SPLINE POLYNOMIAL TO DESCRIBE A FREE FORM TOOL PATH FOR HIGH SPEED MILLING

Adriano Fagali de Souza, adriano.fagali@sociesc.org.br
Instituto Superior Tupy – IST
Sociedade Educacional de Santa Catarina – SOCIESC
Rua Albano Schmidt, 3333 Joinville – SC 89.201-972

Abstract*. So far, the linear interpolation of small straight line segments has been successfully applied by the CAM – CNC systems to describe tool path for milling free form geometries. This approach represents a great simplicity to calculate the path (CAM) and to accomplish the NC program (CNC). Die and mould and aerospace industries are the mainly users of this technique. Due to the technological evolution of all components of these manufacturing chain, from cutting tools up to machine centers, the machining process has becoming faster and the linear interpolation methodology does not satisfy an efficient process any longer, specially for High Speed Milling (HSM) applications. It causes oscillation on the feed rate and affects negatively the whole machining process. Therefore, the current paper investigates the usage of a Spline polynomial to describe a free form tool path in order to replace the linear method. A mathematical description of a Spline is presented and High Speed Milling experiments were accomplished. The results show the machine keeps higher feed rate value with a NC program containing Spline codes rather than the linear counterpart. However, this approach still has a lack of development. Feed rate oscillations still happens and injuring the whole machining process. The influences of the tolerance for tool path calculation for machining process were also evaluated.*

Keywords: *Free form geometries; Spline Polynomials; NC program.*

1. INTRODUCTION

The evolution of the CAD/CAM systems and CNC machines has allowed free form shapes products to be easily manufactured. Today, 3D forms can be realistically modelled in a 3D-CAD system, the NC programs for these complex shapes can be generated in a commercial CAM system, and the parts can be machined in a CNC machining centre. Such parts, which usually are dies and moulds, represent a key position on the overall production chain, affecting costs, quality and lead-time of a product (Boujelbene *et al*, 2004). Lately, several limitations and drawbacks in die and mould manufacturing have been discussed. The application of the HSM technology in this manufacturing area is being taken as a driving force for this industry (Rigby, 1993). It is believed that the manual rework can be reduced quite substantially applying the HSC at the finishing milling operation. Timesaving on hand finishing can reach 80% and the costs savings concerning to this process can reach 30% (Schulz and Finzer, 1999).

According to Geist (1999), what is more important than increasing the cutting speed is increasing the feed rate [mm/min]. High frequency spindles in combination with high feed rate are a more precise characterization for HSM, especially in die and mould manufacturing. Several articles show many authors referencing feed rate to HSM as high as 20.000 mm/min. However, this paper shows that high feed rate values are not realistic for milling free form geometries, even when using appropriate high-speed machines with resources such as look-ahead function and CNC with a very low block processing time. Reductions and oscillations on the programmed feed rate can be observed during milling such kinds of geometry (Souza, 2001).

Normally, the real feed rate cannot even reach the value set in the NC program and varies widely along the machining path. These oscillations increase cutting time, the milling time estimated by the CAM becomes quite inaccurate and the feed per tooth varies, as well as the load on the cutting tool. Variations on feed per tooth affect the surface roughness and the load on the tool affects dimensions and surface texture on the workpiece. In many cases, for free form milling, the feed rate oscillates duo to CNC/Machine processing limitation, especially when the linear interpolation is applied, with continuity C^0 (Souza and Coelho, 2006). The usage of spline polynomials to represent a free form tool path is a promise to solve this limitation.

The current paper addresses this issue. First, it is done a mathematical description of a Spline polynomial. After, an investigation about feed rate oscillation is carried out. A representative workpiece was machined in high speed, using NC programs with the ordinary linear interpolation and programs with Spline codes. Both sort of NC programs were generated by a commercial CAM system. The influences of the tolerances required for CAM calculation were also investigated.

The surface's visual aspects and feed rate variation during machining were assessed. The milling experiments were accomplished on a commercial high-speed milling machine, using P20 steel, and also a commercial high-end CAD/CAM system was used. A 3D free form workpiece was properly designed for these experiments and a real-time monitoring system was developed to investigate the feed rate variation according to machine position during the milling operation.

2. FREE FORM GEOMETRY REPRESENTED BY CAD SOFTWARE

High-end CAD software uses complex mathematical functions, known as Splines, that allows the designer to model free form geometries. The Spline functions came about in the late XVII century, with the mathematician Charles Hermite, who used a polynomial equation to define a curve with two points, one at the start (P_0) and the other at the end of the curve (P_1) and also two tangent vectors associated with these points (P_0') (P_1') (Bates, 1997). Figure 1 illustrates the Hermite curve.

