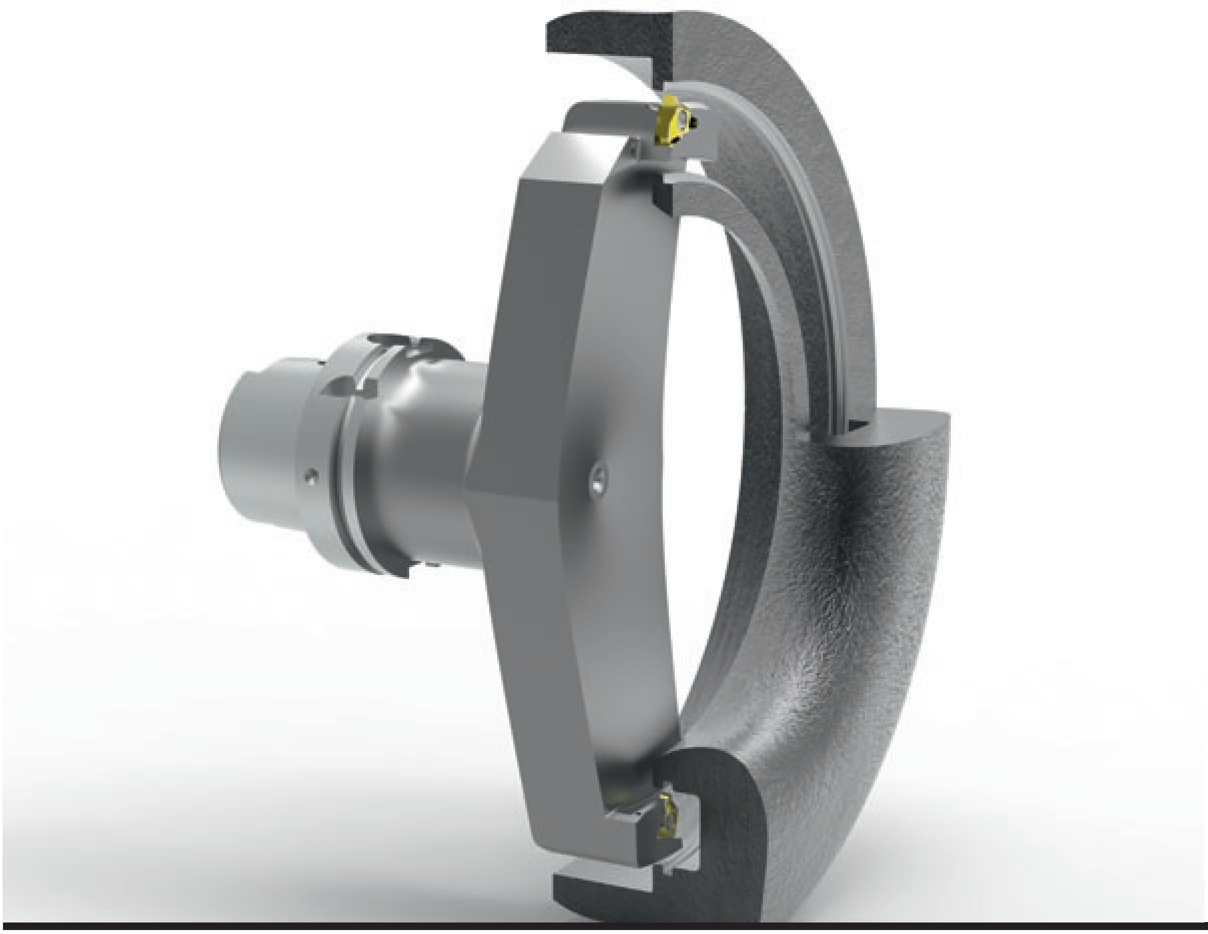
TECHNOLOGY
REPORT

How does interpolation turning work?

01 | Interpolation turning



Prof. Dr.-Ing. Eckehard Kalhöfer | Dr.-Ing. Dirk Sellmer

Interpolation turning has been up until now a little-known machining process, that can be used to realise turning processes on machining centres. It is used, for example, for recesses on cubic workpieces that could otherwise be produced using circular milling. However, many other geometries typical for turning, for example tapers, can also be produced using interpolation turning on machining centres.

