EFFECTS OF INTERPOLATION TYPE ON THE FEED-RATE CHARACTERISTIC OF MACHINING ON A REAL CNC MACHINE TOOL

Petr Vavruška*

The article is focused on the choice of interpolation types for toolpaths in NC programs. Interpolation type affects the interpretation of an NC program by the control system used in a CNC machine tool. It is important for a production engineer to know about the consequences of the use of various interpolation types in a specific CAM system for toolpath creation. For testing purposes, a general profile of a blade typically utilized in energy devices was used, with toolpaths for contour milling of its profile based on three interpolation types available in the CATIA CAM system (linear int., linear + circular int. and spline interpolation). The comparison of toolpaths is based on the number of blocks of the NC program, feed-rate profile measurement and measurement of machining time. For the machining and measurements, a real three-axis machine tool with the Sinumerik 840D control system has been used.

Keywords: *toolpath, interpolation, feed-rate*

1. Introduction

Machining of complex parts is associated with a number of technological difficulties. These are for example the influence of residual stress during the milling of thin-walled parts, uneven distribution of residual material along the machined surface before finishing operations, vibration of the tool and workpiece during machining etc. One of the key factors in this respect is appropriate preparation of NC programs for the production of specific components, today associated with the use of a specific CAM (CAD/CAM) system and a specific postprocessor. Preparation of toolpaths in the CAM system is dependent on programmed machining strategies which are different for various CAM systems. Some CAM systems are in fact based on the same computational core, but their technology functions need not be based on identical settings. It is clear that if the computational cores of two different CAM systems are not identical, the calculated toolpath points differ as well; it is impossible to determine this factor by comparing the generated toolpaths on the part model.

For these reasons, the article focuses on a comparison of interpolation types for toolpath creation in an NC program. The type of interpolation will affect the interpretation of the NC program by a control system. Selection of interpolation type is done by a technologist in the CAM system before the NC program is generated. Undoubtedly, the most commonly used type of interpolation is linear interpolation, which can be generated by any CAM system with small demands on the postprocessor (in case of three-axis machining) and can be interpreted in every control system. Another option is a combination of linear and circular interpolation.

* Ing. P. Vavruška, Department of Production Machines and Equipment, Faculty of Mechanical Engineering, Czech Technical University in Prague, Horská 3, 128 00 Prague 2

The postprocessor must be able to analyze input data (Cutter Location data), process data containing a circular interpolation and interpolation plane and then correctly transform this data into the NC program. Some systems offer the option of using a non-standard type of interpolation called a spline interpolation. For machining on a CNC machine, it is advisable for a production engineers to have some knowledge of the implications of a given type of interpolation used in a particular CAM system. Therefore, this article shows a comparison of the three types of interpolation for contour milling of a profile. Again, the postprocessor must be able to analyze these blocks with spline interpolation of input data and put them among other data respecting the NC program syntax required by the control system.

Based on information from practice, experts come in contact with spline interpolations only rarely; many production engineers are still relying on a combination of linear and circular interpolations. There is often very little time for order processing, meaning it is impossible to try new approaches. For this reason, spline interpolations are used mainly at the academic level, where only their mathematical apparatus is usually analyzed, e.g. in [1], [2] and [3].

The most often presented advantage of spline interpolation is that the toolpath should lead to an elimination of jumps in feed-rate, guaranteeing the fluidity of the cutting tool's movement. The other advantage is that NC programs may contain fewer blocks. Article [4] describes the results of a comparison between linear interpolation and spline interpolation, but only for 5-axis machining focused on surface quality, not on feed-rate profile along the toolpath. For this reason, it was decided to measure the feed-rate profile along the toolpath using a contactless sensor. For the purposes of test toolpath creation, a profile of a turbine blade was chosen; the toolpath was then created in the CATIA CAM system, as it allows the use of linear interpolations, combinations of linear and circular interpolations and spline interpolations (NURBS). For the interpretation of NC programs, the Sinumerik 840D control system was used on an LM1 experimental machine tool.

The article also shows the measured characteristics of feed-rate between the tool and workpiece using different values of toolpath computation tolerance, including observations on the machining time.

2. Testing part

Toolpaths in CAM systems are created interactively with surfaces or curves on a 3D model of the workpiece. For the purposes of applying toolpaths in CAM systems for the analysis of our NC programs, it was necessary to select a suitable 3D model which would represent a real shape of a workpiece from the industry and allow the creation of several toolpaths based on the same shape but with different attributes (toolpath computation tolerance value, different types of interpolation). NC programs were created on the basis of three different interpolations: 1) spline interpolation, 2) linear interpolation, and 3) a combination of linear

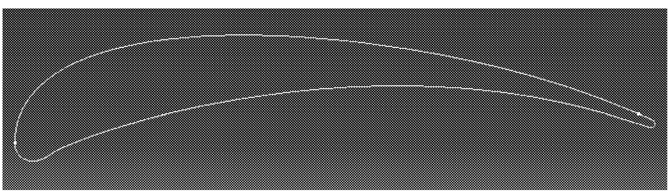

Fig.1: Profile of the blade used for testing

and circular interpolations. On the basis of each interpolation type, toolpaths were created with seven different toolpath computation tolerance values.

The curve in Fig. 1 has been created as a cut of a blade that meets the requirements mentioned above. It is a profile of a real blade used in practice. The blade profile is composed of general curves, so it is not an entity described analytically. Therefore, this profile is more than suitable for our purposes; due to its size, it is also very suitable for the subsequent creation of a test part consisting of a matrix (columns and rows) of such profiles in accordance with the requirements.

