Proceedings in MANUFACTURING SYSTEMS

Proceedings in Manufacturing Systems, Volume 9, Issue 1, 2014, 41–46 ISSN 2067-9238

IMPLEMENTATION OF HIGH CUTTING SPEED TECHNOLOGIES IN MACHINE TOOLS FOR INCREASING PRODUCT QUALITY

Dorel ANANIA[1,*], Andra PENA[2], Cristina MOHORA[2]

[1]Lecturer, PhD, Machines and Manufacturing Systems Department, University Politehnica of Bucharest, Romania
[1]Lecturer, PhD, Machines and Manufacturing Systems Department, University Politehnica of Bucharest, Romania
[1]Prof., PhD, Machines and Manufacturing Systems Department, University Politehnica of Bucharest, Romania

Abstract: *In this paper some researches made for using high speed cutting technologies for machining parts on milling center are presented. The goal of this first study was to compare the roughing machining time for the same part by changing the cutting parameters based on the type of the tool trajectories. For cutting parameters calculation in classic theory, it is not taken into account the tool trajectories over the part surfaces. Another goal was to check the machine behavior for specific machining cutting conditions with high depth of cut and high cutting speed using an end mill tool for roughing procedure. The comparison was made between classic roughing with parallel trajectories and an innovative tool path programs implemented in Cimatron CAM software. The technique used is useful for machining small parts with cavities where high feed cutting tools cannot be used.*

Key words: *High speed machining, machine tool, VoluMill, metal cutting.*

1. INTRODUCTION

Today requirements in industry are more and higher form point of view of product quality, delivery time and cost. Companies try to achieve these goals by using *efficient* computer aided design and computer aided manufacturing (CAD-CAM) software, machine tools and cutting tools. In this way the productivity is increasing up to 40–50% comparative to the traditional methods. But this has an inconvenient generated by the high prices of these components and also by the "broken link" between them [6].

Cutting tool and machine tool technologies have become more sophisticated over the years, but some experts believe full productivity gains have not been made because of the inherent shortcomings in the off-line generation of tool paths.

In CAD-CAM you can do almost everything on the computer but you are limited by the technological conditions. Also the tools are now more and more efficient and can support high load at high feed. High-performance geometries are designed for optimum speeds and feeds and ensure a customer a heightened return on their cutting tool investment [1].

The performances of the machine tools also are increasing. The stiffness and the accuracy of the machine nearby the dynamical behavior define the quality of the parts [2]. No matter how sophisticated a machine tool is, it cannot do anything without a tool and a tool path, (Glenn Coleman).

* Corresponding author: Splaiul Independentei 313, district 6, 060042, Bucharest, Romania,
Tel.: 0040 21 402 9724;
E-mail addresses: *dorel.anania@upb.ro*, (E. Anania), *andra_lita@yahoo.com* (A. Pena), *cristinamohora@yahoo.com* (C. Mohora)

But if all this factors (CAM software, machine tools, and cutting tools) are not linked it will be high costs and small productivity increase.

Today there are researches in this direction.

By using CAM techniques one more parameter (beside classic cutting parameters) can be optimized in order to obtained productivity (high rate chip removal, low load on the tool and machine) and this is tool path [9].

The tool path has always been the weak link in the chain –machine tool; cutting tool; CAM software, typically forcing the use of cutting parameters that are on the side of caution rather than productivity. These cautionary tool parameters are detrimental to the tools performance and fate.

In numerical controls programmers tried to replicate the human skills in manual machining.

Every time when a sharp corner appears in the tool path, the load of the cutting tool increases because the engage of the tool in material is higher (Fig. 1).

