Journal of Machine Engineering, Vol. 16, No. 1, 2016

Received: *22 January 2016* / Accepted: *03 February 2016* / Published online: *10 March 2016*

CNC, postprocessor, feed-rate, optimization

Vaclav ZABRANSKY[1*]
Petr VAVRUSKA[1]

PRODUCTIVE MACHINING OF COMPLEX SHAPE PARTS

This paper deals with technological conditions that are achieved during multi-axis milling operations. The work is focused on the improvement of quality and productivity of machining of complex shape parts. The importance of feed-rate analysis during multi-axis machining operations is mentioned. A postprocessor algorithm for correction of relative feed-rate during multi-axis machining operations has been proposed. The postprocessor generates corrected feed-rate for each block of an NC program when a multi-axis interpolation is found. Thanks to this new postprocessor algorithm, better conditions are achieved during multi-axis machining operations and also a shorter time for NC program execution when it is used without the Tool Center Point transformation. Practical benefits and the functionality of the postprocessor algorithm have been tested by machining the blade containing multi-axis tool paths.

1. INTRODUCTION

NC program creation is a complex process. An NC program should reflect the experience of its creator – the production engineer. Nowadays it is possible to produce almost any shape of the product by five-axis CNC machine tools and it is necessary to use CAM software for generating CL data (cutter location data) for these complex-shaped parts. The translation of CL data to an NC code that can be processed by a real machine tool is done by the compiler called postprocessor. This compiler affects the form of the generated toolpath and also generates various technological functions. The postprocessor can be programmed to generate various improvements and features that are activated based on the settings in the CAM system or CL data. The production engineer should know if there are some cases in which the CAM system used can generate a toolpath that can have a bad influence on the tool movement. The manner in which the toolpath is prepared has an influence on the machine tool behaviour. The machine tool behaviour has an influence on the machining time and the quality of the machined part. In practice there is often very little time for order processing and hence it is not possible to try new approaches. Therefore, we can find the use of spline interpolations mainly at the academic level, where only the

[1] Czech Technical University in Prague, Faculty of Mechanical Engineering, Department of Production Machines and Equipment, Prague, Czech Republic
[*] E-mail: v.zabransky@rcmt.cvut.cz

mathematical apparatus is analysed mostly, e.g. in the literature [10],[7] and [8]. The application of multi-axis spline interpolation in one CAM system can be found in the literature [5].

In the past, many attempts were made to optimize the feed-rate. The vast majority of solutions focused on generating the feed-rate with respect to the cutting depth (or currently removed volume of material). Different algorithms are used in calculating the feed-rate and mathematical models of cutting tools are also adjusted to their continuous optimal cutting geometry development. This issue is the subject of the following texts in the literature[1],[2],[3],[6]. The principles on which the calculation method for calculating the feed-rate is based can be found in these papers, but none of them deals with the actual machine tool control system and rotary axes configuration of machine tools used for machining.

Therefore, this paper deals with the possibilities of increasing productivity of machining of complex shaped parts by influencing cutting conditions in the NC code done by the postprocessor. Tests were performed on the machine SP430 1100 SY 2 and partly on the machine Multicut 630, both from the production of Kovosvit MAS and both with the control system Sinumerik 840D sl. CAM software used for creating the toolpath was NX 9.

2. POSTPROCESOR WITH THE FEED-RATE CONVERSION

Machining of geometrically complex parts is often confronted with the requirement for non-constant cutting conditions. It means affecting the feed-rate or spindle speed during the cut so that constant cutting speed is achieved at the point of contact with the workpiece or so that spindle load is constant. Setting up the feed-rate regulation using CAM software seems to be the most appropriate solution. Many CAD/CAM systems are able to compute optimized toolpath for roughing operations. In the case of SurfCAM production engineers can use the function TrueMill to create the toolpath that are optimized to keep constant wrapping of the tool so the tool can move with higher velocities. If they should use e.g. the MasterCAM or the HSMworks, they are able to apply adaptive toolpath. Adaptive toolpath are alternative functions to the TrueMill but the calculation uses other principles. There are also various simulation systems that can verify the NC program and even offer the possibility of automatic NC code modification, for example the Vericut system from CGTech [12], which provides the OptiPath function OptiPath, which modifies the feed rate along the toolpath (Fig. 1). If we use this function, the feed-rate in NC program is adjusted to the current machined material volume in each tool step. This optimization is based on the volume of current material removal. Other systems with a similar function are WorkNC from Sescoi [14] or NCsimul from Spring [13]. However, none of CAD/CAM systems is able to adjust the feed-rate during multi-axis machining owing to the current position of the tool against rotary axes. This optimization can be achieved by the postprocessor.

