Manual programming of milling operations with variable orientation of tool axis in SINUMERIK 840D sl numerical control system

Programowanie operacji frezowania ze zmienną orientacją osi narzędzia w układzie sterowania SINUMERIK 840D sl

MAREK KROK *

DOI: https://doi.org/10.17814/mechanik.2017.5-6.56

Described is the way of programming milling operations with variable tool axis in relation to a machined surface. Presented are functions of numerical control SINUMERIK 840D sl produced by Siemens. This article presents basic functions related to the kinematics of the machine tool and the purpose of its application. The methods of programming the orientation of a tool in the working space of the machine are described. For better illustration of the presented issues, examples are also shown.

KEYWORDS: machine tool programming, variable tool axis programming, machine tool kinematic

The main purpose of machining with variable tool axis orientation is to achieve high geometric-shape accuracy of the workpiece to ensure high productivity. Increasing demands of constructors in many fields of industry - aiming among others. For the best performance in terms of liquid flow, ergonomics of use, or even aesthetic reasons, the resulting surfaces become more complicated and less tolerant. The digital model of a machined workpiece usually comes from a CAD system, while an operation program is generated in a CAM system based on geometric and technological calculations. The process of generating an NC program involving adaptation of intermediate data to a specific CNC machine tool is often marginalized due to the limitations of the postprocessor by the technologist. This translates into the NC code form defined by the postprocessor developer, which is not always optimal in the given case.

Surfaces to which variable axial workpiece machining is applied can be divided into three main groups:

- free-form surfaces - most often found in the production of injection molds,
- aerodynamic surfaces - found in parts of aircraft engines such as blades or rotors,
- rectangular surfaces - present in structural parts in the form of ribs or other reinforcements.

Each of these groups requires different configuration of machine control functions. Good knowledge of these functions is crucial to properly prepare a technology program - whether it is generated automatically (in CAD/CAM) or manually created by the developer.

Compensation for kinematics

In order to facilitate the work of the programmer and machine operator, the manufacturers of numerical control systems introduced dynamic kinematics compensation of the machine for simultaneous four or five axes (six axes for special applications such as laminate machine tools). Five-axis compensation is most commonly used.

In the SINUMERIK 840D control system, the kinematics compensation function activates the TRAORI function. Its task is to shift the linear support of the machine so that when the tool axis orientation changes by the rotation of the machine tool axis, the end of the tool remains in constant geometric relation with the coordinate system of the workpiece (fig. 1). The compensation function separates the compensation movement associated with changing the orientation of the tool axis from the programmable path of the workpiece contour tool, eliminating the need to introduce kinematics compensation in the post-processing step [1].

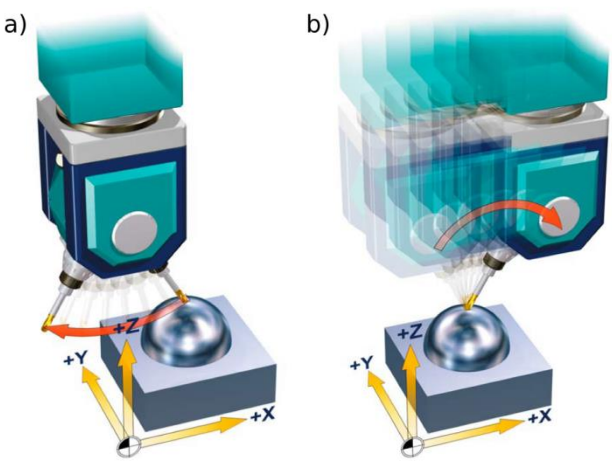

Fig. 1. Changing the tool orientation: a) without activating the TRAORI function, b) activating the TRAORI axis [1] requires the implementation of kinematics compensation at the post-processing stage [1].

The five-axis machine tool kinematics consists of three linear axes: X, Y, Z, working in the right-hand Cartesian system, and two additional rotary axes, which can be configured in different configurations, depending on the purpose of the machine. For the purpose of kinematics compensation, there are three types of five-axis kinematics:

- "P" - rotary axes are located on the side of the machine table and their rotary movement moves the workpiece in the working space,

* Mgr inż. Marek Krok (mkrok@prz.edu.pl) – Politechnika Rzeszowska

- "T" - rotary axes are implemented by the spindle, and their rotational motion causes a change of orientation of the tool in the machine space,
- "M" - mixed kinematics, where one axis causes the object to rotate, and the other - to change the orientation of the tool in the working space of the machine.

In order for kinematics compensation to work properly, the control system must include information about the type of kinematics and the values of the compensating vectors in the individual axes. These data are written in machine parameters of the machine tool [2].