The traditional tool path is limited by four technological features [1]:

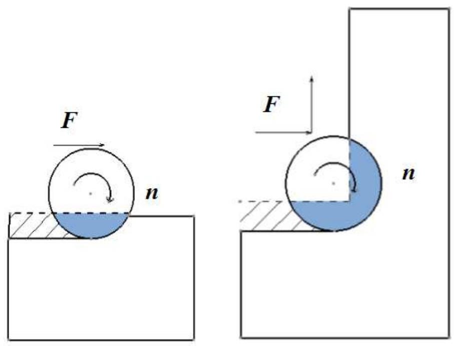


Fig.1. Tool load increasing in sharp corner:
a – normal tool load; *b* – sharp corner tool load.

- initial full cut into the material;
- stepping over between cuts;
- feeding into new areas of the part;
- overloading in corner.

Classic algorithm for milling selected portions of a workpiece by a cutting tool of a numerical control machine fallows the next step:

- a main tool path is generated, the main toolpath comprising one or more cuts;
- compute one or more transition toolpaths, each transition toolpath defining a transition area intersecting one or more of the cuts of the main toolpath;
- trim one or more cuts of the main toolpath near two or more points of intersection between the main toolpath and the transition toolpaths;
- connect the trimmed portions of the main toolpath with one or more connecting moves such that each connecting move is mostly inside one or more of the transition areas;
- mill the selected portions of the workpiece by moving the cutting tool in accordance with the cuts, transition toolpaths, trimmed cuts, and connecting moves.

There have been many attempts over the years to improve upon the traditional toolpath. Parts of these attempts are based on the pick geometry and offset it according with tool diameter. There are some algorithms for detect and improve approaches in order to help a path that has flaws [3].

2. NEW APPROCHE IN TOOLPATH GENERATION – VOLUMILL

New researches are made in order to generate the tool path so that the defects are never there in the first place.

A high-quality cutting tool, driven by innovative tool path programs can increase metal removal rates as much as 10 times.

Some results of these researches were implemented in VoluMill algorithms.

These can be used for all 2 axes and 3 axes rough milling from simple parts (prismatic one) up to complex freeform molds.

In addition, VoluMill is capable of dynamic adjustments to the axial depth of cut and/or the feed rate so as to never exceed a pre-set rate of material removal. As a result, VoluMill can be safely used at feeds and speeds many times greater than what is recommended by cutter manufacturers [3].

In addition to achieving much higher material removal rates than typical machining, VoluMill has also been shown to double, treble, or even quadruple Milling cutter life when used correctly [4].

In contrast with other types of High Speed Machining toolpaths, VoluMill toolpaths are typically used with axial depths of cut up to the full flute length of the Milling cutter. They are also effective in high speed shallow-axial, deep-radial cutting; a step over between passes of up to a radial depth of cut a_e of 100% of the tool diameter can be safely used without leaving any uncut material.

VoluMill is a revolutionary new programming tool path technology that corresponds to the new developments in cutting tools and machine tools by using existing CAD/CAM functionality for geometry and input/output functions. It dynamically manages cutting strategies, feed rate and depth of cut, and minimizes force and heat buildup in both the part and the tool by maintaining the programmed material removal rate.

The idea is to generate a path where the tool load is constant on the cutting tool.

3. CASE STUDY – USE OF A HIGH SPEED REMOVAL STRATEGY FOR STEEL MILLING

Our study is based on VoluMill strategies implemented in Cimatron E11 CAM software. The approaches were from 2 directions. One is a comparison with standard procedure and the second is from the right way of applied the VoluMill roughing procedure.

Our study is based on the fallowing steps:

1. prepare the part:
   a. planar milling;
   b. drilling a technological hole for (pre-drill beforer milling);
2. classic metal cutting strategy – roughing with parallel trajectories;
3. new strategy –volumill first scenario – roughing all part with one depth of cut of 4 mm and the second depth of cut of 10mm;
4. new strategy –VoluMill second scenario –roughing all part with one depth of cut of 14 mm and the second one with 4 mm;

In all three cases (from step 2, 3 and 4), the machining time was monitored.

The machining parameters were calculus based on the part material, type of tool, number of flutes, type of holder and depth of cut.