The postprocessor with the possibility of machining productivity improvement is dealt with most in [9]. The possibility is mentioned that the feed-rate correction algorithm for four and five-axis milling is applicable if the control system function Tool Center Point

(TCP) is not in use. The algorithm was implemented in a postprocessor (Fig. 2 and Fig. 3) and the generated NC code contains the feed-rate correction (Fig. 4), which can be seen in in the red frames.

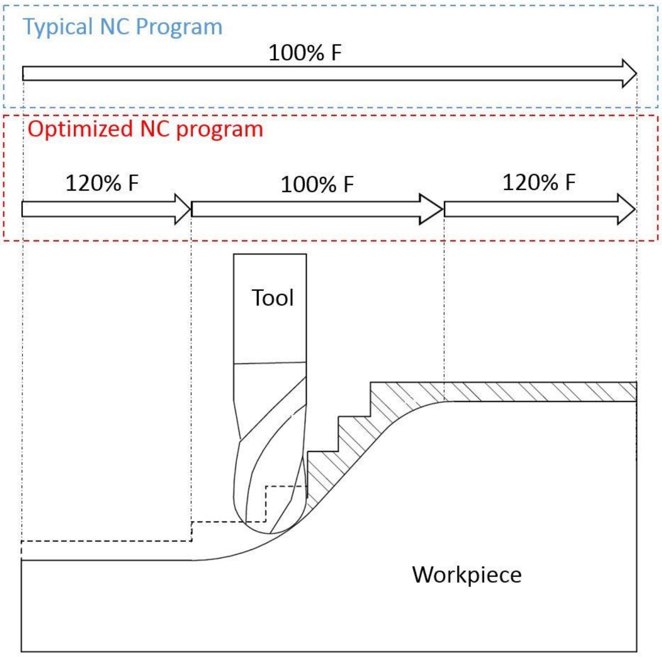

Fig. 1. OptiPath function

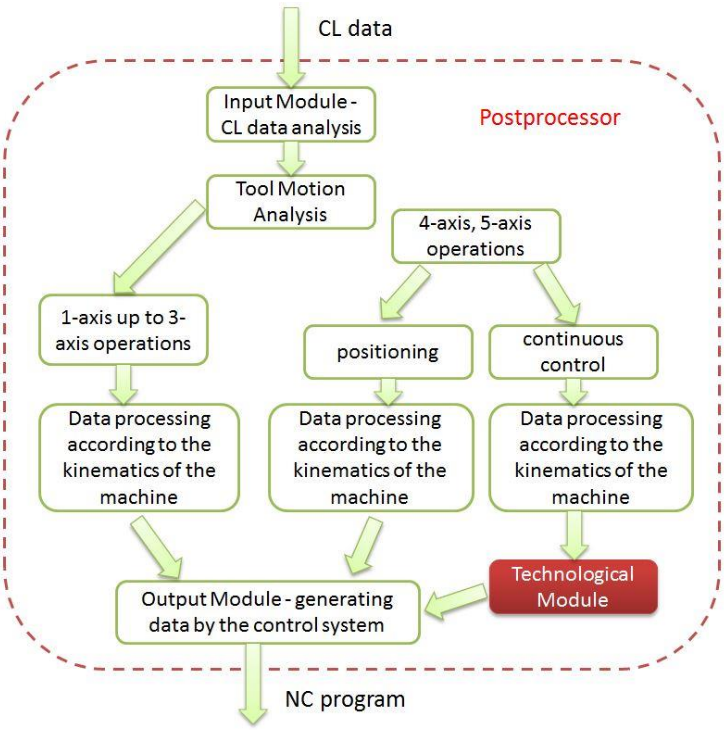

Fig. 2. Schematic location of technological module postprocessor

Fig. 3. Implementation of the technological module in NX Post Builder

```
N1950 ;Cutting
N1960 F348.843
N1970 Y-5.796 CA1=DC(40.81)
N1980 F417.064
N1990 X1.926 Y-5.915 Z-10.818 CA1=DC(40.683)
N2000 F658.977
N2010 X2.016 Y-6.148 Z-10.832 CA1=DC(40.281)
N2020 F831.971
N2030 X2.119 Y-6.376 Z-10.847 CA1=DC(39.774)
N2040 F1171.170
N2050 X2.248 Y-6.595 Z-10.862 CA1=DC(39.061)
N2060 F1738.268
N2070 X2.336 Y-6.698 Z-10.869 CA1=DC(38.524)
N2080 F1811.384
N2090 X2.417 Y-6.788 Z-10.876 CA1=DC(38.024)
N2100 F3077.219
```

Fig. 4. NC code after correction

An older version of similar algorithm was proposed in [4]. It is the same method of application (in postprocessor), but different coefficients of feed-rate correction (K_V in the case of [9] and K_F in the case of [4]) are computed in this literature for conversion of the programmed feed-rate. The coefficients are not identified experimentally or by measurement. K_F and K_V are determined based on the feed-rate prediction in the postprocessor. This method of correcting the feed rate is based on the proportion of the distance of the path points in CL data to the distance of the path points of the NC program. While K_V considers the effect of the distribution of the feed rate for rotary axes, K_F does not. The comparison of the impact of these coefficients on the productivity can also be seen in this paper.