An important prerequisite for the use of this process is that the machine has a main spindle that can be operated as a position-controlled axis. If this precondition is met, in many cases the advantages of interpolation turning over circular milling or re-clamping the workpiece in a lathe can be exploited. This article describes the machining process including the NC programming and a few important relationships for the application and the tool selection.

Process

How does the process work?

For interpolation turning the main spindle on the machining centre is switched to position-controlled operation (also called axis operation). It can then be operated like a rotating axis. During recessing or facing on lathes, the cutting edge forms a spiral as it moves towards the workpiece. Here the feed per revolution is the pitch of the spiral. This spiral movement is normally approximated during interpolation turning on machining centres using semicircles, i.e. the feed axes move in a semicircle during circular interpolation (in the x-y plane) and at the same time the main spindle is synchronised to the movement of the feed axes (Figure 1). The centres of the semicircles are slightly displaced in relation to the central axis of the recess. The result is movement of the cutting edge that is very similar to the spiral during conventional turning on lathes. (The maximum error on the radius of the path actually covered compared to the spiral is approx. 5% of the radial feed per revolution f. At a feed of 0.15 mm the maximum error compared to the spiral is approx. 7.5 µm.)

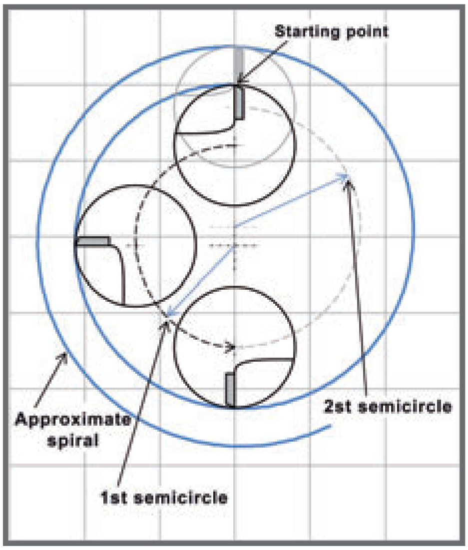


Figure 1: Orientation of the tool synchronised to the position in the xy-plane

INDEX

Process

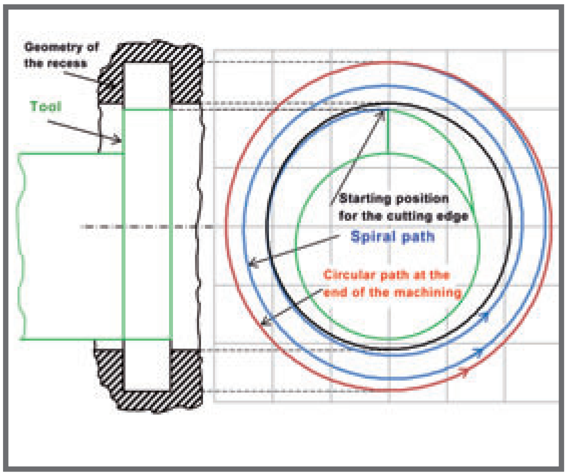


Figure 2: Tool movements during interpolation turning (spiral, circular path)

After reaching the final diameter, one or two circular paths are completed without radial feed (Figure 2). Contrary to the spiral path that is only approximated, here – apart from the inaccuracies of the feed axes – the theoretically correct path is covered „exactly".

Similarly, it is also possible to turn longitudinally or to turn tapers. Here, along with the movement of the C axis (the main spindle) a movement in the z axis is also superimposed on the circular path in the x-y plane.

Applications and advantages of the process

Applications and advantages of the process

The most common application for interpolation turning is recesses on workpieces that are manufactured on machining centres. A tool for the process is shown on Figure 3. Inclined recesses, relief grooves and the rotary machining of face surfaces are also possible (Figure 4). Examples of workpieces for recessing are brake master cylinders, brake calipers, hydraulic valve housings, swivel bearings or gearbox housings. It is necessary to manufacture slots in all these workpieces, for example for sealing rings. Compared to circular milling, which can also be used to machine these slots, interpolation turning is in many cases quicker. The cause of the long machining time during circular milling is the excitation of vibration in the often long circular milling cutters (which are also not very rigid) due to the interrupted cut during milling. To limit the amplitude of the vibration, relatively small feeds and/or material removal rates must be used for machining.