Programming methods

In the SINUMERIK 840D sl control system, a kinematically dependent and independent tool axis tilt change programming method can be distinguished.

In the **machine-related orientation** approach, the technological program contains the actual angular values for the rotary axes (*A, B, C*) in the machine concerned. At the program generation stage by the vector postprocessor, the tool orientation is translated into direct rotation of the rotary axis. In this case, the program can only be run on a kinematics machine strictly defined in the postprocessor. The program only contains data related to the position of the tool point in the workpiece coordinate system (WCS) and the angular values of the rotary axes with respect to the machine coordinate system (MCS). The disadvantage of this method is the need to have a dedicated postprocessor that includes a kinematic transformation matrix that allows the intermediate data in vector form to be changed to the kinematically dependent absolute values of the rotary axes. The advantage of this solution is that it is possible to perform a more accurate simulation of the machine to verify the collision of the process using the inverse post-processor [1].

In the **workpiece-related orientation** approach, the technological program includes a vector-oriented tool orientation in the workpiece layout (WCS). The technology program can be launched on machines having different kinematics variants. No dedicated postprocessor is required to generate the program, and the tool orientation translation into the physical rotation of the rotary axis is performed by the control system. The kinematically independent program contains the coordinates of the tool point and the orientation of the tool axis vector in one of several methods. Typically, the orientation of a tool axis is expressed by unit vectors, virtual angles of rotation, or normal vector programming to the surface being machined. Since no direct values of rotation of the rotary axis are programmed, for a control system in the case of redundant machine tool kinematics, the tool orientation given may have several correct solutions; Collision analysis is not possible without the use of tools such as VNCK (Virtual NC Kernel) [1].

Programming the tool orientation in a kinematically independent system

The SINUMERIK 840D sl control system offers many possibilities for programming the tool orientation in a machine tool working space. It is impossible to present all here, thus we will discuss the three most popular ones.

The vector orientation of the tool in the SINUMERIK 840D control system is expressed by the *A3, B3, C3* keywords that describe the axial vectors of the axes, and the tool direction is projected on the *X, Y, Z* axes respectively. The vector direction of the tool axis corresponds to the straight line from the tool code point in the direction of the holder (fig. 2). The scale of the length of unit vectors is irrelevant and the direction is determined on the basis of their mutual proportions. Vector orientation programming is a non-modal instruction; words not programmed at default assume a value of 0. Vector orientation programming is a recommended tool orientation programming method, and the accuracy of linear values should be at least five decimal places in metric and a minimum of six decimal places for vector orientation programming. High accuracy tolerances for axis and vector values are due to errors in mathematical operations on floating-point numbers such as rounding errors, truncation, and numerical instability [3].

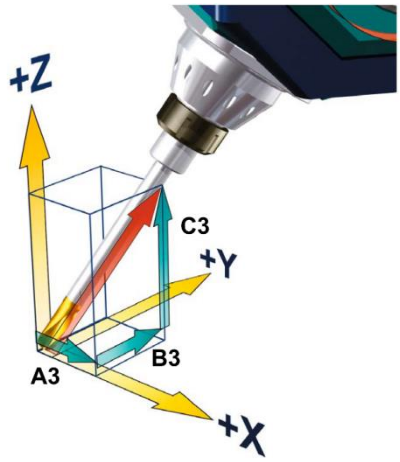

Fig. 2. Programming the tool orientation through unit vectors [1]

Another popular tool orientation programming method is based on the use of virtual turning angles by keywords *A2, B2, C2*. Depending on the setting of the preparatory functions in group 50, angles can be interpreted inter alia. as the angles of Euler ORIEULER or RPY (*roll pitch yaw*) ORIRPY. Virtual angles transform one Cartesian system to another, and the axis of the tool aligns with the *Z* axis of the end system. If Euler angles are used, the rotation of the system with respect to the *Z* axis is by the angle of the angle in *A2* expressed in degrees. The transformed system is then rotated according to the new vector *X* by the value *B2*. Next, the rotation of the system relative to the new *Z* axis by the value of the angle *C2*. In turn, in the case of programming the *RPY* angles, the rotations are performed along the *Z, Y', X''* axis with angles *C2, B2* and *A2*, respectively. Virtual angles are limited in CAM systems because of the need for special postprocessors. However, their advantages are felt in parametric programming where virtual angles are the result of mathematical relationships describing curvilinear surfaces and can be applied directly [3].

The third most common method is normal vector programming for the surface to be processed by the keywords *A4, B4, C4, A5, B5, C5*. Index words 4 describe the normal vector for the surface to be processed at the start point of the block, while index 5 describes the normal vector at its end. By default, the tool is set perpendicular to the machined surface during interpolation. In addition, using the LEAD and TILT commands, the pitch and tilt angle values can be programmed, where the forward angle is interpreted as the deviation of the tool axis in the feed direction, while the tilt angle is expressed in a plane perpendicular to the normal direction at the tool-to-work contact point. It is measured from the feed direction (fig. 3) [3].