In Volume strategies, the depth of cut can be up to two times tool diametr.

These strategies can be applied for small and medium parts and cavities.

For the test we used a HAAS CNC Milling Machine VF2ss with the fallowing characteristics:

- Machine type: vertical milling center with working space *X* / *Y* / *Z*: 762/402/508 mm.
- Maximum workpiece weight 680 kg;
- Toolholder type:BT40
- Main spindle power: 22.4 kW;
- Main spindle rpm: 12 000 rpm;
- Maximum feed: 20 m/min;

The part material is steel C45 (Brinell hardness HB: 255) with the characteristics shown in Table 4.

The part dimensions are: X 110 mm; Y 105 mm; Z 115 mm.

A demonstrative 3D model was created in Cimatron E11 (Fig. 3) All dimensions are in millimeters

In order to underline the effects of VoluMill strategies some preparation machining were made.

Table 4

Chemical composition % of grade C45 (1.0503)

Cr + Mo + Ni = max 0.63							
C	Si	Mn	Ni	P	S	Cr	Mo
0.43 – 0.5	max 0.4	0.5 – 0.8	max 0.4	max 0.04	max 0.045	max 0.4	max 0.1

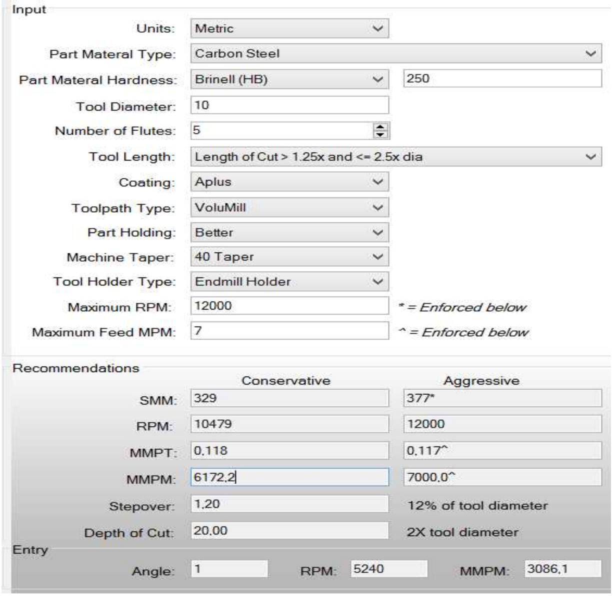


Fig. 2. Machining parameter determination.

For machining parameters calculus, the fallowing formulas were used:

$$v_c = \frac{C_v \cdot D^{q_v}}{T^m \cdot t_l^{x_v} \cdot s^{y_v} \cdot t^{w_v} \cdot z^{n_v}} k_v \,, \tag{1}$$

$$n = \frac{v_c}{\pi \cdot D} \cdot 1000 \,, \tag{2}$$

$$F = z \cdot n \cdot f_z \,. \tag{3}$$

where:
F – feed rate [mm/min];
x_v, y_v, w_v, n_v – exponents;
v_c – cutting speed;
C_v – cutting conditions coefficient;
$k_v = k_{ns} k_M k_k k_s$ – corrections coefficients taking into account the cutting tool material, design of the cutting edge, material characteristics, stock;
T – tool life in minutes;
t – depth of cut in millimeters;
s – feed per revolution [mm/rev];
t_l – length of the cutting edge [mm];
z – number of teeth of the cutting tool;
D – tool dimeter;
f_z – feed per tooth.

A software (*Milling advisor*) was used for fast calculation based on these formulas (Fig. 2).

First a face milling (Fig. 4) was made by using parallel trajectories with a Walter end milling tool with D = 40 mm and z = 6.

The cutting parameters used are: feed rate 1 591 mm/min; feed per tooth 0.1333 mm; cutting speed 250 m/min, spindle speed 1 990 rpm.