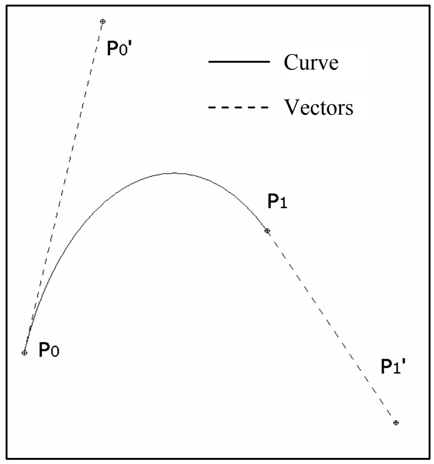


Figure 1: Hermite curve. Points and vectors

For example, using a third degree polynomial for two points, such as:

$$p = p(u) = k_3u^3 + k_2u^2 + k_1u + k_0 \quad (1)$$

The derivative of Eq. (1.1) is:

$$p' = p'(u) = 3k_3u^2 + 2k_2u + k_1 \quad (2)$$

Taking the values of ***u=[0-1]*** as a continuous interval and assuming the polynomial begins and ends within this dominium, let us calculate the values of ***p*** and ***p'*** at the extremes:

$$p(u=0) = p_0 = k_0 \quad (3)$$

$$p(u=1) = p_1 = k_3 + k_2 + k_1 + k_0 \quad (4)$$

$$p'(u=0) = p'_0 = k_1 \quad (5)$$

$$p'(u=1) = p'_1 = 3k_3 + 2k_2 + k_1 \quad (6)$$

Now there are four equations as a function of four parameters, k_i (i=1…4). Parameters k_0 and k_1 were directly found by Eq. (3) and (5), respectively, and if these results are substituted into Eq. (4):

$$p_1 = k_3 + k_2 + p'_0 + p_0 \quad (7)$$

Appling now Eq. (5) into Eq. (6):

$$p'_1 = 3k_3 + 2k_2 + p'_0 \quad (8)$$

Then solving Eq. (2.7) and (2.8) into the values of k_2 and k_3, the results are:

$$k_2 = 3(p_1 - p_0) - 2p'_0 - p'_1 \quad (9)$$

$$k_3 = 2(p_0 - p_1) + p'_0 + p'_1 \quad (10)$$

Now, substituting the obtained values for k_i (i=1…4) into the Eq. (1), the parametrical polynomial resultant is:

$$p(u) = p_0(1 - 3u^2 + 2u^3) + p_1(3u^2 - 2u^3) + p'_0(u - 2u^2 + u^3) + p'_1(-u^2 + u^3) \quad (11)$$

The tangential vector and the extreme points of the curve are helpful for modelling free-form geometries. However, the approach developed by Hermite does not represent a friendly interface to the ordinary CAD user, as it depends on the derivative of the equation. In order to overcome such inconvenience, one of the first computer representations of free form curves and surfaces was developed by Pierre Bézier, in 1972. Since then it has become the most popular method for curve design used in graphic packages and CAD system (Larent, 2001). He developed a methodology to represent these kinds of entities on a computer, in order to implement the software UNISURF at the Renault industry. Bézier used a polygon for controlling the curve in order to substitute the initial contour condition presented by Hermite. The degree of the Bézier polynomial equation is 1, less the number of the polygon control points.

The curve is plotted by a cubic polynomial. The points $\boldsymbol{p_0}$ and $\boldsymbol{p_3}$ of the Bézier curve are equivalents to the Hermite $\boldsymbol{p_0}$ and $\boldsymbol{p_1}$. Taking the Hermite parametrical polynomial, Bézier defined the derivative at the initial and final points as 1/3 of the difference between the consecutive points, of the new polygon, i.e.:

$$p'_0 = 3(p_1 - p_0) \tag{12}$$

and

$$p'_1 = 3(p_3 - p_2) \tag{13}$$

Substituting $\boldsymbol{p'_0}$ and $\boldsymbol{p'_1}$ into Eq (11), it results into Bézier formulation:

$$p(u) = p_0(1-3u+3u^2-u^3)+p_1(3u-3u^2+u^3)+p_2(3u^2-3u^3)+p_3(-3u^2+3u^3) \tag{14}$$

Using the excel software, the Bézier formulation was implemented and a curve was plotted according to the control point's coordinates, as illustrated in Figure 2.