Fig. 2 shows the finished 3D model including the matrix of blades used for toolpath creation. The testing part was created by copying the profile for machining on a real CNC milling machine tool, taking into account a mill with a diameter of 8 mm.

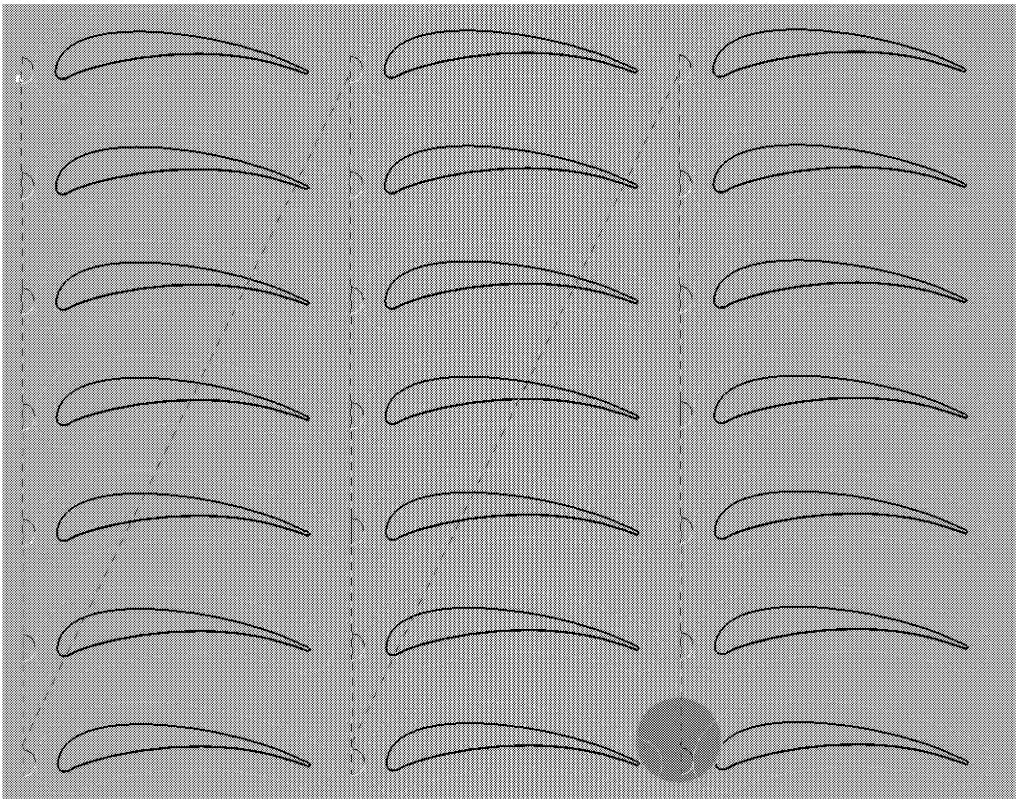

Fig.2: Finishing toolpaths in CAM system

In the CAM system, the following range of toolpath computation tolerance values was selected: 0.002 mm, 0.004 mm, 0.008 mm, 0.016 mm, 0.02 mm, 0.04 mm and 0.08 mm. The function called 'Profile Contouring' suitable for the creation of standard three-axis operations, has been used in this case as well. Subsequently, in the 'Part Operation' section, subsection 'Machine' and 'Numerical Control' only the following options can be selected: '3D linear Interpol.', '2D circular Interpol.', '3D circular Interpol.' and '3D NURBS interpolation'. This option can be used to set the format of the toolpath's description in the output file from a CAM system, i.e. the CL data. Operations for finishing toolpaths are shown in Fig. 2.

3. Analysis of NC programs

After toolpath creation in the CAM system, the toolpaths in the form of CL data were translated by the postprocessor into corresponding NC programs to be used on a real machine tool and its control system. NC programs have been compared for the number of blocks with interpolation type, providing more information about the length and size of the text file with NC programs and with it the memory requirements for their storage. It is necessary to add that for all NC programs, we also calculated the number of interpolations required for approach and retract from the cut, but not for the arrival at the desired level where the tool will be milling (only movements in the Z axis). Tab. 1 summarizes the number of interpolations in partial NC programs under specified toolpath computation tolerance values in the CAM system. With high demands for tolerance, the lowest number of generated interpolations occurs when spline interpolation is used; a combination of linear and circular interpolations generates a relatively small number of interpolations in NC programs as well. The advantage of circular interpolation against linear interpolation is a longer tool motion described by one interpolation. In case of linear interpolations only, the NC program generates a large number of points the tool must pass through; NC programs with linear interpolations are therefore most comprehensive. The number of interpolations in NC programs however is not critical for the quality of the NC program; it is important to analyze these NC programs in terms of feed-rate characteristics achieved along the toolpath.

number of NC blocks [-]	**toolpath computation tolerance value [mm]**						
	0.002	0.004	0.008	0.016	0.02	0.04	0.08
linear interpolation	393	283	199	140	129	91	65
linear and circular interpolation	276 (209+67)	185 (132+53)	119 (76+43)	69 (29+40)	61 (27+34)	39 (8+31)	29 (4+25)
spline interpolation	163	124	92	72	73	60	49

Tab.1: Number of interpolations in NC programs

Fig. 3 shows toolpath differences in NC programs that are based on different interpolation types. The left column contains toolpaths formed using spline interpolation. In the middle column, there are toolpaths containing only linear interpolations and the right column consists of tool paths conceived as a combination of linear and circular interpolations. The top row shows toolpaths calculated with the highest emphasis on the value of toolpath tolerance. The following rows (downward) contain toolpaths computed with a gradually reduced toolpath tolerance value.