A second operation was drilling (Fig. 5) using a 16 mm drill with one cutting edge with 180 degree. The drilling parameters used are: feed 430 mm/min; feed per revolution 0.0793, cutting speed 272 m/min; spindle speed 5 420 rpm.

The third step was for roughing.

In order to compare the classic way with the new high speed material removal techniques two strategies were developed in Cimatron E11 software. The tool was an endmill tool with 4 teeth and 10 mm diameter.

First it was a classic roughing procedure with parallel tool path with 0.5 depth of cut. The machining parameters used were: feed 908 mm/min; feed per tooth 0.058 mm; cutting speed: 123 m/min, spindle speed 3 915 rpm.

The total time for this roughing procedure was 23 min and 43 sec (Fig. 6).

Secondly, a VoluMill technique was used for milling the same surfaces in a rough procedure with the same tool and using machining parameters: depth of cut 14 mm; feed 7 376mm/min; feed per tooth 0.18 mm; cutting speed: 314 m/min; spindle speed 10 000 rpm.

In this conditions, the total time was 2 min and 23 sec (Fig. 7).

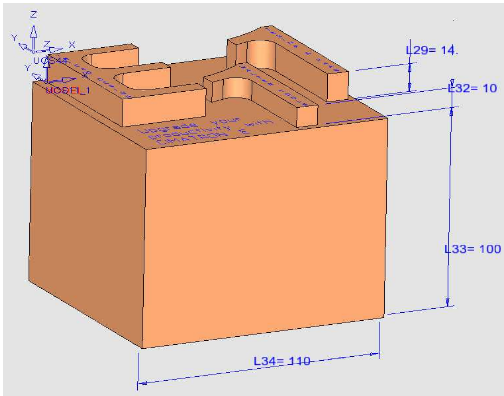


Fig. 3. Demonstrative part.

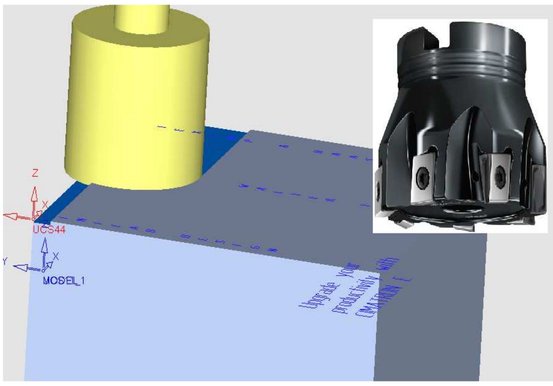


Fig. 4. Face milling.

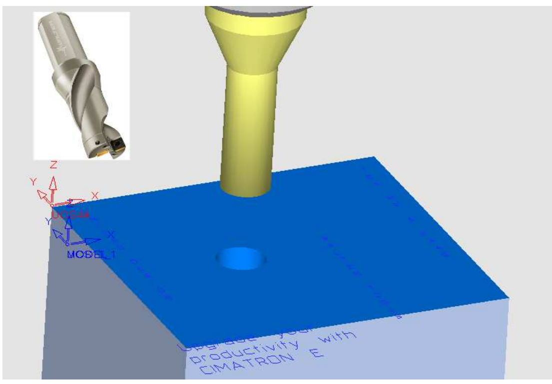


Fig. 5. Drilling.

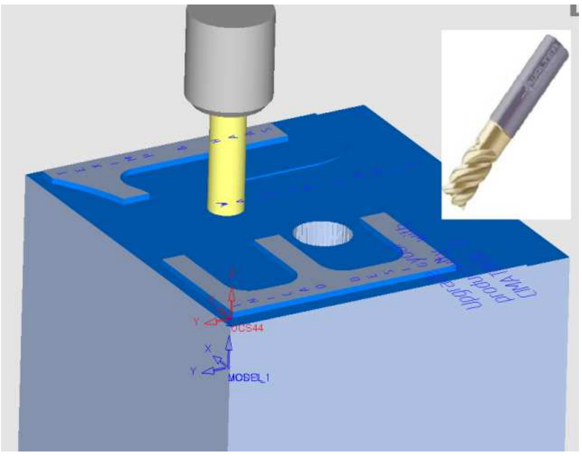


Fig. 6. Classic roughing procedure.

