evropský sociální fond v ČR
EVROPSKÁ UNIE
MINISTERSTVO ŠKOLSTVÍ, MLÁDEŽE A TĚLOVÝCHOVY
OP Vzdělávání pro konkurenceschopnost
INVESTICE DO ROZVOJE VZDĚLÁVÁNÍ

Brno University of Technology - Faculty of Mechanical Engineering

A COLLABORATION ON THE CNC MACHINES USE

Learning Text

Prof. Ing. Miroslav Píška, CSc.
Ing. Aleš Polzer, Ph.D.

Brno 2011

The learning text was made in the frame of project:
Consulting point for the development of cooperation in the area operation innovation and transfer technology.   Reg. Nr.: CZ.1.07/2.4.00/12.0094

„The project was co-financed by the European Social Fund and the state budget of the Czech Republic.“

Contents:

Comment: The introductory document consists from eleven thematic chapters that are simplified and truncated to the whole extent and high lights of the course. The CNC and CAD/CAM technologies undergo permanent development so the next papers will be innovated to the current state of knowledge.

When studying each chapter we recommend the follow the rules:

Time consumption: xx hours

At the beginning of a chapter a recommended time to study it is mentioned. However, that time is only to your navigation so you can plan and manage your work of whole subject or a chapter. That time can be a very long for somebody, but insufficient to somebody else. There are real beginners but professionals too with wide experience in the fields, so the needed time can be very individual.

Goal: Having finished this paragraph you can

describe ...
define ...
solve ...

Then the goals and all skill and knowledge follow.

Explanation

The commentary and explanation of the studied matter follows like a definition of new terms, some interpretations follow, figures, tables and other references follow.

Summary

At the end of each chapter some main terms to master are repeated. If the items are not clear, please, come back to their explanations.

Questions

To prove that you understand and know well the subject of each chapter some questions from the theory follow.

Tasks

So that most of the theoretical terms have immediate significance and use in the data base practice, some practical problems are offered to you to be solved and trained. The importance of the tasks lies in training of your capability in troubleshooting of real situations and problems and getting of know-how.

Keys

The right results of the examples and right answers are mentioned in the final part of the text book. Anyway, use them just after finishing of your self-work to learn and train your own skill and level of knowledge.

Introduction

The fundamentals of machining technology can be traced to the period of the Industrial Revolution from the 18^{th} to 19^{th} century. However, the very significant development of the production branch can be found in the 20^{th} century and this progress has been running up today without any retardation.

This interpretation can be seen as misguided or even false, because the boom of crafts is evident before the Industrial Revolution with flourishing knowledge of skill and so called know-how in machining. For example, the American historian Lewis Mumford takes an in-depth look in the book Myth of the Machine for development of technology and mankind with words: *„What is normally considered as a technical backwardness for the period of 600 years before so-called Industrial Revolution is nothing else than a weird kind of backwardness in the knowledge of history. It is evident that the big advances of the 18th century stem in the prehistoric times or in the Bronze Age“.*

Very high stipulations for metal-working (mainly for military needs) accelerated national economy of many states. Speaking about productivity, machining of metals with tool machines is relatively young. All technologies dealing with metals were concentrated at smith works - up to the 19th century. Speaking about handling with tool machines, then the first mechanical drive can be regarded as a real and substantial progress. The next historical steps are presented with steam engine use and electrical motors lately. However, it is a manual production only. The first elements of automatic control and unmanned machines are linked to the 20th century and some terms changed during the time. For example – the CAM (Computer Aided Manufacturing) was intended originally for direct application of NC machines, robots, and operation logistics of blanks, materials and tools.

The historical development of CNC machines (digital technologies) covered a few fields in parallel including machine elements, production systems, control systems and machine entities. So called hydro-motors, electrically controlled, could be found about the year1950 and lately were substituted with electrically controlled motors. The optical systems for the feed back measurements (linear and rotational) were used. The first so-called NC knee milling machines were adjusted from conventional machines (Ferranti in Scotland, Parson in U.S.A.). The control systems worked on the principle of vacuum lamps (Record Play Back) with rectangular positioning and systems and magnetic data recording. The first NC machining (milling) centre on the base of transistors was developed by the company Kearney&Trecker. The integrated circuits were used firstly in the end of the 60´s and enhanced the production of the parabolic and spline interpolations. The NC machines were integrated into production lines.

The ball screws and hydro-static guides were used in the machine designs in the 70´s. The company Herbert displayed on the market the first turning centre with rotary tools for milling and drilling. The NC systems included memories (ROM and RAM) with their edits (the company Westinghouse). That amenity triggered the first CNC systems. The firm Kearney&Trecker introduced the first FMS (Flexible Manufacturing System).