It can be seen that in some sections of the toolpaths, the points are very densely concentrated. Examples of these sections are marked in the image. Toolpaths with such densely concentrated points may cause fluctuations in feed-rate; significant changes of feed-rate along the toolpath could also lead to large shocks during machining on the machine tool. This could cause vibrations of the tool, damage to the workpiece, failure to comply with prescribed surface quality or damage to machine tool components. In contrast, when using toolpaths with uniformly arranged points, it could be expected that the characteristic of the feed-rate during the execution of NC programs on the CNC machine tool will be smooth. Decreasing toolpath tolerance leads to a reduction of the number of points densely concentrated on toolpath sections and the numer of points of the whole toolpath as well. In the case of toolpaths based on spline interpolations, the control polygon points are listed in the NC program and the control system computes an approximation curve. For tool paths with linear interpolation, the NC program directly lists toolpath points which the control system connects by lines, and when a combination of linear and circular interpolation is used, the NC program also directly records the toolpath points that are to be connected by the control system using the specified interpolation, i.e. linear or circular.

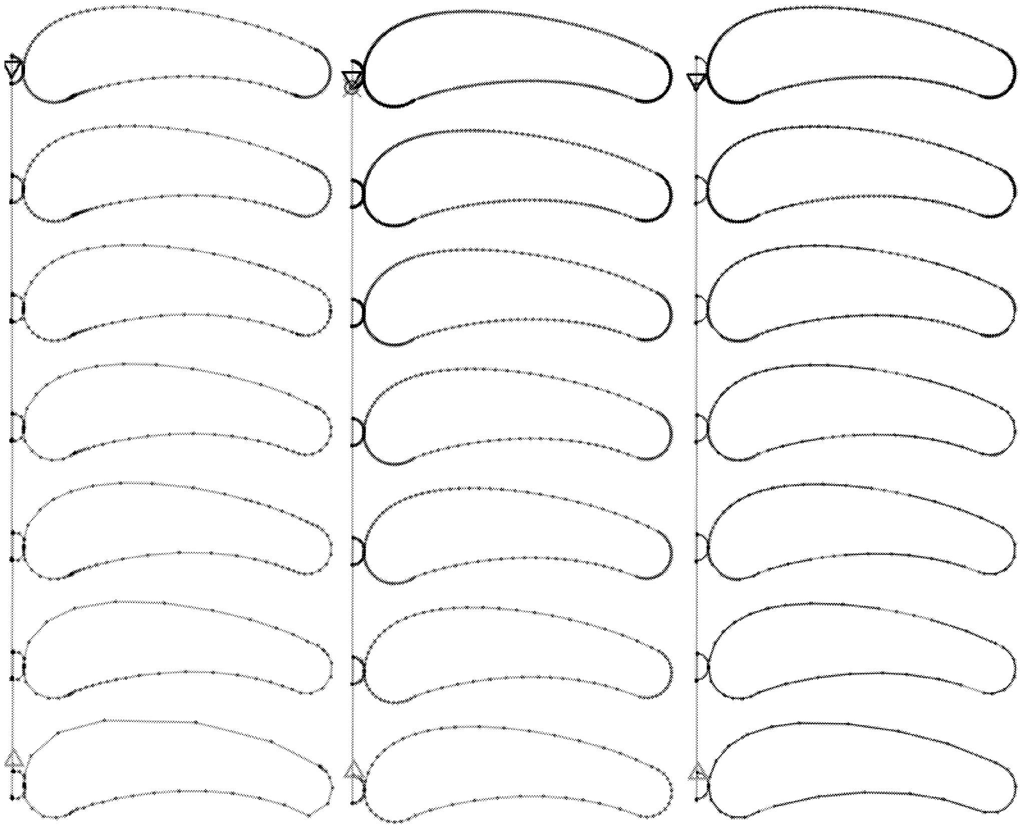

Fig.3: Comparison of NC programs based on different interpolation types (left – spline interpolation, middle – linear interpolation, right – combination of linear and circular interpolation)

The following two figures show a direct comparison of toolpath points of NC programs with linear and spline interpolation. In Fig. 4, it can be clearly seen that with a very high requirement for toolpath tolerance, control polygon points are almost identical to the points of linear interpolations. In contrast, Fig. 5 shows that in the case of rough toolpath tolerance, control polygon points are far from the points of linear interpolations. Nevertheless, the resulting surface machined using spline interpolations is smoother than surface machined using linear interpolations, as shown at the end of the article.