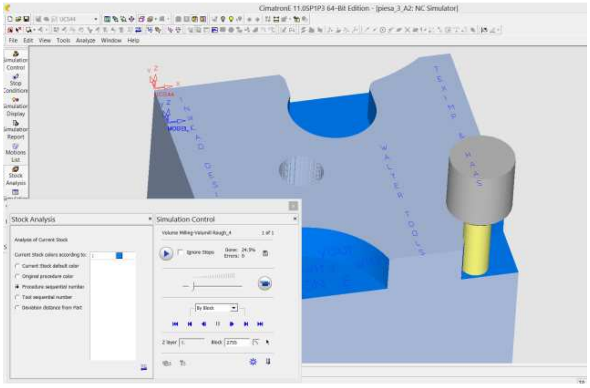


Fig. 7. VoluMill procedure.

According to the part topology, the toolpath was generated in order to eliminate some classic features of the cutting techniques. The way of getting of the tool in and out material was eliminated. The toolpath is special designed for entering the material without any supplementary load of the tool.

In our case, it is allowed to engage from outside of the material. For pocketing, it is recommended to make a hole with a drill with 180 degree. For VoluMill, the starting point will be the center of this hole.

In Fig. 8, one layer of toolpath for classic spiral strategy is presented. As it can be noticed, the trajectories are linear and circular with a lot of inflexion points. In this case, the cutting conditions are not constant and leding to the conclusion that the cutting tool is not being used properly. Also, this is unfavorable for the machine tool. The variations of the cutting forces are inducing a vibratory behavior of the machine having negative influences over the part.

In Fig. 9, the results for VoluMill toolpath are presented. In this case, there is only one depth of cut of 14 mm.

The trajectories are complex curve generated based on VoluMill algorithm for constant load on the cutting tool. For linear movements, there are some variations in the feed. The engaging of the tool in material is done step by step over a curve.

In the NC file it can be noticed some particularities. The feed is not constant over the linear trajectories in order to keep the same load on the tool. Some variations can be quite big but they will not change the cutting condition.

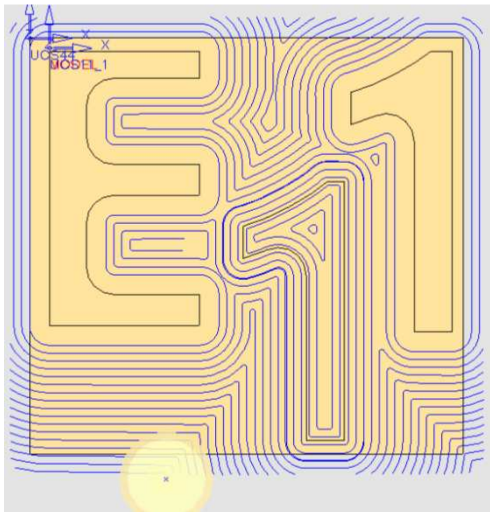


Fig. 8. Spiral toolpath.

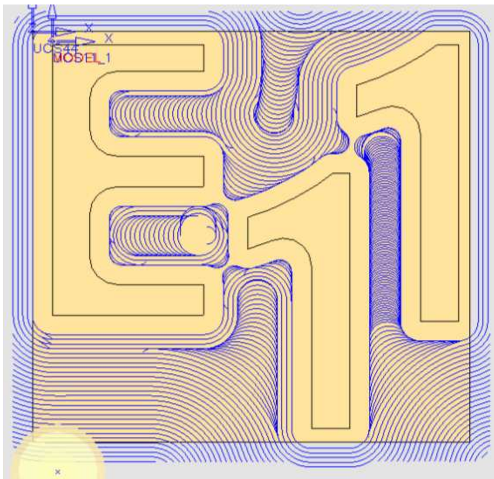


Fig. 9. VoluMill toolpath.