3. MACHINING OF THE BLADE

The blade used for testing can be seen in Fig. 5, where the toolpath of machining the blade can be seen. The tool used for finishing (spherical mill) is parallel to the surface of the blade normal. This is only a testing operation. The final operation is based on the use of the leading angle.

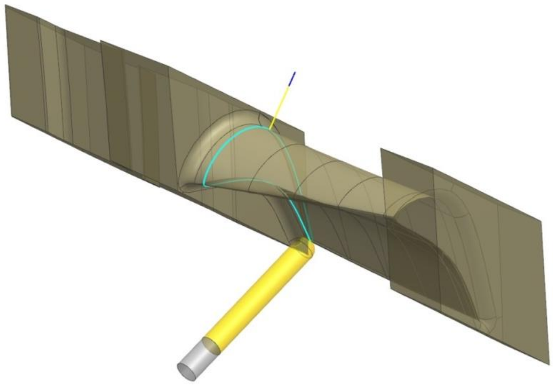

Fig. 5. Testing toolpath on the blade surface

The correction coefficients of feed-rate generated by the postprocessor during the machining of this toolpath are shown in Fig. 6. For clarity, the block numbers where the corrected displacements were generated were placed on the X-axis. Areas with the greatest displacement corrections can be identified in the CIMCO environment. In this case, they were to be found on the edge of the blade where it is necessary to drive around a small radius (trailing edge), see Fig. 6.

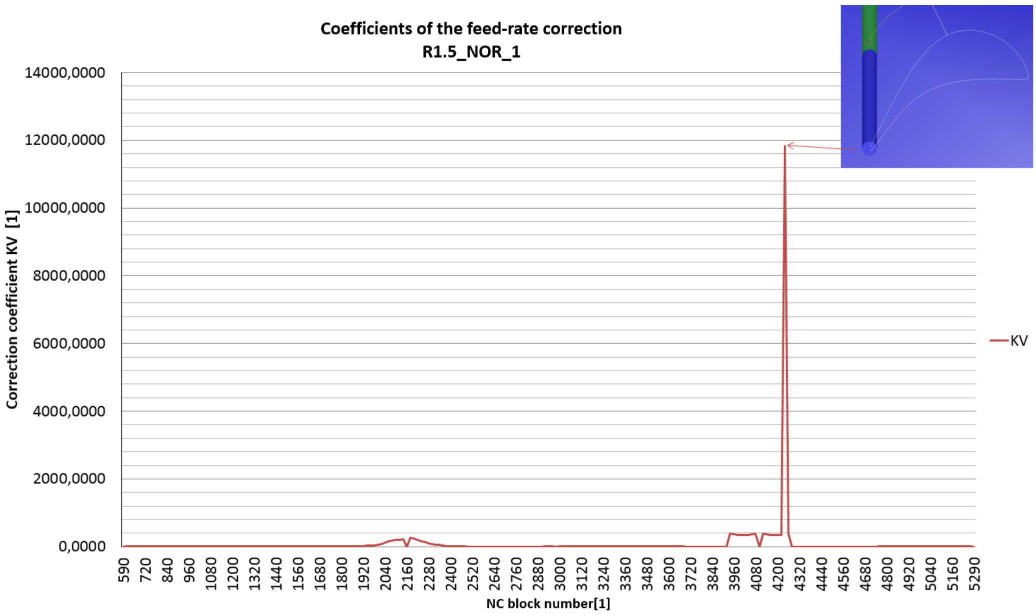

Fig. 6. Graph of the correction coefficient KV according to the block of the NC program

Therefore, an improvement was made to this technological module. Another condition was introduced to the model, the condition being that the correction coefficient in the next block should not be greater than a specified limit. The principle of this filter is to set the maximum correction coefficient step. It is written in the technological module that the correction coefficient in the next block of the NC program must not be greater than the previous + proportional constant. If so, the following corrected coefficient is set to the previous correction coefficient + proportional coefficient. For example Fig. 7 shows the result when the proportional coefficient is set at 200. This value was set because of filtering of an unwanted large step in block 4200 (Fig. 6). The lower the value, the smaller jumps will be filtered. The value of proportional coefficient is set by the user of the CAM software. Fig. 7 shows two areas with an appreciably greater correction coefficient. These can be seen in the areas around the leading and trailing edges of the blade. The higher correction coefficient is a consequence of the large curvature of the surfaces.