Next development in the 80´s is characterized with tool and work-piece magazines, new sensors for monitoring of drives and other mechanisms. The controlling was based on the CNC/PLC with the multi-processor micro-computer structures. This period was important because of the crucial implementation of the centers into technology of cutting.

The high-capacity magazines with inter-operation logistics of tools and work-pieces were applied in the 90´s of the last century. Moreover, the increasing accuracy of piece production, high cutting productivity and performance and the open architecture are rampant. The higher versatility of machined parts led to the higher use of the flexible manufacturing systems.

The beginning of the 21st century was started with a new generation of manufacturing centers, multi-functional machines, integration and unification of the HW/SW (hardware, software). Advanced CAD/CAM systems are integrated into the CNC machines with supervised external workstation. However, the following parts are devoted to the most desired CNC and CAD/CAM technologies of machining today.

1. Fundamentals, technical data and programming of the CNC lathes and CNC turning centers (SPN 12 CNC, SP280SY)

Time consumption: 3 hours

Explanation

Comparing conventional machines compared to the CNC machining the flexibility in machining of various parts prevails as the key advantage. The change from one technology to another one can be done with a change of the control programme (NC programme) and a use of other facilities as tool and measurement equipments. The basic advantage lies in the semi-automatic or fully automatic work. All needed functions (like all motions of cutting tools, set-up of cutting conditions, exchange of cutting tools or work-pieces is achieved by execution of all NC blocks. All necessary information for a machining of a part should be included in the alpha-numerical codes. As the main production information can be mentioned:

- dimension information for a production of each surface parts,
- cutting data information – speeds, feed speeds, cooling fluids, etc.
- additional information (e.g. dwelling time, opening of safety shields, etc.)

Numerically controlled tool machines are designed for automatic work regime primarily. Programming of the machines can be done with a special operational panel directly or with the remote powerful workstations indirectly. So that the NC programming can be seen as CAD/CAM systems, workshop oriented programming and so called ISO programming (sometimes called as programming in the G-code).

This introductory chapter is focused on a typical turning machine and a manufacturing centre with the ISO programming use. These machines span the variety of lathes and a permanent effort to their improvement can be observed. Some information linked to them can be found in other chapters also where e.g. workshop oriented programming will be trained. However, the main attention now will be devoted to up-graded lathe after retrofit and a modern turning centre.

Questions

1. What does CNC mean? (in English and Czech)

2. What is the G-code (is it universal for all NC and CNC machines)?

3. What are the work regimes of the semi-automatic lathe SPN 12 CNC?

4. What is the recommended turn-off routine of the SPN 12 CNC?

2. ISO programming fundamentals for turning

Time consumption: 3 hours

Explanation

The technology of machining conducted on lathes compared to milling is considered to be more productive in chip removal today. It is due to better facilities – lathe centers are equipped with many units and driven tools, so you can easily turn, mill, drill and bore in one clamping or even change a position of clamping with two spindles without stopping the machine. However, that machining cannot be called as „simple“.

The NC (Numerical Control) - has been changed in many ways up-today. First, the recording means and ways of transport, downloads (from the punched cards, punched tapes, floppy-disks, discs, DNC communications, flash-disks to the PC-net or internet connections with server integrations). The abbreviation „NC“ is linked either for sorts of machines or machine control (look at the history mentioned earlier). The main use today is a control of a machine via coded information as the alphanumerical chains and symbols. The programme lines (blocks) consist from words which are converted to electrical impulses or other outcomes for activation of servomotors or other drives for machine work. On contrary to the conventional machines they are not affected with the human factor, but depends on the quality of the NC programme solely. The human factor can take place in this area, but with a strong impact on the technology, productivity and economy. This is why a big effort should be put to all details of technology and intended production, but finally, the machine can work in semi-automatic or fully automatic regime.

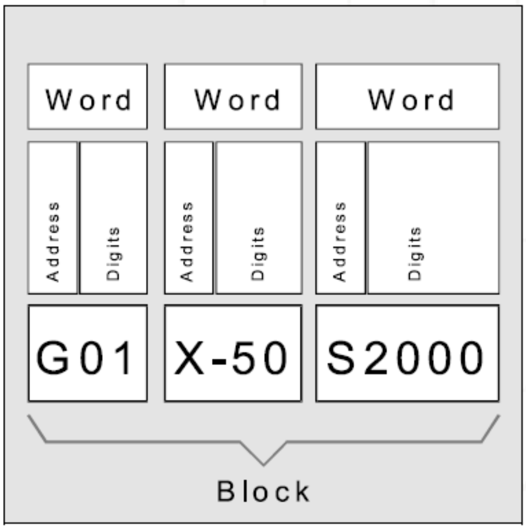


Fig. 2.1 The block format.