For the curves it is the same feed in all cutting points. In this case, the tool load is calculated based on the form of the trajectories. There are small variations in the depth of cut in order to keep a constant feed (Fig. 11).

Also, it can be noticed a large number of cutting points on linear trajectories. Usually, for a classic cutting in linear movement, the tool trajectory is starting from current point P_0 and reaches the endpoint P_1 (G1 X_1 Y_1 Z_1 F), where the start point is (X_0, Y_0, Z0) and (X_1, Y_1, Z_1) is the target point of the tool, F being the feed in mm/min) (Fig. 12).

For VoluMill linear trajectories there are a number of points between P_0 and P_1:

$$\sum_{imin}^{imax} P_i = \sum_{imin}^{imax} G1\ X_i Y_i Z_i\ F_i\ . \qquad (5)$$

The main advantage is that the machined surface is very well covered by the tool and the feed is optimized in each point (Fig. 13).

At the first sight, it seems that the NC file for VoluMill technics will be bigger than that obtained with classical milling strategies. But it is not. In VoluMill toolpath even if there is a bigger number of cutting points there are only few cutting depths (one or two) (Fig. 14), depends of the active apart of the cutting tool. In classic strategies there are a lot of cutting depths (eg. for steel it could be of 0.5 mm, which means that for a total height of the excess material of 14 mm, 28 paths are necessary (Fig. 15) each path having associated the same tool trajectory.

The NC file with VoluMill, after postprocessing with the same NC postprocessor, has 5 485 lines. For the classic toolpath studied (roughing with parallel trajectories), the number of lines in the NC file is 6 544.

The NC file dimension for VoluMill is 173 kB and for parallel roughing is 296 kB.

```
N3119 Y-58.7
N3120 Y-49.29 F7374.1
N3121 Y-4.56 F7376.
N3122 Y-4.26 F6803.6
N3123 Y-3.97 F7376.
N3124 Y-3.67 F7037.1
N3125 Y-3.38 F7376.
N3126 X0.02 Y-3.08
N3127 X0.05 Y-2.79
N3128 X0.08 Y-2.65
N3129 X0.1 Y-2.5
N3130 X0.14 Y-2.36
N3131 X0.17 Y-2.22
```

Fig. 10. VoluMill NC file-linear trajectory definition.

```
N4677 X91.53 Y-32.02 F15000.
N4678 X91.8 Y-31.87 Z-14.92 F7376.
N4679 X92.04 Y-31.67 Z-14.93
N4680 X92.24 Y-31.43 Z-14.95
N4681 X92.38 Y-31.15 Z-14.97
N4682 X92.47 Y-30.86 Z-14.98
N4683 X92.5 Y-30.55 Z-15.
N4684 Y-30.43
N4685 Y-30.42 F3641.
```

Fig. 11. VoluMill NC file-complex trajectory definition.

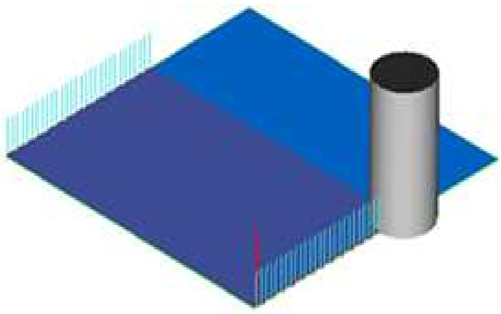


Fig. 12. Classic NC linear interpolation with G1.