4. Feed-rate measurement

The feed-rate of a tool reference point relative to machined surface must be measured using a contactless method – a laser sensor for distance measurement, such as the one used

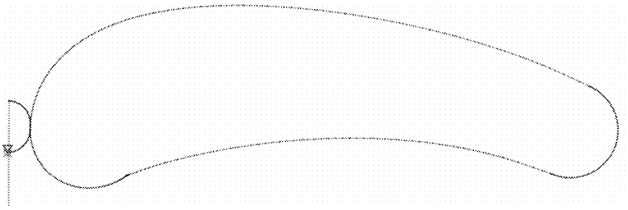

Fig.4: Comparison of points of NC programs based on linear and spline interpolations (for toolpath computation tolerance value of 0.002 mm)

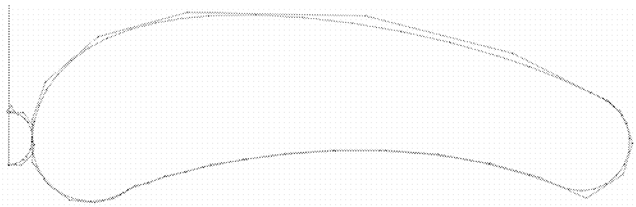

Fig.5: Comparison of points of NC programs based on linear and spline interpolations (for toolpath computation tolerance value of 0.08 mm)

in optical or laser computer mice. It is however essential to ensure the sensor's data are read in a periodically repeating time loop. For the purposes of feed-rate measurement, the ADNS A9500 sensor by Agilent has been tested. This type of sensor offers point sensing settings of resolutions from 200 cpi (counts per inch) up to 5700 cpi.

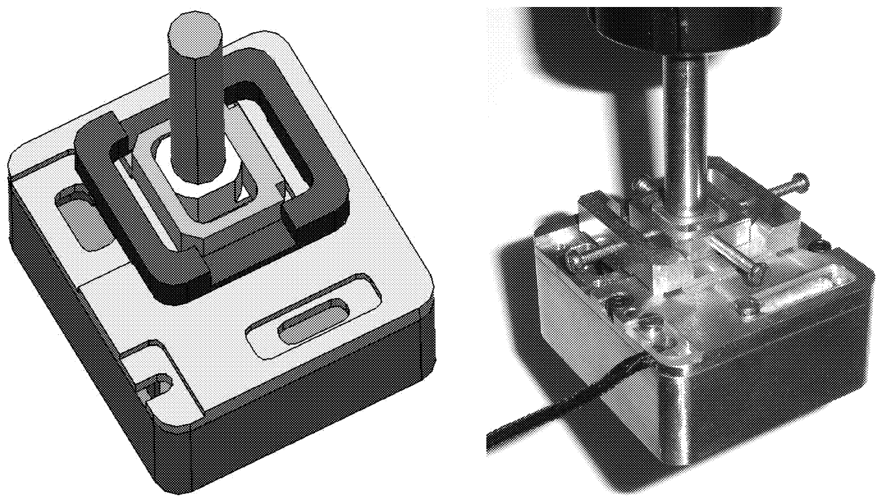

Fig.6: Device for feed-rate measurement (left – CAD model, right – finished device)

Fig. 6 shows on the left a proposal of a fixture for the sensor's mounting, and the eventual clamping of this assembly into a tool holder which secures the whole set in a machine tool spindle. At first it is necessary to center the sensor relative to the axis of a spindle, which is done by two adjustable perpendicular parts of the fixture; a hole in the bottom of the fixture serves as a loop holding the sensor itself. In Fig. 6 on the right, the whole assembly is clamped in a tool holder; centering screws aligning the sensor with thea spindle axis can be seen here, as well as a communications cable. It is important that the working gap between the sensor and surface is properly set during the measurement; this can be easily ensured using gauge blocks and values of gaps listed in the manufacturer documentation of the particular sensor.

Fig. 7 shows the experimental machine tool LM1 (in the workshop of the Department of Production Machines and Equipment, CTU in Prague, Faculty of Mechanical Engineering), with the measuring sensor clamped in the spindle. The sensor is connected to a laptop computer recording the measured values. The machine tool is equipped with direct linear drives in all three linear axes, meaning the machine is able to achieve high dynamics during the machining process, which is very desirable especially in HSC technology (High speed cutting). When using HSC technology, a small cross section of a chip is milled at high feed-rates. The principle of this technology was used in this case of blade profile cutting as well, with the value of the feed-rate at 4000 mm/min.

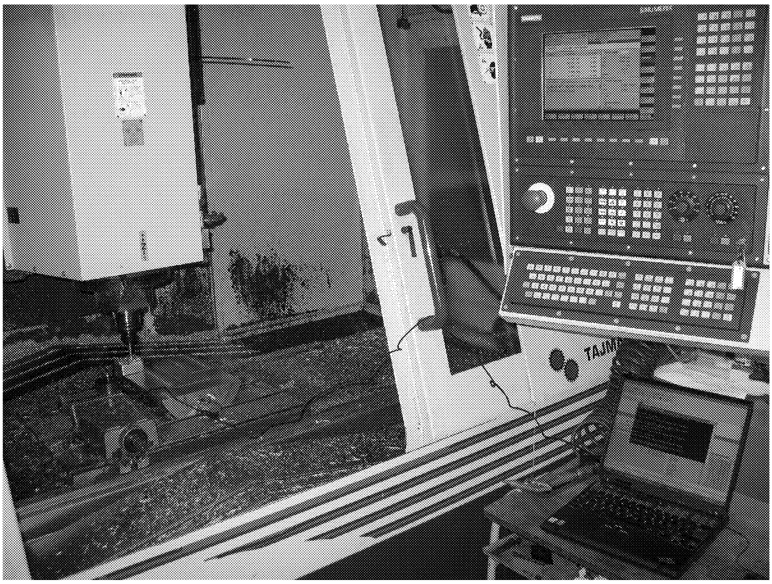

Fig.7: Experimental machine tool with direct linear drives and the Sinumerik 840D control system

Fig. 8 shows a sensor in its fixture and its position relative to the test surface during the feed-rate measurement. The picture gives an illustration of the working gap size settings (1.6 mm from the surface of the bottom fixture to the surface of the test workpiece). The figure also shows a white sheet of paper that has been glued to the test workpiece to increase the quality of the surface for measurement, because the workpiece was made of aluminum alloy the surface of which is glossy and reflective.