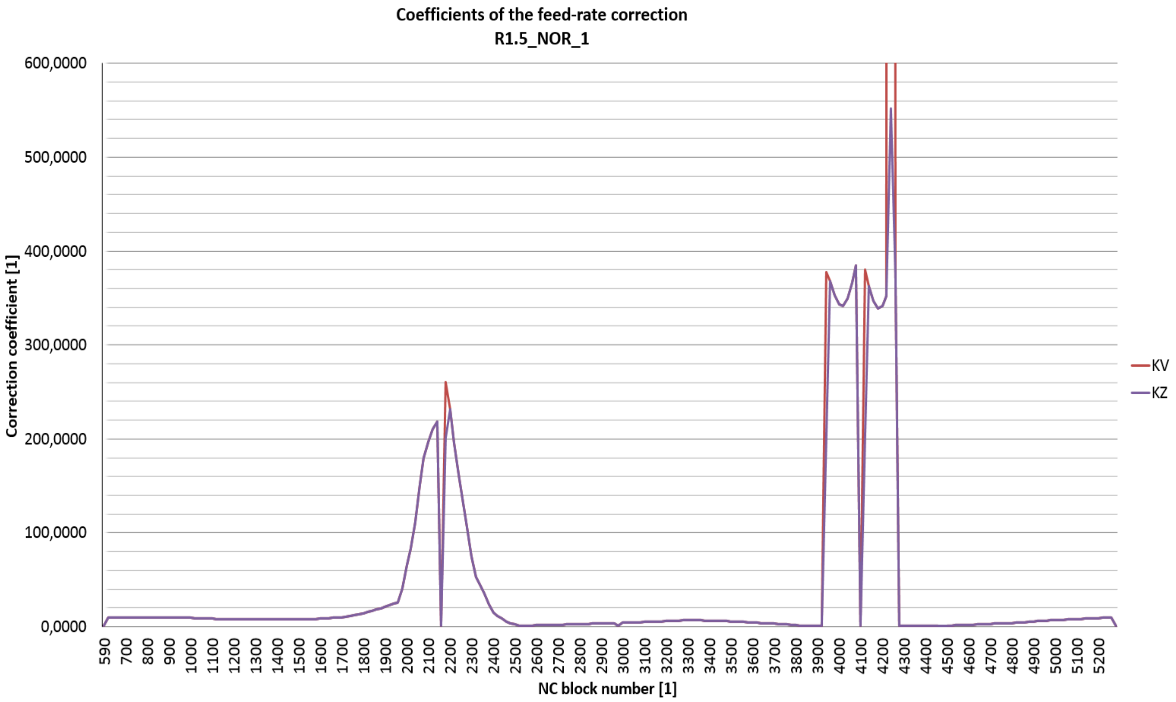

Fig. 7. Comparison of not smoothed (KV) and smoothed (KZ) feed-rate correction coefficient depending on the number of block of the NC program

In this case the large step of programmed feed-rate occurs within one NC block. Although it is required by the technology, we work with real machines and drives so that a step of such magnitude is not desirable in terms of fluency and cutting compliance with good quality and the quality of the workpiece surface. That it actually is not such a great step of feed-rate may be greatly influenced by the Look Ahead function, which could smooth out the step to some extent. Look Ahead might not always be switched on. Therefore it is better for large steps which would rather deteriorate than increase machining productivity, to be partially smoothed by the postprocessor.

The size of the of the correction coefficient step was set in the technological postprocessor module. The result of this extension is shown in Fig. 7. It can be seen that the large steps were filtered by new coefficients KZ, computed using an improved algorithm.

These corrections were now prepared for real machining of the blade on the four axis machine tool MAS SP 430 SY 2 1100.

4. THE TOOLPATH ERRORS AND THE CHOICE OF FINISHING STRATEGIES

Continuous machining around the leading edge was not considered because of the complexity of blade geometry, which is shown in Fig. 5. Multi-axis milling around the small radius on the edges leads to considerable unproductivity when the TCP function or correction of feed-rate is not used. This unproductivity is seen in Fig. 6, where the feed-rate correction coefficient around the edge of the blade reaches approximately the value of 12 000 [-]. This means that if the function TCP is not active, the feed-rate would be very small, and certainly the technology conditions are not respected. This is one of the reasons why it is necessary to think about the setting of finishing operations. Another reason for the setting of finishing operation was that the NX CAM system has a problem with generating toolpath. It is recommended to use the Streamline function for manufacturing complex shapes. There is an advantage in the possibility of scalloping any shape of workpiece by Streamline. The issue with the streamline setup is a different control surface and surface to be machined. Streamline creates a surface on which the grid is given by the chosen criterion, as the above-mentioned scallop or boundary of machining. This grid is then projected onto the surface to be machined. However, there may be errors and issues in this projection. There are significant errors in the finishing toolpath if we look closer at the part in Fig. 8 Toolpath is green and the errors are in the red rectangles.