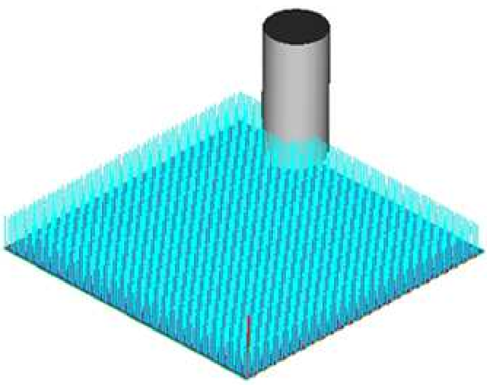


Fig. 13. VoluMill NC linear interpolation with G1.

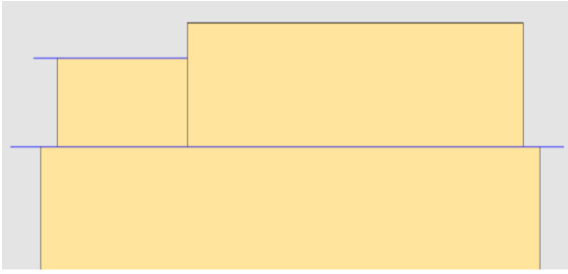


Fig. 14. Depth of cut for VoluMill (14 mm) one path.

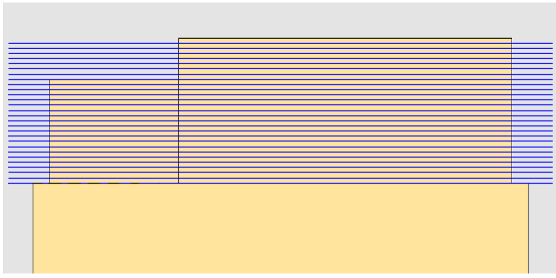


Fig. 15. Classic parallel roughing (0.5 mm) 28 paths.

4. CUTTING STRATEGY FOR APPLYING CORRECT VOLLUMIL TECHNIQUES

A second study presented in this paper is the implementation of the VoluMill in part machining taking into account the part topology.

Studying the part, it can be noticed two steps (Fig. 3). In the first step the cutting is done with 4 mm depth and the second one with 14 mm depth. At the first view, the logic is to machine the first depth and then the second depth.

If we apply the VoluMill strategy in this manner, a machining time of 4 minutes and 57 sec will be obtained (Fig. 16, a).

But in this strategy the correct way is to use the biggest depth first and then continue with the smaller depth but only in the un-machined area (Fig. 16, b). The machining time in this case is 2 min and 23 sec.

a b

Fig. 16. Cutting strategies for VoluMill.

5. CONCLUSIONS

It wastes a lot of time when the tool is engaged in the material. The feed is changed and the trajectories are complex (starting with a ramp entering and ending with a complex helix with a very small radius and angle).

Another inconvenient is that there are a lot of rapid and max feed movements between cuts.

Also in corners there are some unwanted phenomena like overloading the tool, very fast acceleration and deceleration of the linear axis of the machine. This will affect the tool life and also the machine.

This traditional known problems will affect in the end the quality of the machined parts and the costs due to the longer machining time.

The tool path has always been the weak link in the chain machine tool-cutting tool-CAM software. This was an aspect which was not taken into account when the machining parameters were calculatedd.

VoluMill is a revolutionary programming tool path technology that follows the new developments in cutting tools and machine tools by using existing CAD/CAM functionality for geometry and input/output functions.

In VoluMill toolpath the strategies of tool engagement in material are eliminated. On the all length of the trajectory, parameters as the axial depth of cut and feed are controlled in order to obtaine a constant load on the tool.

This means long tool life, less vibration on the machine, lower tool load (lower cutting forces and torques).

The tool is optimal used by cutting with all the active part.

The cooling is made only with air, all heat being eliminated through the chips.

By using VoluMill techniques there are a very good impact from the ecological point of view:

1. The cooling liquid is not used (it is known that it is very toxic for the environment);

2. Obtaining lower loads in the cutting, the power needed from the machine tool will be smaller, so the energy consumption will be optimized.