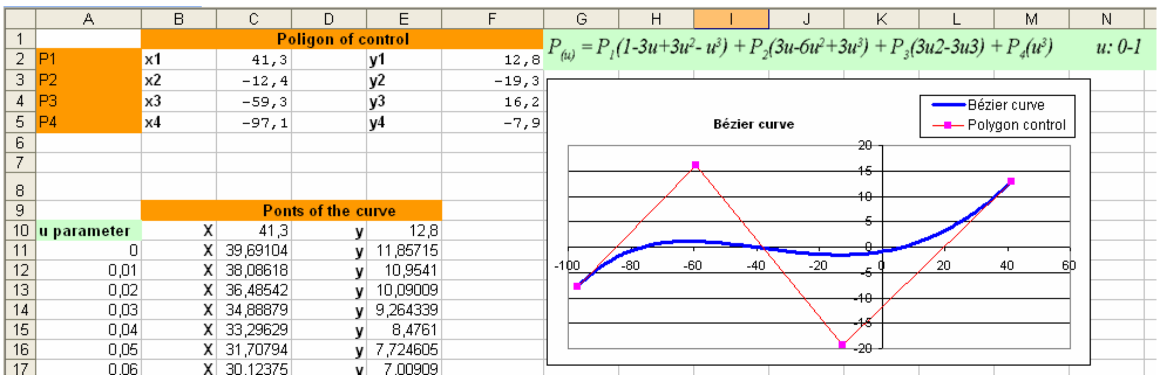


Figure 2: Bézier curve plotted in graphic software

The polynomial is now a function only of the values at the controlling polygon excluding its derivatives. Therefore, any desired free form curve or surface is driven by a polygon formed by control points. The Bézier method has two main inconveniences: the polynomial degree depends on the number of control points, which can compromise the computation task and secondly, only global curve alterations can be done. Any modification can affect the complete curve. Other Spline polynomials have been developed to supply the limitation from Bézier curve. The Spline state of art is known as NURBS – Non-uniform rational B-spline, which has many advantages, compared to the Bézier ones, including the computational simplicity and more flexibility to geometrical representation (Mahon and Browne, 1993).

3. CNC PROGRAM USING SPLINE POLYNOMIAL

The idea to use a parametric polynomial to represent curves remains very useful for computing applications. For many years, Spline has been used in computer graphic packages, especially for CAD application. Much effort is now being spent on using Spline polynomial to describe free form tool path.

To generate the tool path, the CAM program first calculates the cutter contact path (CC) over the 3D CAD geometry. Then, the cutter location path (CL), which represents the tool path is calculated by offsetting the CC (Lo, 2000). The most traditional method to describe a free-form tool path is the interpolation of straight-line segments, with continuity C0. This is easily transformed into the ordinary G01 code, according to DIN 66025 (Dürr and Schünemman, 1999). The CAM adjusts the tool path segments inside a tolerance band defined by the user, known as "chord error". The smaller the tolerance band, the closer the tool path will be to the CAD model. Thus, linear segments are generated by such procedure and, for highly curved paths, the number of segments increases drastically, as well as the NC program size. Several inconveniences can be identified when applying this method for milling free form geometries, especially when high feed rates are required.

Alternatively, circular, associated with linear interpolation, can be used to describe a free form path. This combination can keep curve continuity C^1 and has shown many advantages over the line segments alone (Qui *et al*, 1997). This method, however, can only be applied for two dimensional simultaneous movements, since the vast majority of CAD/CAM and CNC are not capable of making free form circular interpolation in three dimensions.

Another approach to describe free form tool paths is investigated in this paper: the use of a polynomial Spline (Fig. 3*a*). Rather then CAD modeling, at this step Spline is applied to represent a tool path. Therefore, the CAM system has to generate NC programs containing the polynomial data and only the information about the curve goes to the CNC. The machine then transforms the polynomial data into speed for the axis movements. For NC programs based on Spline codes generated by the CAM, it requires a new syntax to NC program, which involves descriptions of the polynomial equation. Therefore, the CAM system has to be able to generate such NC codes and the CNC has to be able to execute them. On the current market there is no standard syntax for the Spline's NC code. Each CNC brand has its own syntax to understand this alternative approach of programming, among those able to run Spline programs. Therefore, the CAM has to post-process the Spline NC codes according to a specific CNC. Fig. 3*b* shows the NC Spline program developed according to the Siemens 840D CNC syntax.

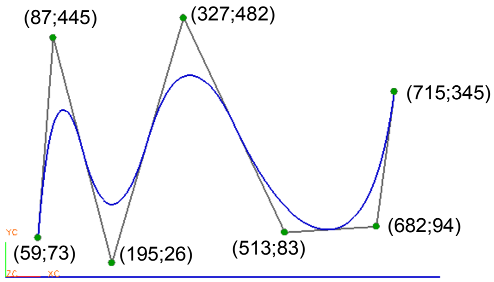


Fig. (a): The Spline curve and its control points

```
POLY
PO[X]=(59,0,0)PO[Y]=(73,0,0)PO[Z]=(0,0,0)
PO[X]=(87,0,0)PO[Y]=(445,0,0)PO[Z]=(0,0,0)
PO[X]=(195,0,0) PO[Y]=(26 ,0,0)
PO[Z]=(0,0,0)
PO[X]=(327,0,0)PO[Y]=(482,0,0)PO[Z]=(0,0,0
)
PO[X]=(513,0,0)PO[Y]=(83,0,0)PO[Z]=(0,0,0)
PO[X]=(682,0,0)PO[Y]=(94,0,0)PO[Z]=(0,0,0)
PO[X]=(715,0,0)PO[Y]=(345,0,0)PO[Z]=(0,0,0
```