There are noticeable jerking movements of the cutter on the edge of the blade. The cutter even performs a jerking movement backwards and forwards. Toolpath errors can be seen in Fig. 8. They are more evident when setting stricter CAM tolerance. In this case the tolerance is set to the value of ± 0.001 mm. The surface is undercut in the size of tenth of millimetres as shown in Fig. 9. Toolpath errors on the edges of the blades were removed by finishing the edges separately in three axes.

Finishing of the suction and pressure sides (Fig. 10 and Fig. 11) is a four-axis operation, when the mill D4 R1.5 is led with an angle of 10° to the workpiece surface. Toolpath errors in Fig. 8, as well as consequences in Fig. 9, were removed by the adjustment of CAM tolerance. CAM tolerance was set to +0.018 mm (Fig. 12) by sequential testing and then the least toolpath errors have been achieved. Compared to the usual tolerance of the blade (+0.1 mm), the CAM tolerance is assessed as sufficient. It is even five times stricter. All CAM errors could not be removed completely (Fig. 13), and therefore are only minimized.

The blade machining was made on a SP 430 SY 2 1100 machine from the production of Kovosvit MAS. The machine stands in the laboratories of the Czech Technical University in Prague. NC codes were generated by an improved postprocessor. A test was run concerning the influence of the control system setting on the final quality of the blade surface and machining productivity on this machine.

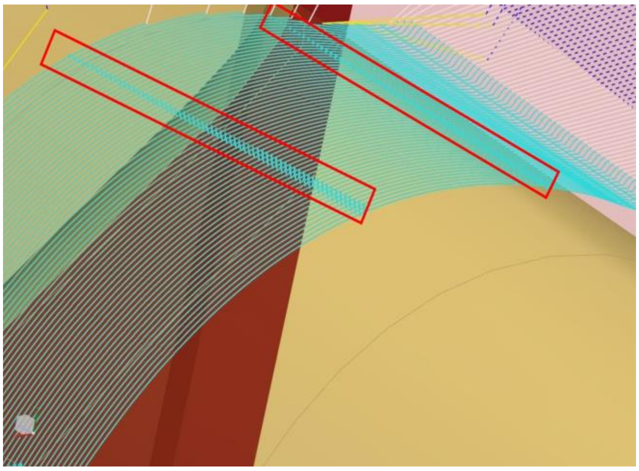

Fig. 8. Toolpath errors (marked by red rectangles)

