System integrated product design, CNC programming and postprocessing for three-axis lathes

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Abstract

New three-axis CNC turning machines have been installed on shop floors of factories in the last few years. Although the machines are still particularly adapted to cylindrical work, they are also used for non-cylindrical machining. The third axis, C or Y, and separately driven, additional milling cutters have eliminated traditional limitations and opened new machining possibilities, i.e. turning and milling the non-rotational shapes. Even the advanced, integrated CAD/CAM systems do not support the milling and turning processes on three-axis CNC lathes or turning centers. Manual part programming for such machine tools, if available, does not facilitate the integration between design and manufacturing. For accurate part programming on three-axis lathes a new interpolation scheme must be applied, using both spiral segments of Archimedes and circular segments rather than linear segments as applied in CNC milling. Specialized, built-in software developed for the three-axis lathes enables to expand CAD/CAM integration to the same extent as in CNC multi-axis milling. The software includes modified postprocessors for three-axis lathes as well. The integrated processes of design and manufacturing are the most important prerequisites for effective concurrent engineering. In general, it means both better economics and quality of complex products, e.g. shaped rolls for rolling operations, cams and camshafts (non-ruled surfaces included), non-rotational shafts, airscrews, prostheses for total hip replacements, etc. © 2001 Elsevier Science B.V. All rights reserved.

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1. Introduction

A new generation of multi-axis CNC milling machines, CNC lathes, machining centers, turning centers, production centers and wire-EDM require new features in manufacturing modules of integrated CAD/CAM/CAE systems. Very often respective features of the machine tool design and characteristics enable a new approach to the machining as a whole, and allow to improve economics and quality of products machined. In order to introduce all arising improvements within integrated processes of design and manufacturing, the new specialized software must be developed and built-in to the CAD/CAM system. In the last years, innovations and new features in CNC machine tools and in the other CNC production equipment initiated a typical hi-tech manufacturing loop: a specific, distinctive application of new features; development of software supporting the application; development of a generalized software for a range of potential applications; expansion of the new manufacturing methods. A specialized software supporting new approach to integrated design and manufacturing processes, as in CAD/CAM systems, plays a vital role in concurrent engineering concept, particularly when virtual processes are concerned. Mostly this is the way of improving the economics of machining, the dimensional accuracy and lead time of a product.

The third axis added to the standard CNC lathes has opened the turning processes to a wide range of new applications. Actually it means milling processes on lathes as well, because milling cutters are used more often than cutting tools. There are optional machine tool features for milling processes: a special spindle drive option for a work piece rotation (C-axis), usually based on stepping motors and also independently driven milling cutters. When one wants to use cutting tools for three-axis turning, some limitations from traditional turning remain, i.e. shape limitations arising from the geometry of a cutting tool (particularly relief angles) and limitations in achieving the recommended cutting speed range due to a reduced range of spindle revolutions.

For effective turning and milling operations on three-axis lathes, both work piece shape modeling as well as cutter tool path (CL file) and its postprocessing should be accurately generated in the same CAD/CAM system. To fulfill these requirements, adequate methods have been developed and as a specialized software linked to the system. This CAM submodule and concept of geometric modeling of work pieces have been written in the object language of the
system (macro language) GRIP Unigraphics. Only then it is possible to machine the complex, non-rotational work pieces with an assumed dimensional accuracy and to semi-automatic the process of design and manufacturing. The methods of a tool path generation include using standard ball end mills, end mills, standard turning tools, and button type turning tools.

2. Generation of a lathe tool path for machining processes

Geometric modeling in the most advanced, integrated CAD/CAM/CAE systems (e.g. CATIA, Unigraphics, CADDS 5, Pro/ENGINEER, I-DEAS, Euclid, etc.) may be successfully used for shapes relevant to the three-axis machining on CNC lathes. Nevertheless, coordinate measuring machines (CMM) enable verification of work piece shapes, with an accuracy of micrometers, along the generated tool path and it turns out that in many cases the achieved machining accuracy is below requirements. Reasons for these errors are usually questionable and a quantitative solution of the problem is a difficult one. The presented approach is intended to separate inaccuracies due to machine tool inaccuracies from inaccuracies due to programming methods applied in the system.