Now we turn to the measured characteristics of feed-rate along the toolpath in NC programs. Due to the length constraints of this article, not all measurements for all values of tolerance can be discussed here, even though all of them have been taken. Measurements for three representative values of tolerance will be shown for all types of interpolation used in NC programs, meaning 9 measurements will be shown in total. It is necessary to add that not all measurements have the same course of beginning over time. This is because the

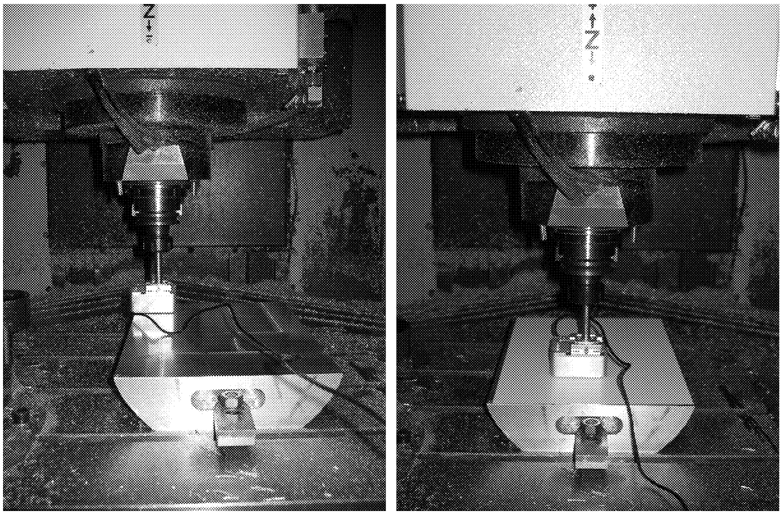

Fig.8: Feed-rate measurement (left – uncovered surface, right – surface covered with white paper)

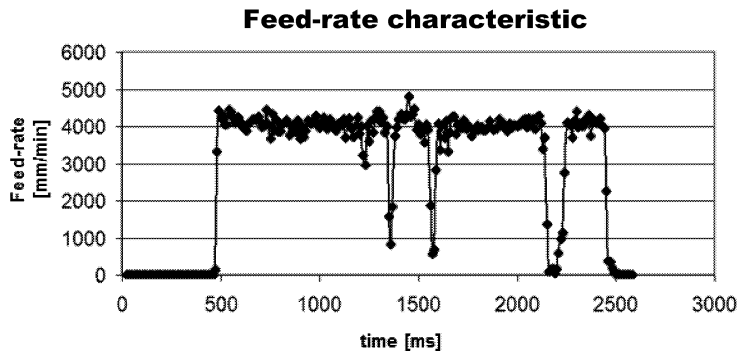

Fig.9: Feed-rate characteristic along a toolpath based on spline interpolations, toolpath computation tolerance value of 0.002 mm and nominal feed-rate value of 4000 mm/min

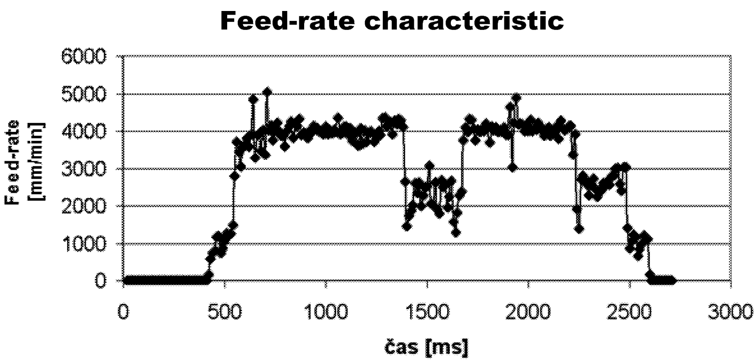

Fig.10: Feed-rate characteristic along a toolpath based on linear interpolations, toolpath computation tolerance value of 0.002 mm and nominal feed-rate value of 4000 mm/min

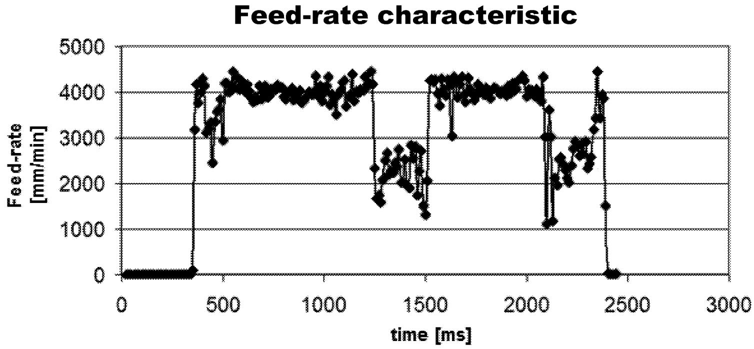

Fig.11: Feed-rate characteristic along a toolpath based on combinations of linear and circular interpolations, toolpath computation tolerance value of 0.002 mm and nominal feed-rate value of 4000 mm/min

feed-rate measurement was triggered manually and it was not always possible to provide the same moment in time to start the measurement. This fact, however, has zero impact on the analysis of the feed rate.