Fig. (b): The respective NC program according to Siemens syntax for Spline

Figure 3: Spline NC program according to Siemens syntax

Several algorithms for Spline calculation were developed. Koninckx and Brussel (2002) presented a real-time interpolator, using a third-order keeping C^2 continuous NURBS. Farouki *et al* (1999), suggest a CNC interpolator using Pythagorean-hodograph curves. Tikhon *et al* (2004) proposed a NURBS algorithm for varying the feed rates in order to keep the cutting tool load constant. Lartigue *et al* (2001) presented a method to generate a tool path in terms of planar cubic B-spline curves, for a smooth free-form surface. Yau and Kuo (2001) developed an algorithm to convert an ordinary NC program containing G01 codes into a polynomial NC code. The last method has been implemented in some up-to-date CNCs. The controller is able to make this conversion on line, during machining. This function creates possibilities of higher feed rate values when milling free form geometries, even when a NC program containing the linear interpolation is in use. However, this approach requires another tolerance band to generate the polynomial program, inside the CNC, losing the accuracy to the original CAD surface, and additionally, it represents one more task to the CNC process, compromising the CNC processing time. It is suggested that the best approach would be if the CNC receives the Spline program already calculated by the CAM system. In this manner, the tolerances are well known and the CNC saves the computational task of converting the G01 codes into Spline codes. The previously mentioned Spline algorithms are still not available on the market.

Today, a high performance CNC can have a block processing time (BPT) less then 1 millisecond. Despite that a bottleneck can still occur when high speed processing is required. Trying to support this technological limitation, new CNC functions are being developed by the CNC suppliers. It is called Look-ahead algorithms, and it has the function to pre-process a number of blocks ahead of time, trying to leave processed blocks ready for use. This function was also taken into account on this investigation.

4. EXPERIMENTAL WORK

The experiments presented in this paper concerns about the feed rate oscillation for high speed milling of free form geometries. According to Lo (2000); Monreal and Rodrigues (2003); Souza (2004), the feed rate oscillates due to the liner interpolation of straight lines segments, ordinarily used to describe a free form tool path. The current investigation aims to furnish a better view about this process limitation, in terms of real-time machining oscillation and its consequence on the surface quality. It is also presented the application of Spline tool path and its influences on the machining process, compared to the linear counterpart. In addition, were also analyzed the the tolerances for tool path calculation, and its influences for feed oscillation.

The influences of cutting parameters, material machineability, and chip formation are not considered in this experimental. This task is a case of study experiment. The cutting parameters as well as the row material were chosen close to the real application, and it was kept constant. Therefore, in this sort of experiment doesn't fit an statistic evaluation.

To investigate the feed rate oscillation using both tool path methods, the experiments were split into:

i) Experiment 1: Feed oscillation for linear and Spline NC programs;

ii) Experiment 2: Influences of the tool path tolerance calculation for machining process.

The Unigraphics NX 2 CAD/CAM system was used to model the workpiece and to generate the Linear and Splines NC programs. The milling operations were developed in a high speed machining center Hermle C800, controlled by the CNC Siemens 840D. This CNC is one of the few capable of interpolating Spline polynomials. The roughness was obtened by a Taylor-Hobson surftest model Surtronic 3P and an optical microscope Zeiss Axiotech, equipped with a digital camera AxioCam MRc. A data acquisition system was developed to obtained from the CNC the feed rate and tool position in a real time presented in preview works (Ferraz *et al,* 2005).

4.1 Experiment 1: Feed oscillation for linear and Spline NC programs

Figure 4a illustrates the representative work piece designed for this experiment. The workpiece was roughed to leave a constant amount of material of 0,2 mm for finishing operation. The finishing milling was accomplished by a 6 mm of diameter, two flutes ball nose cutting tool with; 0,2 mm as step over; feed rate of 4.500 mm/min; and spindle frequency of 21.000 rpm. The finishing tool path is shown in Figure 4b, calculated under a tolerance band of 0,005 mm.

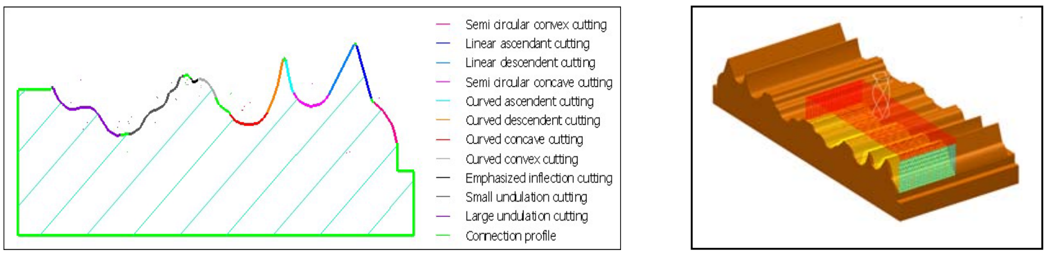


Fig. a: Workpiece *Fig b.: Tool path*

Figure 4: workpiece developed and tool path for finishing

4.1.1 Feed Rate

This experiment will use the data acquisition system, connected to CNC, to get the feed rate and tool position in real time during milling. It is comparing the NC program using the ordinary straight line interpolation method for milling a free form workpiece against the Spline codes generated by the CAM system. The actual feed speed at any tool position along the milling is shown in Figure 5, for both tool paths strategies.