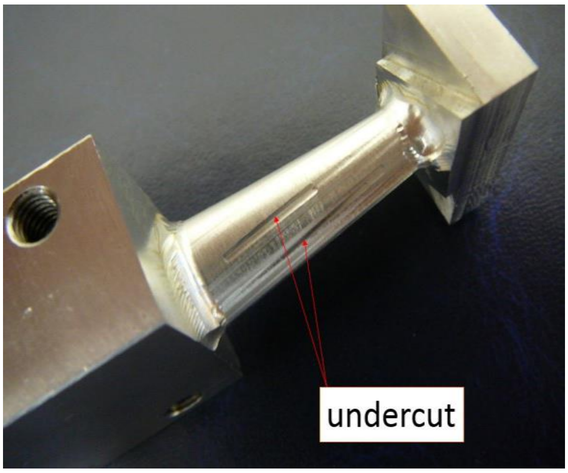

Fig. 9. Consequences of toolpath errors

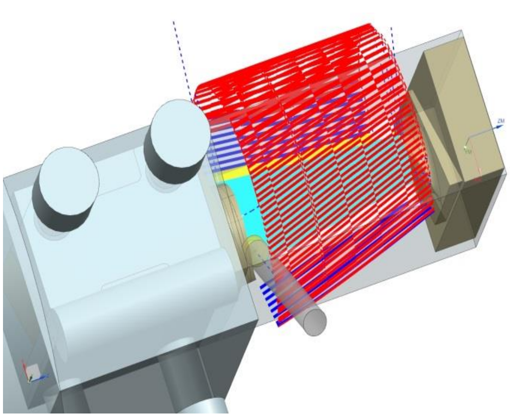

Fig. 10. Suction side finishing

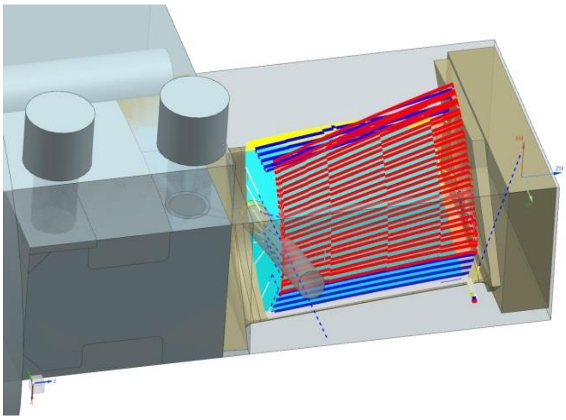

Fig. 11. Pressure side finishing

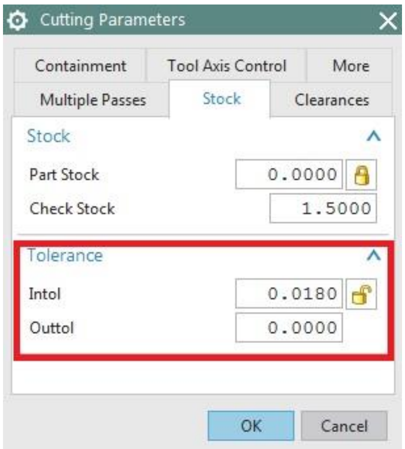

Fig. 12. CAM tolerance setting

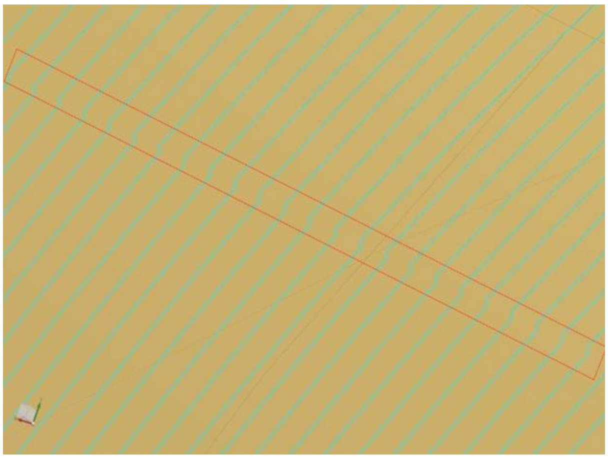

Fig. 13. Toolpath errors (marked by red rectangle)

The test was designed in the following manner. For the same finishing toolpath, an NC code was generated by the postprocessor as follows:

1. Finishing with active TCP;
2. Finishing without TCP;
3. Finishing without TCP with active technological module - KV correction (resp. KZ);
4. Finishing without TCP with active technological module – KF correction.

5. THE TEST OF CHOSEN STRATEGIES PRODUCTIVITY

For verification and mutual comparison, it would be advisable to perform this test on two different machines by the same producer: SP430 and Multicut 630. It was not possible to carry out these four tests on Multicut 630 because of its busy timetable in production.

Table 1. Testing on the SP430 SY2 1100 machine tool

Test	Side	Time [min:s]
1) With TCP	Suction side	37:49
	Pressure side	32:53
2) Without TCP	Suction side	34:41
	Pressure side	33:16
3) Without TCP, KV	Suction side	33:15
	Pressure side	25:43
4) Without TCP, KF	Suction side	38:55
	Pressure side	32:23

Table 2. Testing on the Multicut 630 machine tool

Test	Side	Time [min:s]
With TCP	Suction side	39:29
	Pressure side	37:27

According to the results listed in Table 1, it can be concluded that the most productive strategy was the one without TCP and with KV feed-rate correction. Very similar results are achieved by using TCP and without TCP - with KF feed-rate correction. It is an interesting finding that the switching off of the TCP function can sometimes bring higher productivity than active TCP, as was the case with the suction side of the blade. The surface quality is evaluated in the following section.

6. THE SURFACE QUALITY

The equipment Subtronik 3+ was used for measurements which can determine what surface quality has been achieved for milling. The surface was measured in a direction perpendicular to the line of flow and also in the direction of flow. We can assess sufficient ”scallop”, a parameter influencing the number of passes of the tool, according to the surface perpendicular to the direction of streamlines.

A characteristic of the measured surface on the suction side of the blade can be seen in Fig. 14 The characteristic is already filtered from the overall waviness shape and should therefore constitute notches in the otherwise ideal surface.