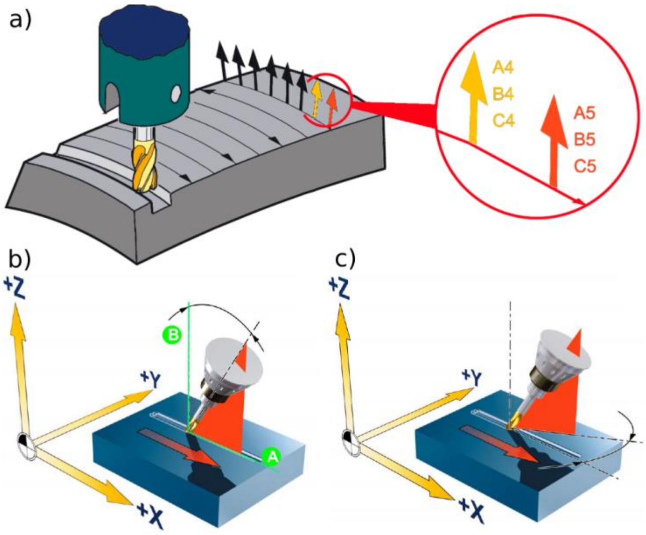

Fig. 3. Programming of the face machining (where: *A* - feed direction, *B* - normal direction for the tangent surface at the tangent point): *a)* tangent angles at the beginning and end of the block, *b)* leading advance angle, *c)* TILT angle [3]

Vector interpolation

The linear interpolation of the tool path uniquely identifies the position of the tool code point while moving to the next programmed position. If a tool orientation change is programmed at the next programmed block, a smooth transition from one orientation to the next is made. Changing the tool orientation with interpolation of the feature point can theoretically be done in any way; therefore, in order to be able to control the trajectory of the vector representing the tool axis, the SINUMERIK 840D sl control system provides commands from 51st group of preparatory functions. This group contains many types of interpolation, including interpolation on the periphery of the cone or with an additional guiding curve, but most commonly ORIAXES and ORIVECT are used.

ORIAXES - rotary axes are interpolated linearly synchronously to the tool path. In the case of this vector interpolation, the intermediate positions during interpolation depend strictly on the kinematics of the machine tool. This method is suitable for ball milling where the cylindrical part does not take part in the contour of the workpiece. Because of its speed, it is recommended for example in the treatment of dies and stamps [1].

ORIVECT - in this interpolation, the tool axis during the change remains in the plane formed by the tool vectors at the beginning and at the end of the block caught at one point. Changing the tool orientation is precise and independent of the machine tool kinematics, however, due to the need for additional rotational displacement of the rotary axis, it is slower. It is used in operations where the periphery or front of the cutter forms the contour of the workpiece - most often in the machining of the surface in the aviation industry [1].

Example

To better illustrate the described functions, an example of chamfering with variable tool axis orientation shown in fig. 4 is shown. The operation consists of roughing using contour processing cycles (not included) and machining two chamfers: yellow (No. 1), which will be machined by the cutter's side and blue (No. 2), which will be machined on the cutter face. The program uses 3D correction of the tool and information comments about the operation of the individual functions.

Fig. 5 shows the results of the machining simulation in SinuTrain version 4.7 edition 2.