The complexity of the shapes being machined on the CNC lathes is related to intricacies of work piece cross-sections along the axis of revolution. The increasing complexity of shapes can be exemplified by boundary curves of cross-sections as follows:

- one curve, defined either analytically or parametrically, e.g. ellipses, closed curves defined by points;
- curves joined tangentially, e.g. NURBS curves defining cams;
- curves or straight lines joined non-tangentially (with edges), e.g. rectangular or polygonal structural elements;
- curves or straight lines with edges, twisted along the axis of revolution, e.g. cross-sections of an airscrew.

Practically, the capabilities of a geometric modeling, available in integrated CAD/CAM systems, are meaningfully limited by the shape of cutting tools and milling cutters. Apart from that, a serious restriction arises from the fact that a tool axis intersects the axis of work piece revolution. All these restrictions can be treated in much the same way as in four- and five-axis milling. The experience from part programming for multi-axis CNC milling has been taken into consideration while analyzing the cutting processes on three-axis lathes.

The differences between cutting processes for multi-axis milling and turning/milling operations occur when a precision machining on three-axis lathes is required, because there are differences in algorithms generating the tool paths.

Interpolation schemes applied in milling processes are based on linear segments, but in turning/milling processes on three-axis lathes the trajectory through interpolation points approximates the segments of spiral of Archimedes.

The spiral of Archimedes is algebraically accurate when a spiral segment lies in a plane as well as acceleration and deceleration motion of the tool are negligible. It is clear from the formula for a spiral of Archimedes, modified for interpolation purposes:

\[ r = a \cdot \Phi \]

where \( r \) is a distance between a point on a spiral and the axis of a work piece rotation (radius vector measured in mm); an increase or decrease of the radius vector in 1° (measured in mm/degree); \( \Phi \) an angle between the line, passing through the point on the spiral and the polar coordinate origin, and the line for \( r = 0 \) and \( \Phi = 0 \). In polar coordinates, all calculations as to spiral segments are much easier than in Cartesian coordinates.

The starting point \( P_1 \) as the radius vector \( r_1 \) and the ending point \( P_2 \) as the radius vector \( r_2 \) define the segment of the unique spiral of Archimedes. It must be pointed out that in such a spiral \( \Phi = 0 \), usually, does not agree with X-axis. From the spiral equations can be derived many useful formulas, e.g. radius of curvature, lengths of spiral segments, tangent and normal lines at the point on the spiral, etc. For example, the formula for a radius of curvature is as follows:

\[ R = \left( r^2 + a^2 \right)^{3/2} \]

Spiral segments as a curve are not available in CAD/CAM systems, therefore, the spiral segments should be substituted by curves, which best approximate them and are available in the system.

After having analyzed errors of approximation according to the established criterion, it has been found that 3° B-spline curves, 3° spline segments, and 3° NURBS curves can be used for substituting the spiral segments.

For example:

- spiral segment \( r_1 = 50 \text{ mm}, r_2 = 45 \text{ mm} \)
  \( \Phi_1 = 0°, \Phi_2 = 90° \)
- points every 5°
  3° spline, error = 0.0001 mm
  3° B-spline, error = 0.00001 mm
  3° NURBS, error = 0.00001 mm
- points every 1°
  3° spline, error = 0.000001 mm
  3° B-spline, error = 0.000001 mm
  3° NURBS, error = 0.00001 mm

The error is a maximum distance between the spiral segment and the curve approximating this segment. At an arbitrary point on the parametric curve, substituting the spiral segment, all necessary geometric properties can be obtained, e.g. tangent vectors, unit normal vectors, unit binormal vectors, lengths of segments, angles of points on segments, distances between the segment and other curves, etc.
During a cutting process, both turning tool or milling cutter are situated on a helical surface as a drive surface and with tool axes passing through the axis of work piece rotation. The tool axis of the turning tool is the axis of the ball end mill approximating the cutting tip. The helical surface is defined as a ruled surface, but a very specific one. The helical curves defining this ruled surface are computed from the parametric equations for cylinders with radius $r_c$:

\[ x = r_c \cos \alpha, \quad y = r_c \sin \alpha, \quad z = h \left( \frac{\alpha}{360^\circ} \right) \]