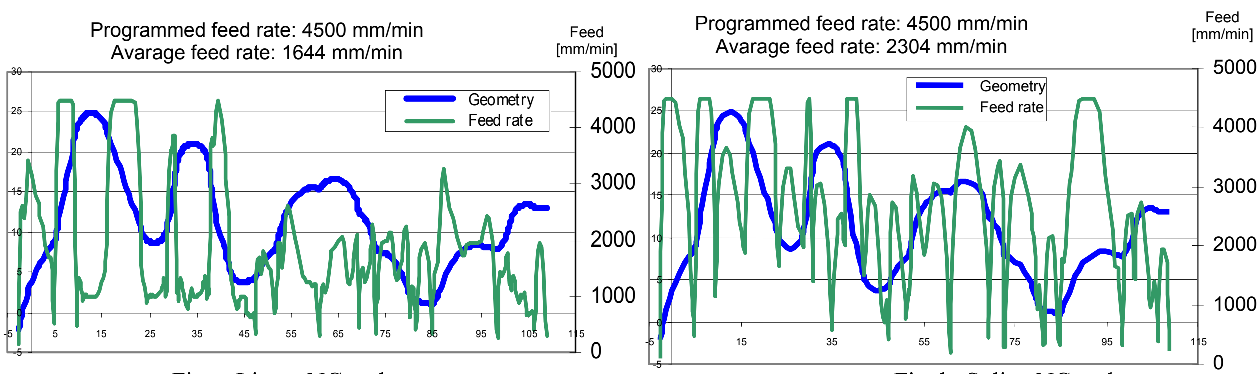


Fig a: Linear NC codes Fig. b: Spline NC codes

Figure 5: The tool path and the feed rate plotted

It can be noted that feed speed for linear NC codes is lower than its counterparts beased on spline polynomials, being 1.644 mm/min and 2.304 mm/min on average, respectively. The proportion, around 1.4 time higher, represents a significant gain, in terms of machining time. However, even using Spline polynomial interpolation, the programmed feed speed was reached just a few times, for short periods of time and strong “vibrations” could still be observed during machining time.

Looking at the NC program length, the one using Spline method resulted shorter than that with linear segments but, it still had a relatively high number of instructions. Each line of the Spline NC program still represents a relatively short tool movement. The algorithms used by the CAM system to calculate the tool paths, based on Spline codes, are still not sufficient to transfer information to CNC machine controllers for segments long enough to avoid conflicts with time of block execution of the CNC machine.

Additionally, in order to verify this issue, a simple tool path was generated by hand, based on a Spline equation, using seven defined control points. The Figure 3 shows the tool path and the Spline NC program, which contains just seven lines. Doing the appropriate procedure, “the same” Spline tool path was obtained by the CAM software, using 0,005 mm as tolerance band. The respective NC program in this time resulted in 232 lines, to describe the “same path”. It is suggested the amount of information generated by the CAM with the spline codes still be a great amount of information for the CNC process in time.

4.1.2 Surface quality

After milling the workpiece, the Ra parameter was obtained. Due to the difficult to get this data from a complex surface, the Ra parameter represents only qualitative information. Photos were taken in specific areas to show the surfaces aspects. Figure 6 shows the Ra medium value of 5 samples and the respective surface texture. Two areas of the workpiece were analyzed. These areas were chosen due to well represent the study. Area I shows the surface quality when feed rate oscillate, by the linear interpolation. In the same area, the workpiece milled by the Spline tool path kept the feed constant. It is noticeable the difference on the surface aspect. The area II shows an injured surface aspect for both tool path methods. In this region, the feed rate had severe oscillations for both cases. The photos show the surface texture of several parallel tool paths. Considering this point, an statistic analysis was omitted.