The previous three figures show the difference of the characteristics of measured feed-rates when using different interpolations in NC programs for the same toolpath computation tolerance value (0.002 mm in this case). Fig. 9 illustrates the feed-rate characteristic along a toolpath based on spline interpolations with three drops in feed-rate. In the case of a NC program with toolpath based on linear interpolations (Fig. 10), there are significant declines in feed-rate in longer sections of the toolpath, similarly as in the case of a toolpath based on combinations of linear and circular interpolations. It is clear that a constant feed-rate along the toolpath was not achieved in any of the cases. This can result in a negative influence on the machining and on the behavior of the machine tool. When machining on the machine tool, surges may occur, damaging the surface as well as the mechanics of the machine tool.

The three figures Fig. 13, Fig. 14 and Fig. 15 contain the characteristics of measured feed-rates along toolpaths in NC programs with toolpath computation tolerance of 0.016 mm. In the cases of characteristics of measured feed-rate based on NC programs with linear interpolations (Fig. 13) and combinations of linear and circular interpolations (Fig. 14), it can be concluded that a constant feed-rate has been achieved along the entire toolpath.

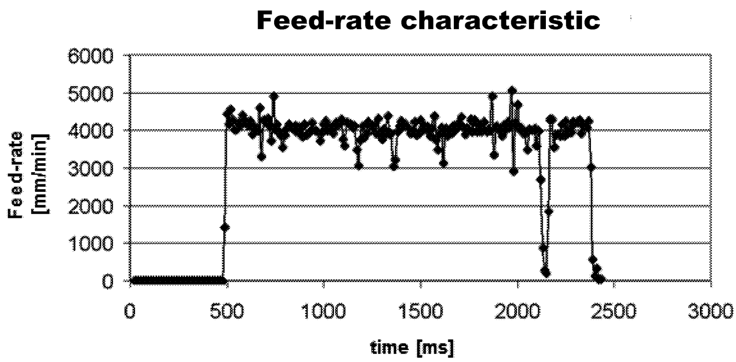

Fig.12: Feed-rate characteristic along a toolpath based on spline interpolations, toolpath computation tolerance value of 0.016 mm and nominal feed-rate value of 4000 mm/min

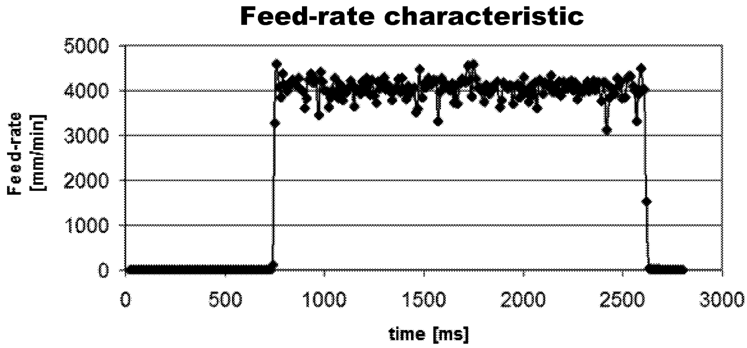

Fig.13: Feed-rate characteristic along a toolpath based on linear interpolations, toolpath computation tolerance value of 0.016 mm and nominal feed-rate value of 4000 mm/min

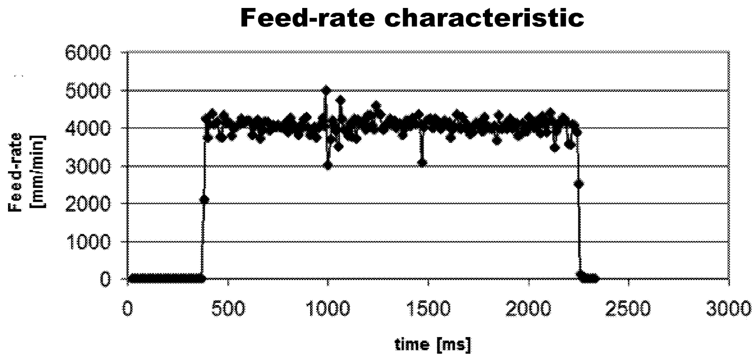

Fig.14: Feed-rate characteristic along a toolpath based on combinations of linear and circular interpolations, toolpath computation tolerance value of 0.016 mm and nominal feed-rate value of 4000 mm/min

In the case of feed-rate measured along a tool path described using spline interpolations (Fig. 12), there is still one occasion of a drop in the feed-rate. If a suitable algorithm for the correction of toolpath with spline interpolations were found, such occurrences of feed-rate decreases could be avoided, achieving a constant feed-rate in this case as well.

The three figures Fig. 15, Fig. 16 and Fig. 17 show the characteristics of measured feed-rates along toolpaths of NC programs with toolpath computation tolerance of 0.08 mm. Here it can be clearly seen that in all of the measured feed-rate characteristics, almost constant feed-rates have been achieved along the toolpath. These measurements show the influence of selection of toolpath computation tolerance value and of the type of interpolation on the feed-rate characteristic along the toolpath. Tab. 2 shows the machining times of all NC programs used for testing. The lowest machining time has been achieved using a combination of linear and circular interpolation, but if it were possible to use an algorithm that would eliminate the occurrences of feed-rate decreases using spline interpolations, then the shortest time and a constant feed-rate along the toolpath would be achieved in this case, which would certainly positively contribute to the quality of the workpiece surface. With rough toolpath computation tolerance values, machining times are almost identical; at this point, however, quality of the machined surface must be taken into account as well.