The structure and content of the Sinumerik NC programmes is derived from the DIN 66025 standard. These programmes are composed from blocks (lines) and each block includes one programme step. The commands consist from the NC words and there is no fixed word order or sequence. However, the end of programme should be the same – the commands M30, M17 or M2 are the mostly used. Every command or word of the "NC" language consists from the address and the numerical part. The numerical part can involve the sign plus or mines, digitals, decimal point and other digitals. The plus sign is optional and similarly if the rest behind the decimal point equals zero so the decimal point can be omitted. A block (Fig. 2.1) must contain all necessary information for one motion or part-work. Some information that should be valid through more lines of a programme is defined as modal.

The total length of a block in Sinumerik can be extended up to 512 characters. There is a recommended sequence of instructions according to the standards, but it is only for easy orientation. The general form of a block can be written in the sequence:

N... G... X... Y... Z... F... S... T... D... M... H...

Words	Explanation
N	Address of block number
10	Block number
G	Preparatory function
X,Y,Z	Positional data
F	Feed
S	Speed
T	Tool
D	Tool offset number
M	Miscellaneous function
H	Auxiliary function

If the addresses have a logical meaning, they may be repeated more times in one block. Manual programming can be risky because of some misprints and mistakes, but it is a history today. As the great amenity many systems today contain syntax inspection and advanced graphical simulations. The simulation can protect powerfully the tools, work-pieces, machines and operators.

Questions

1. Can you define the general ISO block format (G-code programming)?
2. What does the function DIAMON mean?

Tasks

1. Write one programme block for a cutting tool motion from the actual position (X = 70 mm; Z = 10 mm) to the new position (X = 60 mm; Z = -20 mm) along a straight line running with fast feed speed.

Key

Solution of the task number 1:

G0 X60 Z-20

Solution of the task number 2:

Programming in diameters is active.

3. ISO programming of machining with turning cycles use (for Sinumerik 840D)

Time consumption: 3 hours

Explanation

NC programming of manufacturing technologies can be supported with effective use of so called cycles or canned cycles. The cycles contain all parameters/variables for production of technological operations and all needed motions and paths are automatically generated.

The use of the canned cycles also eliminates the human errors.

An overview of turning cycles:

Cycle	**Explanation**
CYCLE93	Grooving cycle
CYCLE 94	Undercut cycle type E and F (according to the DIN)
CYCLE 95	Stock removal cycle
CYCLE 950	Extended stock removal
CYCLE 96	Thread undercut A, B, C and D (DIN)
CYCLE 97	Thread cutting
CYCLE 98	Thread chaining

G-functions and programme frames, that had been valid before the cycle call stays valid after finishing of the cycle. The working plane should be defined before the cycle calling (the plane G18 for turning - plane „ZX“) and the designation is following: the longitudinal axis (the first axis of the plane), and the cross-axis of the plane (orthogonal to the first one). When using the function of diameter programming, the second axis is automatically regarded as the cross axis. The turning cycles in Sinumerik are designed for the active spindle. If the machine has more spindles, one spindle must be defined as the controlling one.

Many turning and other canned cycles are inspecting motions and intended cutting. If a contour violence occurs, the cycle is interrupted and a warning appears on the screen. For example – when turning tapered surfaces, so-called free angle (the angle between the minor cutting edge and machined surface). The angle of minor cutting edge can be set in the range of 0° and 90°.

Tasks

1. Explain all parameters of the fundamental turning CYCLE95.

2. Highlight the difference between the turning cycles *Thread cutting* and *Thread chaining*.

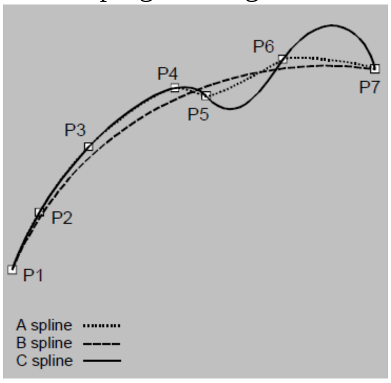
4. Advanced methods of programming (parametrical programming, spline interpolation, special functions)

Time consumption: 3 hours

Explanation

Many attractive surfaces are very different compared to the elementary entities. The reason can be found in a design or in a function of the surfaces. These surfaces can be machined as a matrix of points or a cloud of points from point to point. However, if a mathematical function or any other exact formulation can be defined, the NC programming can be very effective.