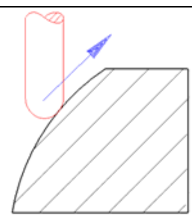
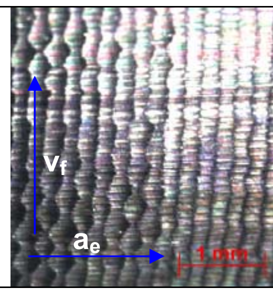
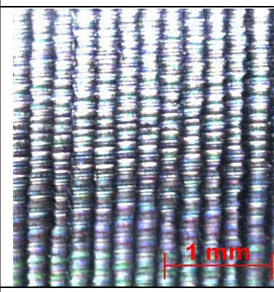
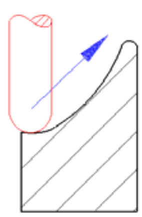
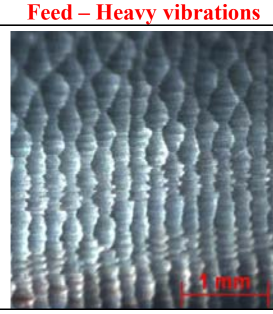
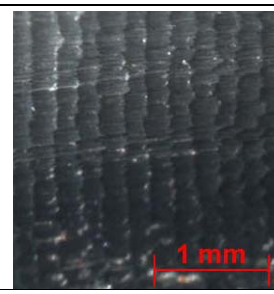
Area	Linear Interpolation	*Spline*
		
Area I	**R_a =0,970 Feed – Heavy vibrations**	**R_a =0,852 Feed – Constant**
		
Area II	**R_a =1,268 Feed – Heavy vibrations**	**R_a =1,008 Feed – Heavy vibrations**

Figure 6: Surface finishing according to the feed rate oscillation

It can be noticed that there are some surface damage due to “vibrations” on specific areas of the workpiece. It occurs due to the feed rate oscillation, and thus, occasioning constant alterations on the cutting tool load. At the Area I, the feed speed varied when the linear path was carried-out, and kept constant when Spline was used. In the Spline case, surface finishing was mainly influenced by cutting parameters, feed per tooth and steep over, unlike in the linear interpolation one, which had a bad surface texture, due to the constant cutting tool load alternation. The Area II represents a region on the workpiece where both strategies produced high variations on feed speed. Therefore, a very poor surface quality was observed. Although the Spline NC program was faster than the linear one, it still presented “vibrations”.

4.2 Experiment 2: Influences of the tool path tolerance calculation for machining process

The calculation process of a free form tool path using the linear interpolation method under a tolerance band (chord error) is well known. Now, the influences of the tolerance band on the NC program size and machining timing are investigated for Spline and linear tool paths.

First, the tolerances applied for Spline tool path calculation must be take into account. In this method, besides the tolerance band used for calculating the linear tool path, another tolerance is required by the CAM software. It is named Fitting Control. The software algorithm uses the linear tool path to calculate the Spline one. Figure 7 represents a sketch of this procedure (Souza and Coelho, 2007).

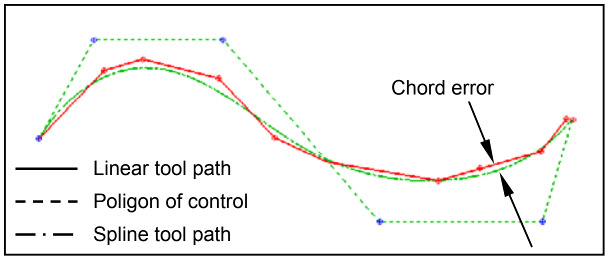


Figure 7: Tolerances required for calculating Spline tool paths.

This procedure represents another tolerance in the machining process, when a Spline tool path is applied.

4.2.1 NC program size according to tool path method and tolerance

This analysis shows the influence of the tolerance value on the NC program size, concerning the method applied to describe a tool path. For Spline path, the same value was set as tolerance band and fitting control in order to ensure the same maximum path deviation for both methods. Table 1 shows the influence of the tolerances and the tool path method on the NC program size.

Table 1: Influences of the tolerances on the NC program size. Linear and Spline tool paths.

Maximum tolerance	Program size [number of NC blocks]	
	Linear	**Spline**
0,005 mm	798	324
0,01 mm	330	222
0,1 mm	232	93

The tolerance influences directly the length of the tool path segment, either for linear or Spline path.
Comparing the tool path method, reduction on the NC programs can be noticed. The Spline programs are smaller than the linear ones, for all the tolerance values evaluated. But, even for the highest tolerance value, the Spline NC program is still huge. It represents an amount of information for CNC processing, and it can compromise the feed rate.

4.2.2 Feed rate oscillation according to tool path method and tolerance

The NC programs evaluated above were run in the milling machine. The average feed rate was obtained by the real time monitoring system. Table 2 shows the differences on the average feed rate according to the tolerances and the tool path methods.