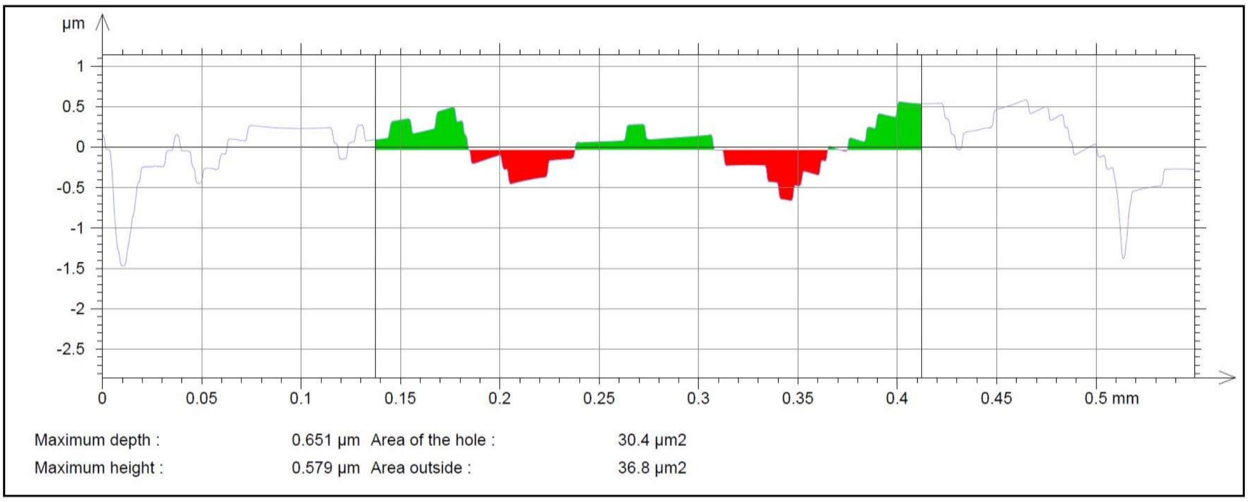

Fig. 14. The measurement of surface profile on the suction surface of the blade in a perpendicular direction to the flow

As can be seen in Fig. 14, the height of inequality at the measuring point is about one micron, sometimes up to two microns. The Scallop set on CAM was one micron. The distance between toolpath lines was 0.13 mm. It approximately corresponds to the distance of peaks and holes highlighted in red and green.

Figure. 15 and Fig. 16 shows an example of measurements of suction and pressure side surface profile, in this case of the blade milled by NC programs based on TCP function. The area of measurement is shown by the orange line and is listed in the characteristic by the value of the difference between the largest peak and the lowest bottom within the evaluation length of approximately 0.5 mm; it should correspond with roughness parameter R_z. This parameter should also correspond to the scallop parameter in the CAM software and therefore we can directly affect the resulting surface roughness. Evaluation length of 4 mm is recommended for roughness parameter R_z between 0.5 and 10 mm according to standards. 4 mm evaluation length was not possible according to the frequency of surface defects and according to the curvature of the blade; therefore the results can only be comparative values between these blades.

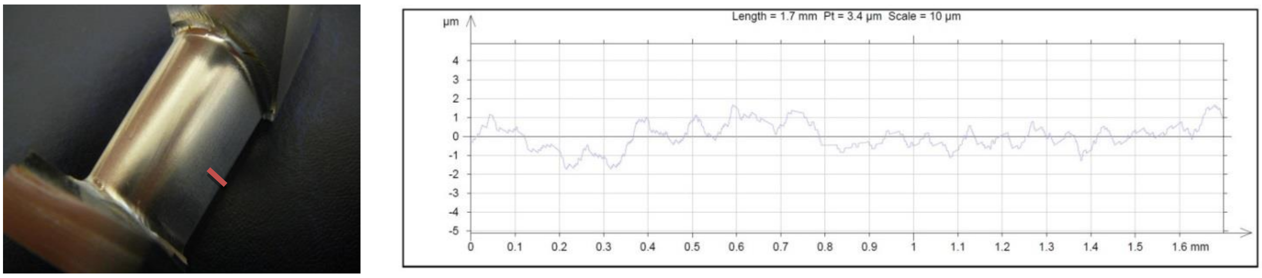

Fig. 15. The example of surface measurement of suction side of the blade milled by NC programs based on TCP function

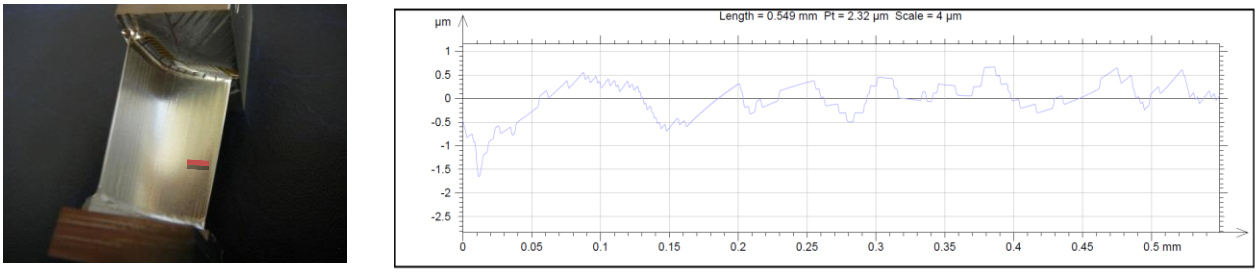

Fig. 16. The example of surface profile measurement of pressure side of the blade milled by NC programs based on TCP function

The evaluated surface is always in the place where there are no surface defects. Surface defects, according to standards CSN EN ISO 4287, are not counted into the surface roughness, but play an important role, therefore they are also evaluated. All results are listed in Table 3, which also shows the evaluation of surface defects. The smallest evaluation number shows the result with the least defects, while the highest number indicates a surface with numerous defects.