Figure 3: Tool for recessing through interpolation turning

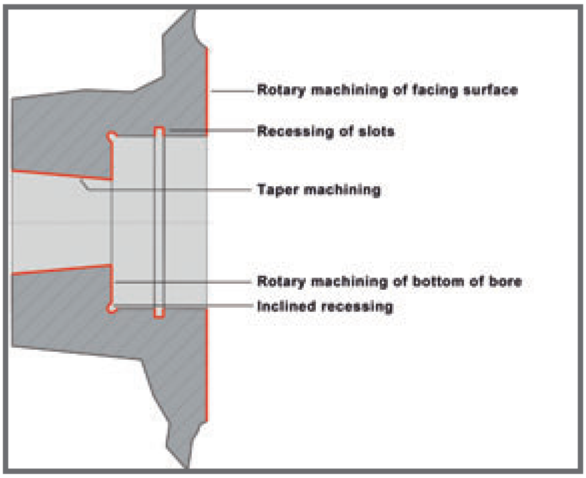


Figure 4: Possible machining processes during interpolation turning

Similar to turning, during interpolation turning the thickness of cut and as a result the cutting force are constant; the tool is not excited into vibration as it is with circular milling. This is an essential advantage of interpolation turning, particularly in the case of tools with long projection.

In addition, the tool shank for interpolation turning can be designed to be significantly stiffer than for circular milling. In many cases these two aspects mean that, compared to circular milling, significantly higher machining volumes and shorter machining times are possible. For example, by changing from

Applications and advantages of the process

circular milling to interpolation turning it was possible to reduce the machining time by two thirds when three seal slots in a differential housing made of GG25.

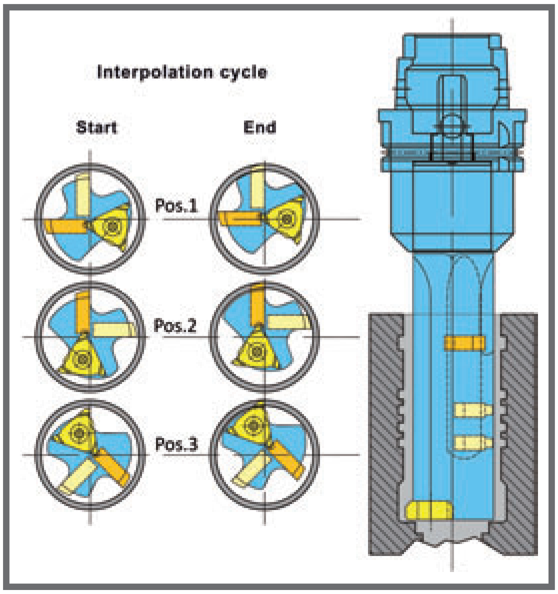


Figure 5: Dependency on the radius of the circular path of the achievable tool speed and feed rate on the path

NC programming

There is a further opportunity to save time if several cutting edges for different machining tasks are integrated into a tool for interpolation turning. Here tangential inserts have an advantage through the reduced weakening of the tool shaft.
Such a tool is shown in Figure 5. First the four middle slots are machined in position 1. Then the tool is rotated by 90° in the middle of the bore and the upper slot machined in position 2. After a further rotation the relief groove at the bottom end is machined in position 3 by recessing and turning longitudinally. If custom tools are viable due to correspondingly high part quantities, productivity is high on the one hand due to the higher material removal rates that are often possible, and on the other hand due to the possibility of arranging several cutting edges for different machining tasks in one tool, and as a consequence saving in tool changing time.

Examples for the machining of face surfaces are thrust bearings in engine blocks and cylinder heads and the machining of control edges in valve housings. In both cases circular milling tools typically have very long projections and are therefore not very rigid, as a consequence circular milling must be undertaken with very low feeds and material removal rates or, for the thrust bearing, a complicated actuating tool must be used. Machining using interpolation turning has a further advantage for the machining of the control edges: the control edge, that is the transition between diameter and face surface, is produced by a large number of cuts during circular milling. As a result the scratches on the individual cutting edges are reflected in the control edge (similar to a saw tooth profile). During interpolation turning, on the other hand, a very even edge is produced, which is not only of advantage for the machining of the control edges.