Table 2: Average feed rate considering tolerance and tool path method

Maximum tolerance	Feed rate average [mm/min]	
	Linear	**Spline**
0,005 mm	1.644	2.304
0,01 mm	2.476	2.604
0,1 mm	2.893	3.124

The Figure 8 shows the tolerance and the tool path method play an important role on the machining feed rate. It propitiates different feed rate oscillations. First, considering the tolerance values, the feed rate average greatly differs. The smaller the tolerance band, the higher the feed oscillation, and therefore, higher the machining timing. It can happen due to the increasing of the number of NC blocks in the NC programs. The feed rate oscillates severely according to the segment length of the tool path which is related to the tolerance value, for both methods. Now analyzing the tool path methods, the Spline program was faster then the linear one, for all the tolerance values analyzed. But, the most significant difference can be observed when the smallest tolerance value was applied. In this case, the Spline program was about 40% faster than the linear one. However, for the others tolerance values, the Spline program not did run as fast as expected. This could be attributed to the algorithm used to calculate the paths, which still calculates a great amount of Spline segments in a non uniform manner. Figure 8 present the relation among tool path method, tolerance value and feed rate.

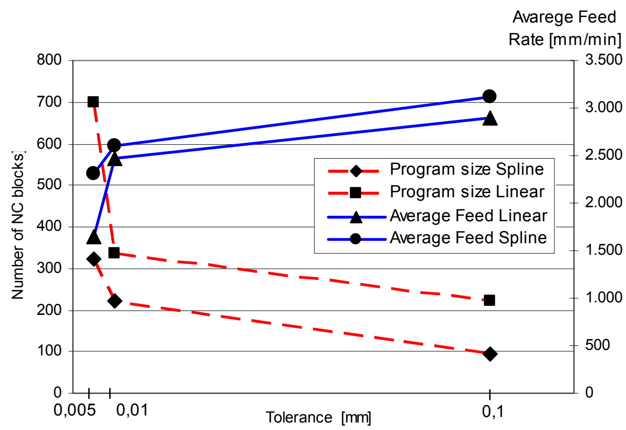


Figure 8: Correlation among feed rate, path method, and tolerance value.

Increasing 2 times the lowest tolerance value represents a reduction about 58% on the NC program size for linear method and its average feed rate increases about 50%. Considering the Spline method, for the same case of tolerance, the NC program size reduces 31% and increases 13% the feed rate value.

When the tolerance was increased 20 times the lowest value, the NC program size reduces about 70%, and the average feed rate increases 76% for the linear tool path. For the Spline method, the NC program size reduced about 71%, increasing about 35% the average feed rate.

Although the linear method obtained higher percentage value on the average feed rate with the increasing of the tolerances value, it is important to notice the feed rate was always higher for the Spline method, according to the tolerance value: 40%; 5%; 7%, respectively from the lower tolerance value.

Due to non linear proportion, a feasible mathematical relation for the NC program size, tool path method and feed rate oscillation was not possible to develop. This relation is not linear and also influenced by the cutting tool geometry and dimensions, surface curvature and especially for the CAM algorithm, which is difficult to predict.

5. CONCLUSION

Having in mind the limitation of the experiments accomplished, where only one representative geometry was analyzed; the machining parameters and row material were kept constant, following industrial application; and an appropriate design of experiments is missed, even that, the conclusions presented in this paper can contribute with this science and technology development.

First conclusion is feed rate cutting parameter represents the bottleneck for applying the high speed milling technology for dies and moulds fabrication, specially those with free form geometries. The surface quality is injured by heavy oscillations on the programmed feed, it increases the milling time, varies the cutting parameter (as feed per tooth), and the cutting tool load becomes inconstant; which affects directly the machined surface texture.

The feed rate oscillates severely when free form geometries are milled in high feed rate, for both methods analyzed to describe a tool path, the linear interpolation and the polynomial Spline. The machine reaches the programmed feed rate very few times, and its oscillations affect the milled surface quality and increase the milling timing.

A significant reduction of the Spline NC program size was expected compared to the linear. However, the small reduction of the NC program size and a not significant reduction of the machining timing by the Spline NC program were remarkable.

The tolerance values to calculate a NC program and the methods applied to describe a free form tool path play an important role for machining process, in terms of machining timing and the surface quality. Higher feed rate values were obtained under a higher tolerance band. It can represent productivity gain; however the user should well understand the geometrical error consequences. For the Spline tool path, the software required two tolerances values. Therefore, the users should well understand its effects in order to have an optimum machining process.

The NC program size is directly related to the tolerance values, required to generate the NC program by the CAM software and the feed rate oscillation is related to the program size. However, this relationship between program size and feed oscillation is not linear. It is more related how well uniformly the NC points are distributed along the trajectory, the larger the segment, the higher and constant the feed rate is. It is also influenced by the cutting tool geometry, surface curvature and especially by the CAM algorithm.

Using the Spline strategy, better results were achieved. The Spline NC program presented higher average feed rate, about 40%, under a tolerance band of 0,005 mm, however by altering the tolerance value, this benefit can be drastically reduced, in a non uniform way. The CAM algorithm employed to calculate the Spline tool path divided the trajectory into a large number of small Spline segments. It still represented a significant amount of information to CNC process.

This work concludes that there is still a lack of development in order to improve the high speed cutting technology for milling free form shapes, considering the three manly technological anchors of this manufacturing process: (i) CAM software; (ii) milling machine; (iii) CNC.