For three-axis lathes, $x$ and $y$ are in the range 20–700 mm, $z$, determined by feeds per revolution, is in the range 0.02–0.3 mm per revolution, i.e. 360° (h — helical lead). It means that in one revolution the helical surface is very close to the plane perpendicular to the axis of work piece rotation. Thus, if a spiral of Archimedes is used, the interpolation segments will be negligibly shorter than they should be, but it means that this difference can be treated as a safety coefficient for a tool path calculation. As a matter of fact, there is also 2 – 4-axis mode of machining on three-axis lathes; it is carried out in sequences of two blocks, in one block $x$ and $y$ or $x$ and $C$ are given.

Typical interpolation schemes have been shown in Fig. 1. Two different shapes make it easier to comprehend all ramifications of the interpolation task.

The developed interpolation method using spiral segments starts from an assumption what is a location of a mid-curve on which the initial interpolation points lie. An offset of a mid-curve from the curve defining inner boundaries of the tolerance range is established by $k \cdot (\text{intol} + \text{outtol})$. When $k = 0.5$, the mid-curve divides the tolerance range by halves. This is mostly applied because it enables effective compensations for the precision machining. Initial cutter location points $P_{1cl}, P_{2cl}, \ldots, P_{ncl}$ are generated by the iteration algorithm, starting from checking locations of spiral points defined in polar coordinates (initially every 0.25 of a segment angle). If any of the points exceeds the radius of an outer curve or is less than the radius of the inner curve, the new iteration step will be initiated. If all points are between inner and outer curves and the minimum distance from these curves is less than assumed accuracy, the iteration is treated as successful and it goes to the next calculation task. Usually, 3–5 iteration steps are required to achieve the assumed accuracy. The accuracy is determined by a fraction of the tolerance range ($\text{intol} + \text{outtol}$), e.g. 0.05 ($\text{intol} + \text{outtol}$).

Points $P_{1cl}, P_{2cl}, \ldots, P_{ncl}$ are calculated without any regard to the shape of the tool. Locating a tool tip in these points as well as a tool axis in direction of the point representing the axis of a piece work rotation, we can establish a minimum relative distance $d_{in}$ between the tool shape curve and inner curve. Then, if the tool shape curve intersects the inner curve, respective cutter location points $P_{1cl}, P_{2cl}, \ldots, P_{ncl}$ are moved along the tool axis by a vector computed from the respective vector $d_{in}$. These are the actual CL data points for postprocessing.

There are special cases when this algorithm fails to find out a compensation vector for CL data points. If we still want to keep the assumed dimensional accuracy, it is necessary to reconsider the geometry of the tool or a position of mid-curve between inner and outer curves. In advanced CAD/CAM systems, there are some part programming statements for five-axis milling, which generate a tool path in a similar way but using linear segments for interpolation. This set of statements can be modified and adapted to the part programming of three-axis lathes. Rulings on the drive surface, representing the helical surface, can serve as an orientation of a tool axis and tool end points can be computed without described software.

But if we cannot achieve the required dimensional accuracy (0.02–0.05 mm) even on machine tools with higher accuracy, the applied software must be verified as well. CMMs, measuring dimensional errors along the generated tool path, help in identifying the source of errors. More often than expected it turns out that dimensional errors, particularly in complex work pieces, result from part programming errors or from algorithm errors inherent in the system.

The specialized programs for a generation of tool paths, described above, can also be used for the purpose of checking the CL data points from other programming.
systems. These points can be read-in and verified in the same way as described. All intersections of an inner curve are listed and graphically presented inside or outside of a tolerance range. Besides, we can establish what should be intol and outtol, defined in the program, in order to achieve required machining tolerances.

Programs have been written in GRIP (Unigraphics) and added as an extension to the system.