The tool life during interpolation turning is often greater than with circular milling due to the continuous cut without vibration. The difference is particularly significant if there is a relatively large tool wrap angle during circular milling. In an example in which the V-band for a turbocharger is machined from high-temperature cast steel, 100 workpieces can be machined with one cutting edge using interpolation turning. By way of comparison, using circular milling with a four-cutting edged tool only 50 workpieces can be machined before all four cutting edges have to be replaced. This means that eight times the tool life per cutting edge is achieved during interpolation turning.

NC programming for interpolation turning

NC programming for interpolation turning will be explained based on the popular NC controller Siemens 840D. In addition to programming, cycles for interpolation turning are also integrated into the controllers from some machine manufacturers.
As already mentioned above, an important prerequisite for the use of interpolation turning is that the machine has a main spindle that can be operated as a controlled axis (C axis). In some cases the C axis (that is the main spindle position-controlled rotating axis) must be placed in an axis group with the other feed axes using the instruction FGROUP. The axes then operate as a path axis. This action is not necessary on other machines. In this case the main spindle operates as a synchronous axis to the path axes (in x, y and z direction), this means it moves synchronous to the path axes and takes the same amount of time as the path axes for the movement in the related NC block.

NC programming

The NC programming will now be explained in detail based on the example of a recess to be machined in a bore. It is also possible to machine a face surface from the inside out using the same NC program, as long as the inside diameter of the face surface is larger than the tool diameter. For better understanding and in particular for the flexible usage of the NC program, R parameters are used in the program such that it can also easily be integrated into custom NC programs. The program is written for a Hermle C30 with Siemens 840D controller. Here the instruction FGROUP is not necessary. The main spindle only needs to be changed to position-controlled operation using the M function M70.

In a previously prepared bore with a diameter of 30.5 mm, a slot is cut to the diameter 36 mm. The slot width is defined by the tool geometry. In the lines N50 to N120 the necessary values for the geometry to be produced, the radial feed per revolution, the radial path of the tool's cutting edge and a retraction plane in the z direction are assigned to the corresponding R parameters. The start diameter for the machining is a little smaller than the bore diameter to prevent collisions. This safety distance must be adjusted based on the tolerances for the specific workpiece. For the radial feed per revolution values between 0.1 mm and 0.15 mm are typical.

In the lines that follow up to N190 various parameters necessary for the machining are calculated (see also comments in the NC program). The stated radial feed is reduced such that there is an integer number of revolutions to reach the final diameter. The diameter of the circular path, which is covered using the feed axes in the xy plane, is given by the machining diameter and the tool diameter (N190) and is typically very small at the start of machining.

After the selection of various G functions and the insertion of the tool, the tool is moved to the position of the bore, the spindle placed in position control (N280) and positioned at the correct angle (here at 270°). This value must be checked in the specific case and corrected if necessary (for example to 90°), such that the tool's cutting edges are orientated as shown in Figure 1.

```
N10 ; Interpolationsdrehen eines radialen
Einstichs in einer Bohrung
N20 ; Beispielprogramm fuer Hermle C30 mit
Siemens 840D
N30 ;-----------------
N40 ; Eingaben:
N50 R1= 30; Startdurchmesser der Bearbeitung
N60 R2= 36; Enddurchmesser der Bearbeitung
N70 R4= 0.12; radialer Vorschub pro Umdrehung
N80 R8= 50; x-Position der Bohrungsmittelachse
N90 R9= 60; y-Position der Bohrungsmittelachse
N100 R10= -20; z-Position für Bearbeitung
N110 R11= 100; z-Position für Rueckzugsebene
N120 R13=14; Umlaufradius der Werkzeugschneide
N130 ;-----------------
N150 R0= 0; Zaehler
N160 R3= (R2-R1)/2; radialer Fahrweg
N170 R5= ROUND(R3/R4)+1; ganzzahlige Anzahl der
Umdrehungen, so dass angegebener Vorschub nicht
ueberschritten wird
N180 R4= R3/R5; korrigierter radialer Vorschub
(damit Enddurchmesser in ganzzahliger Anzahl von
Umdrehungen erreicht wird)
N190 R12= R1-(2*R13); Durchmesser der Kreisbahn
in xy-Ebene
N200 G40; Werkzeugradiuskorrektur aus
N210 G64; Bahnsteuerbetrieb
N220 G71; metrische Massangaben
N230 G90; Absolutbemassung
N240 G54; Nullpunktverschiebung
N250 T="FRAESER12"
N255 L6; Werkzeug einwechseln
N260 G0 Z=R11
N270 G0 X=R8 Y=R9
N280 M70; Spindellageregelung ein
N290 G01 SP=270 F20000
N300 G01 Z=R10 F5000
N310 G17; Arbeitsebene XY
N320 G91; Inkrementalbemassung
N330 G01 X0 Y=(R12/2) F20000
N340 ; Spirale, R5 Umdrehungen
N350 START1:
N360 R12= R12 + R4
N370 G02 X0 Y= -R12 I0 J=-(R12/2) SP=180
N380 R12= R12 + R4
N390 G02 X0 Y=R12 I0 J=(R12/2) SP=180
N400 R0= R0 + 1
N410 IF R0<>R5 GOTOB START1
N420 ; Kreisbahn
N430 G02 X0 Y0 I0 J=-(R12/2) SP=720 TURN= 1;
zwei volle Umdrehungen
N440 G01 X0 Y=-(R12/2); in die Bohrungsmitte
fahren
N450 G01 Z=R11 F5000
N460 M5; Spindel halt
N470 M2; Programmende
```