6. ACKNOWLEDGEMENTS

The authors thank Sociedade Educacional Santa Catarina (SOCIESC), Sandvik Coromant, for supplying the cutting tools used in the experiments; Unigraphics Solutions Brazil, for helping the development of the post-processor used for Spline NC codes; SIEMENS and HERMLE Brazil, for all technical support; Villares Metals for supplying the rough material; the Foundation FAPESP for supporting the HSC Thematic Project under the grant number 00/02962-4; and also the projects, Instituto Fábrica do Milênio (IFM II).

7. REFERENCES

Boujelbene M., Moisan A., Tounsi. N. and Brenier B., 2004, “Productivity enhancement in dies and molds manufacturing by the use of C1 continuous tool path”. I nt. Jour of Mach T & Man. v.44, n.1, pp.101-107.

Rigby P., 1993, “High Speed Milling in the Mold and Die Making Industries”. Inn: Diamond and CBN Ultrahard Materials Symposium, Ontario, Canada.

Schulz H. and Finzer Th., 1999, “Optimization of the Cutting Process Using HSC in Die and Mold Manufacturing”. Inn: 5th International Scientific Conference on Production Engineering - CIM'99. Zagreb

Geist J., 1999, “Influence of HSC-appropriate machining parameters on NC programming”. Inn: Seminário Internacional de Alta Tecnologia – Santa Bárbara, Brazil. pp.57-68.

Souza, A. F., 2001, “Análise das interpolações de trajetórias de ferramenta na usinagem HSC (high speed cutting) em superfícies complexas”. 89p. Dissertação (Mestrado) - Universidade Metodista de Piracicaba, Piracicaba.

Souza, A. F. and Coelho R. T., 2007, “Experimental Investigation of Feed Rate Limitations on High Speed Milling Aimed at Industrial Applications”. Int. Jou. Adv. Manufacturing Technology. DOI 10.1007/s00170-006-0445-2.

Bates C., 1997, “Some well-defined curve”. Am. Mach. v. 141, n.11, pp. 51-54.

Larent, P. and Sablonniere P., 2001, “Pierre Bézier: An engineer and a mathematician”, Comp Aid Geom Des. v. 18 pp. 609-617.

Mahon, Mc. and Browne J., 1993, “CAD/CAM From Principles to Practice. Suffolk”. UK. Addison-Esley. ISBN 0-210-56502-1

Lo, C-C., 2000, “CNC machine tool surface interpolator for ball-end milling of free form surfaces”. I Jour of Mach T & Man, v.40, p.307-326

Dürr, H. and Schünemman R., 1999, "Industrial Aplication of new Approaches of the CAD/CAM process chain for High Speed Machining of Sculptured Surfaces". Inn: International German and French Conference. 2nd. Darmstadt: PTW, pp. 117-120.

Qui, H., Cheng K. and Li, Y., 1997, “Optimal circular arc interpolation for NC tool path generation in curve contour manufacturing”. Comp. Aid. Des. v. 29, pp. 751-760

Yau, H. T., Kuo M. J., 2001, “Nurbs machining and feed rate adjustment for high-speed cutting of complex sculptured surfaces”. Int. Jou. of Production Research, London. v.39, n.1, p.21-41

Koninckx, B. and Brussel, H.V., 2002, “Real-Time NURBS Interpolator for Distributed Motion Control”. Inn: CIRP, v.51 (1) (2002) 315-318

Farouki, R.T, Manjunathaiah, J and Yan, G. F., 1999, “G Codes for the specification of pythagorean- hodograph tool paths and associated feed rate functions on open-architecture CNC machines”. I Jour of Mach T & Man. v.39, n.1, p.123-142

Tikhon, M., Ko T.J, Lee, S.H, Kim,H.S, 2004, “NURBS interpolator for constant material removal rate in open NC machine tools”. I Jour of Mach T & Man. v. 44, pp. 237-245

Lartigue, C., Thiebaut F., Maekawa, T., 2001, “CNC tool path in terms of B-Spline curves”. Comp. Aid. Des. London, v.33, n.4, p.307-319

Ferraz, F., Oliveira J.F.G., Souza, A.F., 2005, “Supervision and Monitoring Technologies in Machine Tools With Open Architecture CNC”. Inn: Congresso Brasileiro de Engenharia de Fabricação. Joinville Brazil.

Souza, A.F., 2004, “Contribuições ao fresamento de geometrias complexas aplicando a tecnologia de usinagem com altas velocidades”. Tese de doutorado. Universidade de São Paulo. São Carlos, SP.

Monreal, M, Rodrigues, C.A, 2003, "Influence of tool path strategy on the cycle time of high-speed milling". Comp Aid Des. v.35, pp. 395-401

8. RESPONSIBILITY NOTICE

The author(s) is (are) the only responsible for the printed material included in this paper.