Scopul acestei lucrări este de a descrie și de a prezenta soluțiile de proiectare tehnologică oferite de softul ArtCAM pentru realizarea unui obiect decorativ.

Corina-Mihaela Matei, PhD, Student Dipl. Eng, Technical University of Cluj-Napoca, Department of Design Engineering and Robotics, e-mail: Corina.Matei@muri.utcluj.ro.com, Office phone: 0264-401664.

Steopan Andrei, PhD, Student Dipl. Eng, Technical University of Cluj-Napoca, Department of Design Engineering and Robotics, e-mail: andrei.steopan@muri.utcluj.ro.com, Office phone: 0264-401664.

Ioan Blebea, Prof. Dr. Dipl. Eng, Technical University of Cluj-Napoca, Department of Design Engineering and Robotics, e-mail: blebea@muri.utcluj.ro.com, Office phone: 0264-401664.