Then the tool can be moved into the bore. The plane for the circular path for the actual machining is selected and a change made to incremental dimensions. The start position is moved to in line N330 such that it is then possible to machine along the (approximate) spiral. Here the feed rate for the machining is also given. With F20000 a feed rate is programmed that is so high, it will definitely not be achieved on the path. The consequence is that the maximum possible feed rate is used on the path. This feed rate is dependent in particular on the dynamics of the machine tool and the radius in the xy plane; it will change during the machining. The spindle speed for the tool and the cutting speed are directly dependent on the feed rate and the radius on the path.

For the actual machining the cutting edge moves along semi-circles of which the diameter is increased by the radial feed after each half revolution (N360, N380). The radius is therefore increased by half the radial feed after each half revolution. After one complete revolution it is thus increased by the radial feed. For this purpose the rotary movement of the main spindle is superimposed on a circular path movement in the plane by the two feed xy planes as described in Figure 1. The semicircles are program-

NC programming

med in the xy plane; the main spindle rotates half a revolution synchronised to this movement (N370, N390). After each complete circle the counter R0 is increased by one (N400) until the number of revolutions R5 and therefore the final diameter is reached (N410). Then there are two full revolutions at the final diameter (N430). During the first revolution the thickness of cut reduces continuously until it is theoretically zero at the end of the first revolution. The second revolution can often improve the accuracy of the diameter produced. Given a sufficiently sharp cutting edge, part of the tool deformation is relieved such that a fine chip is still removed (similar to sparking out during grinding).

In the NC program described, tool radius correction is not used (G40). In principle it is also possible to work with tool radius correction. Then the final diameter for the machining can be adjusted without changing the NC program.

The dynamics of the machine can be influenced by the instruction BRISK and its counterpart SOFT. After the BRISK instruction the feed axes move with full acceleration until they reach the final velocity. With the SOFT instruction (default) a smooth acceleration process is defined and greater path accuracy and lower machine loading achieved. The machine manufacturer can, however, integrate a different method during the implementation of the controller. It must be checked in the specific case by means of tests whether the machining time is reduced or increased by the BRISK instruction.

Process analysis

Process analysis

A few relationships on the dynamics of the machine and the path errors, along with the radial run-out accuracies dependent on these dynamics will be discussed as an aid to understanding the process of interpolation turning. It is assumed the machining is internal such as machining slots in a bore. The considerations can, however, also be transferred to the external machining using interpolation turning. The related trials were undertaken in a Hermle C30 dynamic. Movement was in circular paths in the xy plane during which – if not otherwise stated – the main spindle was position controlled (as for interpolation turning). The measured values were recorded directly from the NC controller using Servotrace. These values are the position values from measuring systems integrated into machine (linear scales for feed axes and encoders for main spindle). This method also means that the effects of tool deformation due to cutting forces on the dimensional accuracy of the workpieces are not taken into account here. As has already been made clear above, the tool speed and the cutting speed for the machining can not be explicitly stated in the NC program. Both are related to the feed rate that can be achieved on the path and are limited by the dynamics of the machine tool. The feed rate that can be achieved is in particular dependent on the radius of the circular path and the axis acceleration. On the usage of path control, the stated (maximum) acceleration of the machine tool is typically not achieved, but a lower acceleration.