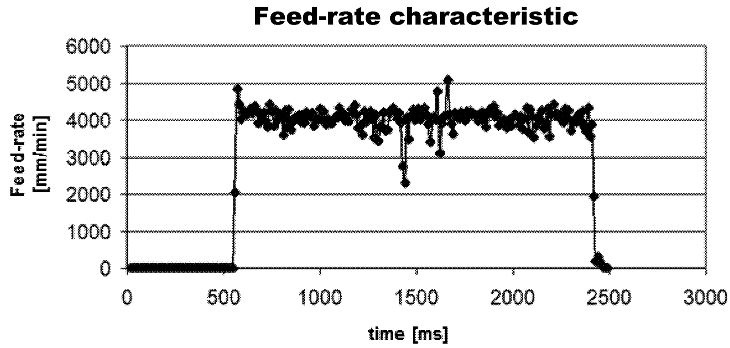

Fig.15: Feed-rate characteristic along a toolpath based on spline interpolations, toolpath computation tolerance value of 0.08 mm and nominal feed-rate value of 4000 mm/min

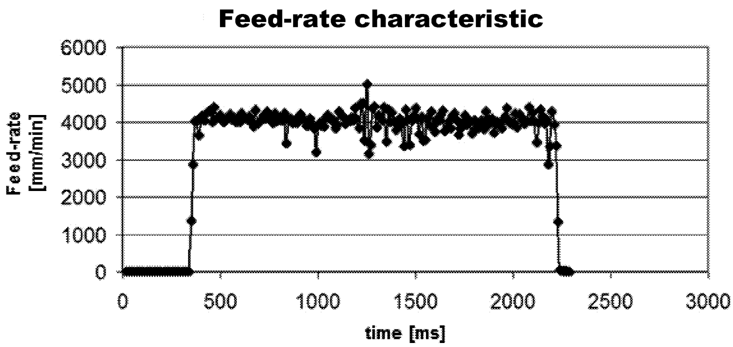

Fig.16: Feed-rate characteristic along a toolpath based on linear interpolations, toolpath computation tolerance value of 0.08 mm and nominal feed-rate value of 4000 mm/min

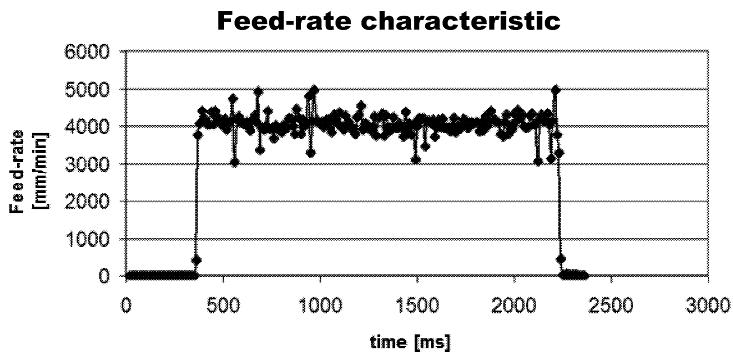

Fig.17: Feed-rate characteristic along a toolpath based on combinations of linear and circular interpolations, toolpath computation tolerance value of 0.08 mm and nominal feed-rate value of 4000 mm/min

5. Machining of the test part

The following detailed picture of blade surfaces (Fig. 18) very clearly shows the difference in the quality of blade surfaces after the roughing operation using a roughing mill (serrated mill with facet – chamfer of teeth corners). Completed test parts after finishing operations

machining time [ms]	toolpath computation tolerance value [mm]						
	0.002	0.004	0.008	0.016	0.02	0.04	0.08
spline interpolation	2060	2030	1980	1970	1960	1940	1940
linear interpolation	2200	2070	2000	1960	1930	1940	1940
linear and circular interpolation	1980	1940	1900	1920	1930	1900	1910

Tab.2: Machining time of different NC programs

of the CAM system for NC programs with different interpolations and different toolpath computation tolerance values are shown in Fig. 19. In this picture, the surfaces of blades after machining can be clearly seen; allowing a comparison of blade surfaces on the machined test parts.

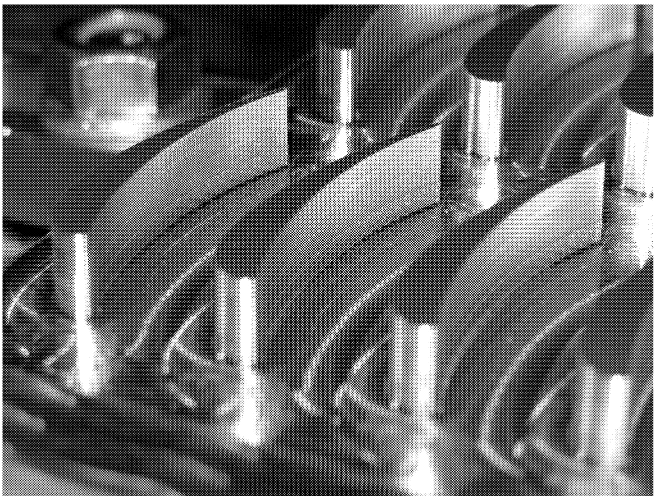

Fig. 18: View of rough milled blades (before finishing operations)

The last figure of this article (Fig. 20) shows in detail the quality of blade surfaces machined using NC programs with spline interpolations (left) and linear interpolations (right), both for the highest tolerance values. It can be clearly seen that the NC program which uses spline interpolations has achieved a very smooth surface even in with small demands for toolpath accuracy; a quite different result was achieved using linear interpolations, with a very angular surface of the blade. The best results have been achieved using spline interpolations and the combinations of linear and circular interpolations. In these cases, very good surface quality was possible at low machining times. To achieve a constant feed-rate along the toolpath with very high demands on surface quality, the use of spline interpolations seems appropriate; there is, however, still room for further improvement in a better description of the toolpath.