3. Postprocessors for the three-axis CNC lathes

Postprocessors are important components of the part programming modules of CAM systems. In advanced CAD/CAM systems there are postprocessor generators for CNC lathes, but they are limited to the traditional mode of turning, it means to the two-axis CNC. Time consuming and expensive writing of the postprocessors in high level languages such as Fortran, has been substituted by writing the specification of the machine tool on special forms available as input data to the generator programs in the system. Practically, it is a semi-automation of postprocessor development.

For some, non-typical control units or mechanical drives, postprocessors obtained from the generator must be carefully tested. Three-axis lathes, as a new design, are not supported by postprocessor generators in the systems. Usually, there are three modes of turning/milling operations on the three-axis lathes. These modes are:

1. first mode, traditional turning operations based on the control of x and z axes;
2. second mode, milling and turning operations based on the control of x, y, z axes;
3. third mode, milling and turning operations based on the control of x, z axes and rotational C-axis.

It depends on a shop floor, but mostly used are traditional turning operations even on three-axis lathes. Partly, it is due to the fact that a manual preparation of NC machine programs precludes a production effectiveness. Therefore, the postprocessor should cover all three modes of operations. First of all, using programming tools available in the system, the postprocessor for the first mode is developed and it is for a primary postprocessing. In the primary postprocessing as an input is cutter location data and as output is a NC machine program. The primary postprocessor covers all main lathe operations, as roughing operations, finishing operations, grooving operations, threading operations, drilling operations.

To integrate all modes of operations a secondary postprocessing has been implemented. This secondary postprocessor consists of specialized programs modifying the output obtained from the primary postprocessor when the third axis is being used. In the secondary postprocessing two input data files are resolved and compared: the cutter location data file and the output file from the primary postprocessor.

The main steps involved in the secondary postprocessing are as follows:

1. read-in the postprocessor words from the CL file, and searching for words defining modes of operations;
2. strict comparison of respective records in the CL file with the output block from the primary postprocessing and correction of the block if required;
3. computing the y-axis or C-axis from six coordinates of the CL file;
4. writing the final blocks of NC machine programs.

Programs covering these steps have been written in the GRIP (Unigraphics) and can be operated from dialog windows of the system. In order to be sure that NC programs are reliable, the secondary postprocessor must be thoroughly verified and tested, starting from simple shapes and ending on the most complex shapes. A complex shape case is described in the next section.

4. Case studies — integrated process of an airscrew design and manufacturing

To verify the three-axis turning the integrated design and manufacturing of airscrews has been chosen. The steps carried out are:

- programming the design concepts of airscrews and generating the various geometric models of airscrews;
- generating the shape of airscrew on shape rolls and machining the shaped roll using the CNC lathe, when shape rolling is assumed as a production method;
- programming the turning process and machining airscrews using the CNC lathe, when cutting process is assumed as a production method.

As a matter of fact, turning the airscrew blades has not been elaborated in the paper, because this process has been applied so far in Polish aircraft industry, using templates on traditional lathes rather than a third axis of CNC lathes.

Advanced CAD/CAM/CAE systems enable integration of design and manufacturing of such complex products as airscrew blades. These systems are also necessary in order to make concurrent engineering a feasible solution for airscrew production. The traditional manufacturing of airscrew blades still includes a hand operated metalworking, and then the surface quality and dimensional repeatability are below required level. Even application of CNC machining does not eliminate many shortcomings arising from cutting processes (e.g. shape deformation after cutting operations). Notice that the airscrews are still mostly built from duraluminum. A new approach to the manufacturing is closely interrelated to a new approach to the design of airscrew blades. The presented concept of design and manufacturing for airscrew blades can be carried out by using the most advanced CAD/CAM/CAE systems. Geometric modeling of airscrew blades has been semi-automated, and as a result the design concept...
can be reproduced for various design data sets. Therefore, a new prototype version can be generated very rapidly. The design concept has been defined by using the object language (GRIP language) of the system. The created solid body of airscrew undergoes the scrutinized geometric and engineering analysis, e.g. dimensions and shape analysis, mass and dynamic analysis, finite elements analysis, etc. The proposed manufacturing method of airscrew blades replaces metal-removal processes by shape rolling. Reproducing the airscrew shape on work surfaces of shaped rolls and meeting the requirements of the rolling process cause considerable difficulties. They can be overcome by a specialized software, which optimizes the position of the airscrew axis and chords of airfoils for transformation onto two shaped rolls. In addition, the software can be used for designing the required series of rolls (roll-pass design). The specialized CAM submodule has been developed in order to apply the three-axis CNC lathes or four-axis milling machines for machining the shaped rolls.