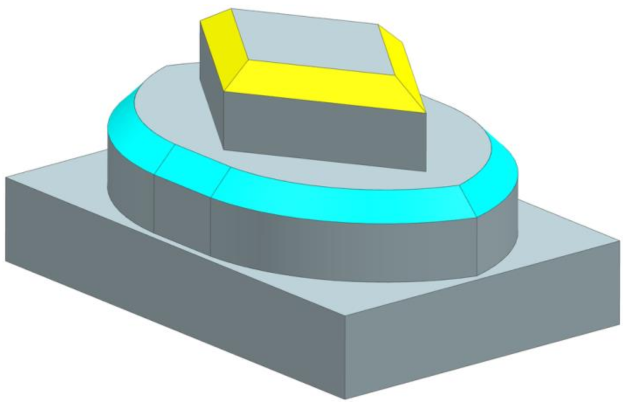

Fig. 4. Workpiece with marked phases

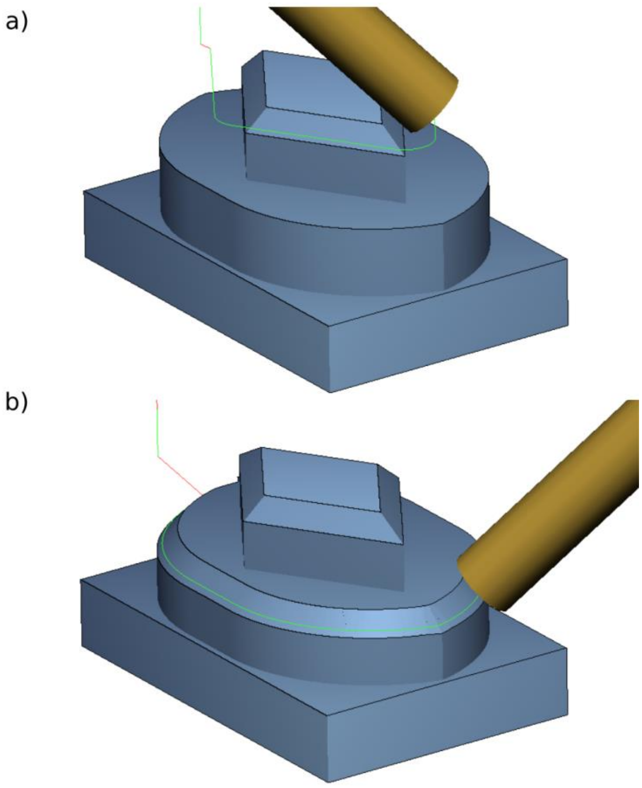

Fig. 5. Simulation result of phase machining phase 1 - machining of cutter's periphery, b) phase 2 - machining of cutter's face

```
N10 G0 G17 G40 G64 G90 G94 G54 ; Funkcje przygotowaw-
cze
... ; Obróbka zgrubna
N190 T="CUTTER 10" D1
N200 M6
N210 S6000 M3 F1000
; Faza nr. 1: obróbka pobocznicą freza
N220 G0 Z30
N230 G0 X30 Y0 ; Dojazd do punktu startu
N240 G0 Z-3
N250 CUT3DC ; Aktywacja korekcji 3D
N260 TRAORI ; Aktywacja kompensacji kinematyki
N270 ISD=1 ; Obniżenie narzędzia wzdłuż osi
N280 ORIVECT ; Aktywacja interpolacji osi narzędzia
N290 G17 G90
N300 R0=SQRT(20*20+13*13) ; Parametr pomocniczy
N310 G0 X20 Y0 A3=-13 B3=-20 C3=R0 G42
N320 G1 X0 Y13
N330 A3=13 B3=-20 C3=R0
N340 X-20 Y0
N350 A3=13 B3=20 C3=R0
N360 X0 Y-13
N370 A3=-13 B3=20 C3=R0
N380 X20 Y0
N390 X30 Y0 G40 C3=1


; Faza nr 2: Obróbka czołem freza
N400 G0 Z30
N410 G0 X40 Y0
N420 G0 Z-3
N430 G1 Z-10
N440 TRAORI ; Aktywacja kompensacji kinematyki
N450 ISD=2.1 ; Obniżenie narzędzia wzdłuż osi
N460 ORIVECT ; Aktywacja korekcji 3D
N470 G17 G90
N480 OFFN=-4 ; Odsunięcie od zaprogramowanego konturu
N490 R0=SQRT(24*24+5.658*5.658) ; Parametr pomocniczy
N500 G0 G42 X29 Y0 A3=1 B3=0 C3=1
N510 G1 A3=24 B3=5.658 C3=R0
N520 G3 X5 Y19 I=AC(5) J=AC(-5.658) A3=0 B3=1 C3=1
N530 G1 X-5 A3=0 B3=1 C3=1 ;
N540 G3 X-29 Y0 I=AC(-5) J=AC(-5.658) A3=-24 B3=5.658
C3=R0
N550 G1 A3=-24 B3=-5.658 C3=R0
N560 G3 X-5 Y-19 I=AC(-5) J=AC(5.658) A3=0 B3=-1 C3=1
N570 G1 X5 A3=0 B3=-1 C3=1
N580 G3 X29 Y0 I=AC(5) J=AC(5.658) A3=24 B3=-5.658
C3=R0
N590 G1 G40 X40 Y0C3=1
N600 G0 Z100
N610 TRAFOOF ; Dezaktywacja kompensacji kinematyki
N620 M30
```

REFERENCES

1. Milling with SINUMERIK Mold making with 3 to 5-axis simultaneous milling, 09/2013, 6FC5095-0AB10-0BP3.
2.SINUMERIK 840D sl / 828D Job Planning, 10/2015, 6FC5398-2BP40-5BA3.
3. 840D sl SINUMERIK Operate 5-Axis-Workshop Technology Milling, 01/2016. Siemens TAC.