6. Conclusion

This article presents a detailed comparison of programming methods used in the creation of NC programs using linear interpolations, spline interpolations and combinations of linear

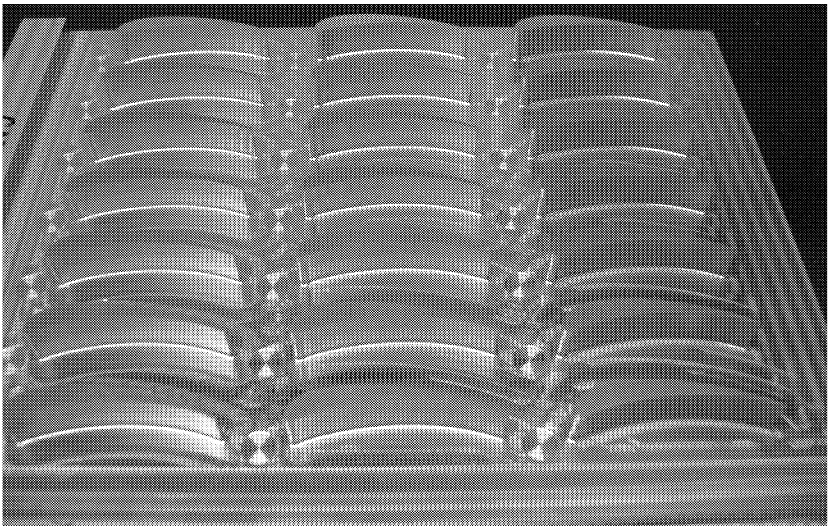

Fig. 19: View of finished blades milled using NC programs based on different interpolation types and different toolpath computation tolerance values

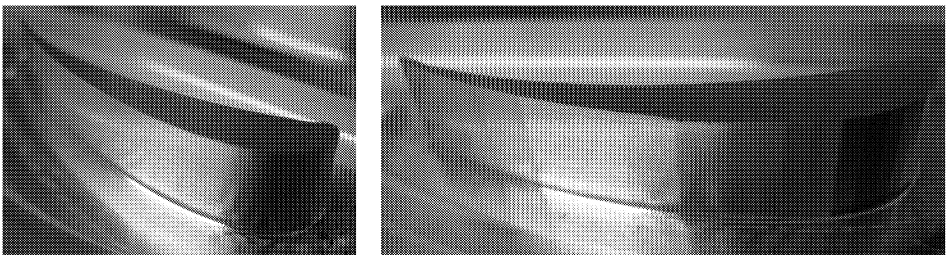

Fig.20: Close-up view of finished blades milled using NC programs based on different interpolation types but with the same toolpath computation tolerance value of 0.08 mm (left – spline interpolations, right – linear interpolations)

and circular interpolations. A test part was designed including a matrix of blade profiles, and used to create a set of toolpaths in the CATIA CAM system. The corresponding postprocessor has been programmed to convert the CL data into NC programs using spline interpolation of the CATIA CAM system. With this postprocessor, a testing of NC programs has been carried out, focusing on the types of interpolation mentioned above and their mutual comparison. This analysis showed that some of the CAM toolpaths create errors in machining that may cause large fluctuations in feed-rate, meaning the tool can vibrate during machining. This may result in damage of the machined surface, negatively impacting the value of the part. Best results in terms of time demands and surface quality have been showed by NC programs based on a combination of linear and circular interpolation. Very good machining times and surface quality have been also achieved when using spline interpolation for all selected toolpath computation tolerance values; there is, however, still room for further improvement in a better description of the toolpath in some cases of using spline interpolations. It is now possible to compare these results with the outputs of another CAM system in another, similarly conceived article.

Acknowledgement

A part of the presented results was supported by Ministry of Education and Sport of Czech Republic, grant No. 1M0507 and a part was supported by the Grant Agency of the Czech Technical University in Prague, grant No. SGS10/262/OHK2/3T/12.

References

[1] Wang F.C., Yang D.C.H.: Nearly arc-length parametrized quintic-spline interpolation for precision machining, Computer-Aided Design, Vol. 25, Issue 5, 1993, p. 281–288
[2] Müller M., Erdös G., Xirouchakis P.: High accuracy spline interpolation for 5-axis machining, Computer-Aided Design, Vol. 36, Issue 13, 2004, p. 1379–1393
[3] Pateloup V., Duc E., Ray P.: Bspline approximation of circle arc and straight line for pocket machining, Computer-Aided Design, Vol. 42, Issue 9, 2010, p. 817–827
[4] Langeron J.M. et al.: In A new format for 5-axis tool path computation, using Bspline curves, Computer-Aided Design, Vol. 36, Issue 12, 2004, p. 1219–1229
[5] Vavruška P., Fornůsek T.: Programming Methods for Machining of Complex Shape Parts, Research report no. V-11-075, CTU in Prague, Research Center of Manufacturing Technology, 2011, p. 71 (in Czech)

Received in editor's office: March 28, 2012
Approved for publishing: November 21, 2012