Semi-automation of the design procedures does not mean that the new rules of design are applied, included algorithmic relationships for almost each design step. And this is a crucial point in developing programs for any class of complex products, in order to implement concurrent engineering techniques. In general, the product from a traditional design and from software-based design within integrated CAD/CAM systems should be the same. The only differences are time (minutes rather than months) and reliability of engineering analysis, what means increased quality and productivity. Also, simulation of all important design, engineering and manufacturing steps eliminates, to a significant degree, the necessity of prototyping after each new design version.

The shape rolling is being proposed as a new production method for airscrews. Of course it would not be proposed for a traditional design due to enormous complexity of roll shapes for airscrew forming.

The sequence of procedures in the specialized CAD submodule:

**Modeling the upper and lower shaped rolls**

1. Iterative searching for the optimal orientation of the airscrew model; division of the model on two parts by division surface
2. Changing the airscrew axis orientation and division surface orientation
   - Checking if the roll perimeters criterion is met
3. Transformation of the upper surface of the airscrew to the work surface of the upper roll
4. Transformation of the lower surface of the airscrew to the work surface of the lower roll
5. Creating the surfaces and the solid bodies of the upper and lower shaped rolls
6. Mass and dynamic analysis of the shaped rolls

Hundreds of data items must be established in order to parameterize the shape of an airscrew. Approximately for 1 airfoil definition 22 points are used, and for a blade definition 7–11 airfoils are required along an airscrew axis. It means that strictly interactive approach within the CAD/CAM system is ineffective, particularly when the concurrent engineering techniques are considered. This was a main reason for developing a specialized software for airscrew design and manufacture.

Mostly airscrews are made up of four main parts: an airscrew blade, a cylindrical shank, a transitional part between blade and shank, and a blade tip. Each of the parts is modeled separately and then, after Boolean operation of adding, a solid body of the airscrew is created.

Two airscrew surfaces must be precisely transformed on work surfaces of two rolls, and then the rolls cease to be cylinders. The resulting shape of rolls can be seen in Fig. 2. Two airscrew surfaces are obtained by intersecting the airscrew surface with a ruled surface defined by extreme points on trailing edge and edge of attack. The two cylinders and airscrew model must be precisely located in the common coordinate system, and then the orientation of the airscrew can be iteratively changed. Orientation of the airscrew in relation to the cylinders cannot be arbitrary, because in this case the perimeter of the upper roll would not be equal to the perimeter of the lower roll, and slips between rolls would occur. Searching for optimal orientation of the airscrew the criterion of the equality between roll perimeters must be
Two movements are sufficient in order to find optimal orientation:

1. rotation of the airscrew axis in the plane of airscrew rotation (plane XZ),
2. rotation of the airscrew cross-sections around the airscrew axis.

When the optimal orientation is determined the two airscrew surfaces are transformed to the respective rolls.

It must be pointed out that the procedure presented above includes only the last stage of the shape rolling. The roll-pass design process and establishing the blank shape is being investigated and adequate software is being developed. Practically, it means that for as many repetitions of the procedure as many rolling stages are planned.

There are two ways of CNC machining suitable for shaped rolls: three-axis turning and four-axis milling. Four-axis milling is rather conventional and its advantages and disadvantages are known.

Much more challenging is the three-axis CNC turning or CNC milling on three-axis lathes. Turning and milling the non-rotational shapes, by using the three-axis CNC lathes and turning centers, create opportunities for economical and precision machining of such products, as shape rolls for shape-rolling operations, cams (with non-ruled surfaces included), camshafts, non-rotational shafts, prostheses for total hip replacements, airscrews, etc.

5. Conclusions

The three-axis turning/milling using CNC lathes has a great potential for change in production methods and techniques used for complex products and oriented towards improvements in economics, quality and effectiveness of cutting processes.

Further Reading