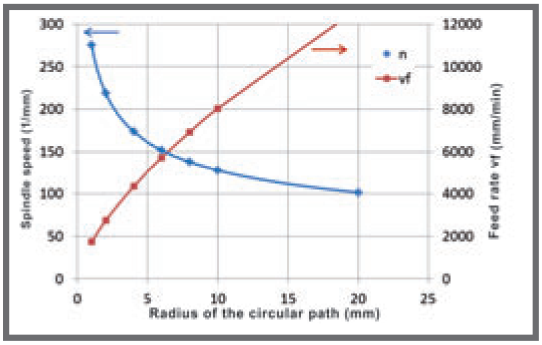


Figure 6: Dependency on the radius of the circular path of the achievable tool speed and feed rate on the path

An example of this relationship is shown for the machine tested in Figure 6. For this purpose circular paths of varying radii were covered during which movement of the main spindle was also in position control and was synchronised to the feed movement. (Similar measurements in which only the circular path is covered and the main spindle is not synchronised with the circular path movement produce the same values for the feed rate. This means that the feed rate that can be achieved is not limited by the synchronous movement of the main spindle in relation to the feed movement.) The larger the radius of the circular path, the higher the feed rate and the lower the tool speed. (As the main spindle is synchronised to the circular path movement, the rotary frequency of this circular path movement is equal to the tool speed.)

The spindle speed that can be achieved increases significantly as the radius of the circular path reduces. A specific machining diameter is therefore machined

Process analysis

faster and with higher cutting speed with a small circular path radius and correspondingly larger tool diameter than with a larger circular path radius and smaller tool diameter. For a short machining time the aim is therefore a circular path with a radius as small as possible in the xy plane, combined with a relatively large tool diameter (the diameter of the circular path is the difference between the machining diameter and the tool diameter). The tool diameter must as a minimum be a little smaller than the diameter at which the machining is started. Then the radius of the circular path is a little larger than zero. The spindle speeds that can be reached are relatively low such that an imbalance in the tool only causes very low imbalance forces.

The radial errors on the setpoint position for the tool's cutting edge during one tool revolution are shown in Figure 7.

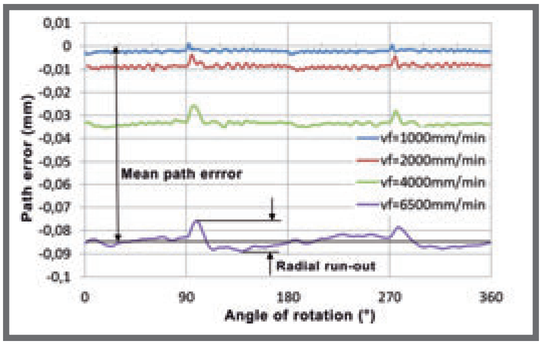


Figure 7: Path error versus the angle of rotation with varying feed rates and a circular path radius of 8 mm

They are caused almost entirely by the path error on the circular movement (in the xy plane).

Summary

The contribution of errors in the synchronisation of the main spindle with the movement of the feed axes is less than 0.1 µm in the tests undertaken. The mean path error (that is the difference between the programmed radius and the actual radius of the movement) increases with the square of the (actually achieved) feed rate. During these tests a circular path radius of 8 mm was always programmed and varying feed rates were used.

The actual diameter produced is a little smaller than the final diameter programmed for the machining. The error on the diameter can therefore be significantly reduced by a reduced feed rate during the last one or two revolutions at constant diameter. The radial run-out (the difference between the largest and smallest path error during one revolution) also increases (approximately linearly) with the feed rate.

The related values for the radial path error for different radii of the circular path (from 1 mm to 8 mm) and the feed rates that can be achieved are shown in Figure 8. The feed rates that can be achieved and the path errors increase with the increasing radius of the circular path.