Table 3. Comparison of achieved surface quality

Blade	R_z value [µm]		Evaluation of surface defects [-]	
	Suction side	Pressure side	Suction side	Pressure side
1) With TCP	3,4	3,3	3	3
2) Without TCP	1,4	2,4	5	3
3) Without TCP, KV	3	2,1	5	5
4) Without TCP, KF	3,1	4	3	3
5) With TCP – Multicut 630	1,8	1,6	2	2

Regarding the surface quality, the best results are achieved when using TCP. The blade machined by TCP comprises less visible defects compared to the others. A very similar surface of the blade was machined without TCP with KF feed-rate correction, with comparable productivity. It can be seen that by using KV coefficient, productive operations as well as suitable roughness can be achieved.

Surface differences are also visible between the blade machined on the machine tool SP430 and Multicut 630. The blade from Multicut 630 has fewer surface defects and roughness values R_z are approximately twice as good as the blade from SP430.

7. CONCLUSION

The findings mentioned in this paper can be interpreted as possibilities how cutting conditions as well as productivity can be influenced by suitable adjustment in the postprocessor. A proposal was made to extend the existing technological module to include filtering of large steps in feed-rate. Simultaneously, the real influence of a variously modified NC code on the productivity of blade finishing operations and surface quality was tested. It can also be seen that linking simulation models in the CAM system with a postprocessor has potential for future increasing of machining productivity. This paper shows that even CAM software is not an omnipotent tool for creating toolpath and should be used cautiously. The achieved results point to the fact that the TCP function can be well substituted by the proposed correction function of the postprocessor, but the results do not fully answer the question when it is better to use the control system function TCP and when the proposed correction. The results clearly show that by using the proposed postprocessor function for feed-rate correction the productivity of machining is higher than in the case without TCP and also in some case with TCP. The results also indicate that the same productivity and surface quality can be achieved by using TCP, even without TCP using the appropriate corrections, as in the case of blades 4) in Table 3.

ACKNOWLEDGMENTS

This paper has received funding from the Technology Agency of the Czech Republic.

REFERENCES

[1] BAEK D.K., KO T.J., 2008, *Feedrate scheduling for free-form surface using an NC verification model*, In International Journal of Machine Tools & Manufacture, 48/2, 163-172.

[2] FENG H.Y., SU N., 2005, *Integrated tool path and feed rate optimization for the finishing machining of 3D plane surfaces*, In International Journal of Machine Tools & Manufacture, 40/11, Amsterdam, Elsevier, 1557-1572.

[3] FERRY W.B.S., 2008, *Virtual five-axis flank milling of jet engine impellers*, Vancouver, A thesis submitted in partial fulfillment of the requirements for the degree of doctor of philosophy, The University of British Columbia.

[4] FORNŮSEK T., RYBÍN J., 2008, *Optimalization of NC code*. In Proceedings of International Congress MATAR PRAHA 2008, ISBN 978–80–904077–0–1, Praha, 137-141.

[5] LANGERON J.M. et al., 2004, *A new format for 5-axis tool path computation, using Bspline curves*, Computer-Aided Design, September, 36/12, 1219-1229.

[6] LEE H. U., CHO D.W., 2003, *An intelligent feedrate scheduling based on virtual machining*, In The International Journal of Advanced Manufacturing Technology, 22/11-12, 873-882.

[7] MÜLLER M. et al., 2004, *High accuracy spline interpolation for 5-axis machining*, Computer-Aided Design, November, 36/13, 1379-1393.

[8] PATELOUP V. et al., 2010, *Bspline approximation of circle arc and straight line for pocket machining*, Computer-Aided Design, September, 42/9, 817-827.
[9] VAVRUŠKA P., 2013, *Technological extensions of postprocessors for multi-axis machine tools*, Praha, A thesis submitted in partial fulfillment of the requirements for the degree of doctor of philosophy, CTU in Prague, Faculty of Mechanical Engineering.
[10] WANG F.C., YANG D.C.H., 1993, *Nearly arc-length parametrized quintic-spline interpolation for precision machining*, Computer-Aided Design, May, 25/5, 281-288.
[11] https://support.industry.siemens.com
[12] http://www.cgtech.com/
[13] http://www.ncsimul.com/
[14] http://www.worknc.com/