3. Also by obtaining a lower machining time, the machine will work less, so the energy consumtion will be optimized.

In the presented case study some conclusion are obtained, as follows:

The VoluMill techniques can be succesfuly applied on all types of machine tools.

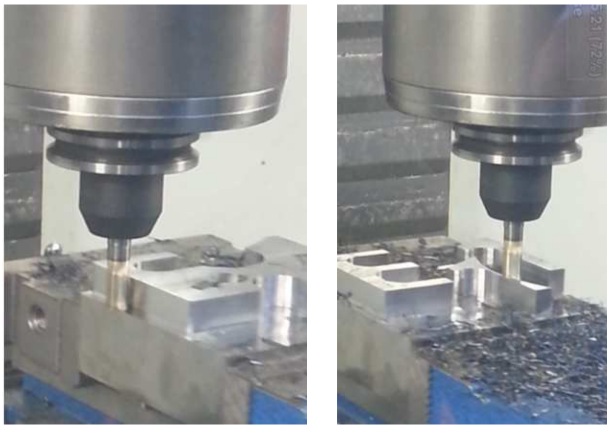
Due to less vibration it can be used for roughing on less stiff machines.

The machining time is about 10 times smaller than in classic machining.

The best way is to start with the maximum depth of cut and then to machine and remachine with smaller depths.

In Fig. 17,a it is presented the cutting test made for a maximum depth of cut of 14 mm. Figure 17.b shows machining with the second depth of 4 mm (smaller). The maximum amount of material was machined in the first case.

This technique increases the cutter life of two, three or four times when used correctly.



a b

Fig. 17. VoluMill cutting test: *a* – test with depth of cut *t* = 14 mm; *b* – test with *t* = 4 mm.

The maximum depth of cut can be up to two times the tool diameter.

The tool path technology makes dynamic adjustments to the depth of cut and/or the feed rate so as not to exceed the pre-set volume of material being removed at any given time.

As next research, we propose to measure the forces and moments during cutting for the analysis strategies in order to establish the optimal machining parameters.

REFERENCES

[1] C. Ispas, M.D. Paraschiv, L. Laboreau, F.D. Anania, *Research concerning the numerical errors correction for a NACA profile surface on a 5 axis machine tool*, Academic Journal of Manufacturing Engineering, Ed. Politehnica, vol. 6, ISSUE 3/2008, pp. 75–81.

[2] C. Ispas, C. Mohora, F.D. Anania, M.D. Paraschiv, *Virtual design methods for the new products*, Proceedings of the International Conference on Manufacturing Systems ICMaS, 22 November, 2007, Bucharest, Romania, pp. 105–108.

[3] G.E. Dieter, *Mechanical metallurgy*, SI metric edition, McGraw-Hill, 1988.

[4] L. Edwards, and M., Endean, Manufacturing with materials, Butterworth Heinemann, 1990.

[5] J. Beddroes, M.J. Bibbly, *Principles of metal manufacturing processes*, Arnold, 1999.

[6] G. Coleman, *Tool paths: weak link no more* `http://www.fabricatingandmetalworking.com/2010/01/tool-paths-weak-link-no-more`.

[7] M. Zapciu, D. Anania, C. Bisu, *Conceptie si Fabricatie Integrate* (Integrated design and manufacturing), Bren Publishing House, 2005.

[8] C. Ispas, M. Zapciu, C. Mohora, D. Anania, *Product development using CAD-CAM-CAE software and internet facilities,* The International Conference of the Carpathian Euro-Region Specialists in Industrial Systems, Baia Mare, Romania, 2006, pp. 1224–326.

[9] D., Anania, M. Zapciu, C. Mohora, *Modelling of the PC MILL 100 Machine tool and milling process using DELMIA V5R17*, The 19th International DAAAM SYMPOSIUM "Intelligent Manufacturing & Automation: Focus on Next Generation of Intelligent Systems and Solutions", 22–25th October, 2008, pp. 0607–0608.