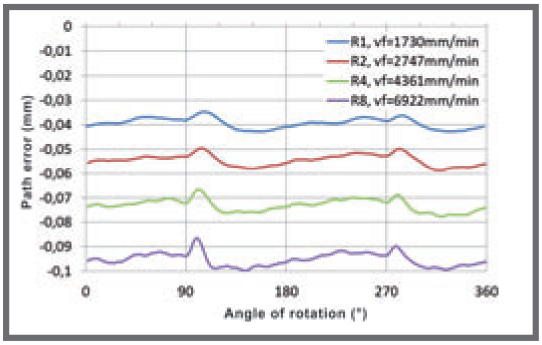


Figure 8: Path error versus the angle of rotation at varying circular path radii and maximum achievable feed rate in each case (R1: 1 mm radius, ..., R8: 8 mm radius)

A specific machining diameter is therefore machined more accurately with a small circular path radius and correspondingly larger tool diameter than with a larger circular path radius and smaller tool diameter. At the same time with a smaller radius for the circular path the machining time is also reduced, as already shown above.

Summary

Rotary machining is possible on machining centres using interpolation turning. An important application is the recessing of slots during external machining as well as during the machining of bores. Here the process competes with circular milling and has a number of advantages over this method. Due to the continuous cut the machining process will excite less vibration than during circular milling such that often higher material removal rates and therefore faster machining is possible. The productivity advantage is particularly high if tools with long projection need to be used, and in the case of relatively small bores. Compared to a circular milling tool, the tool shank for interpolation turning can be designed to be stabiler and stiffer. Due to the continuous cut the

Summary

tool wear is often lower than with circular milling. On the usage of custom tools, several cutting edges for different tasks can be arranged in one tool, as a result tool changing time can be saved. For some machining cases, such as control edges on valves, an improvement in quality compared to circular milling is achieved by the even cutting action.

Along with recessing slots, e.g., face turning, taper turning and inclined recesses are possible using interpolation turning on standard machining centres. With expanded programming systems that go beyond the manual programming described here, complex machining tasks such as chamfering the transition between a bore and a plane are possible if the bore is not perpendicular on the plane. The process can be used effectively in large scale mass production where a short machining time is in the foreground. Using simple, economical tools – such as an internal lathe tool for cutting slots – the process is also of interest for small scale production with its great flexibility.

During the interpolation turning of a recess the relative movement between cutting edge and workpiece is achieved by superimposing a circular path generated using the feed axes and the rotary movement of the main spindle, which is synchronised to this path movement. The related NC programming has been discussed in detail. In addition, cycles for interpolation turning are available on some machine tools.

Typical values for the radial feed per revolution are between 0.1 mm and 0.15 mm. The cutting speed cannot be stated directly in the NC program. It is defined by the tool speed, which in turn is dependent on the actual feed rate achieved on the circular path. The feed rate is dependent on the radius of the circular path and the acceleration of the feed axes. In general the cutting speed is relatively low due to the limited scope for acceleration by the feed axes. It is calculated using the usual equation for the cutting speed, where the machining diameter is used and not the tool diameter.

For a machining time as short as possible, the tool diameter should be as large as possible for internal machining such that the radius of the circular path covered is as small as possible at the start of the machining. At the same time this characteristic is positive for the accuracy of the diameter machined. This accuracy can also be increased by undertaking the circular path movement at the end of the machining with reduced feed rate. A sharp tool cutting edge also improves the accuracy due to the lower static displacement of the tool.

Prof. Dr.-Ing. Eckehard Kalhöfer
is the holder of the Chair for Machining
at Aalen University.
eckehard.kalhoefer@htw-aalen.de

Dr.-Ing. Dirk Sellmer
is Head of Research at MAPAL Dr. Kress KG.
dirk.sellmer@de.mapal.com

TECHREP 01-E-01-030-0113-WD Printed in Germany. Right of technical modification reserved.

INFO: www.mapal.com

Further information regarding this subject can be found on our homepage or detailed information can be requested through:

info@de.mapal.com

MAPAL Präzisionswerkzeuge Dr. Kress KG
Obere Bahnstraße 13 | 73431 Aalen
Phone +49 7361 585-3820
Fax +49 7361 585-150