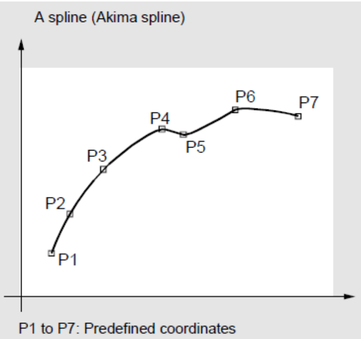
The ISO programming can be extended with so-called higher mathematics and used for turning, milling and drilling.



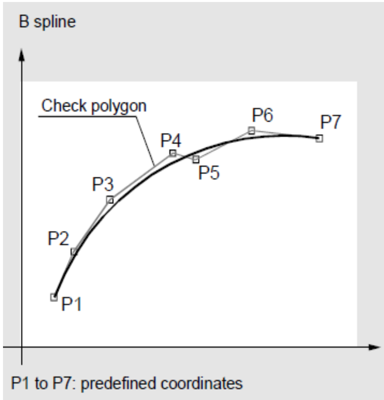
The spline interpolation function can be used to link series of points along smooth curves. Splines can be applied, for example, to create curves using a sequence of digitized points. There are several types of spline with different characteristics, each producing different interpolation effects. In addition to selecting the spline type, the user can also manipulate a range of differentparameters. Several attempts are normally required to obtain the desired pattern. Programming is easy:

ASPLINEX Y Z A B C or
BSPLINE X Y Z A B C or
CSPLINE X Y Z A B C

A-SPLINE



This curve (Akima – spline) is generated as interpolation of points acquired from digitalization of a surface. The spline interpolation function can be used to link series of points along smooth curves. Splines can be applied, for example, to create curves using a sequence of digitized points. There are several types of spline with different characteristics, each producing different interpolation effects. In addition to selecting the spline type, the user can also manipulate a range of different parameters. Several attempts are normally required to obtain the desired pattern. The A spline (Akima spline) passes exactly through the intermediate points. While it produces virtually no undesirable oscillations, it does not create a continuous curve in the interpolation points. The Akima spline is local, i.e. a change to an interpolation point affects only up to 6 adjacent points. The primary application for this spline type is therefore the interpolation of digitized points. Supplementary conditions can be programmed for Akima splines (see below for more information). A polynomial of third degree is used for interpolation.



B-SPLINE

With a B spline, the programmed positions are not intermediate points, but merely check points of the spline, i.e. the curve is "drawn towards" the points, but does not pass directly through them. The lines linking the points form the check polygon of the spline. B splines are the optimum means for defining tool paths on sculptured surfaces. Their primary purpose is to act as the interface to CAD systems. A third degree B spline does not produce any oscillations in spite of its continuously curved transitions. Programmed supplementary conditions (please see below for more information) have no effect on B splines. The B spline is always tangential to the check polygon at its start and end points.

Point weight:

A weight can be programmed for every interpolation point.

Programming:

PW = n

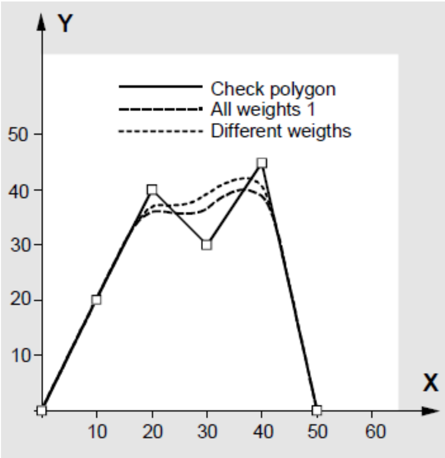
Value range:

0 <= n <= 3; in steps of 0.0001

Effect:

n > 1 The check point exerts more "force" on the curve

n < 1 The check point exerts less "force" on the curve



Spline degree:

A third degree polygon is used as standard, but a second degree polygon is also possible.

Programming:

SD = 2

Distance between nodes:

Node distances are appropriately calculated internally in the control, but the system is also capable of processing user-programmed node distances.

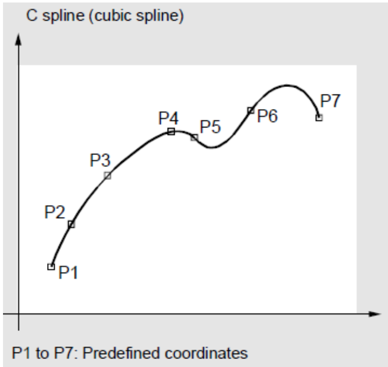
Programming:

PL = Value range as for path dimension

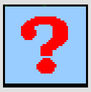
Example of B spline:

All weights 1	Different weights	Check polygon
N10 G1 X0 Y0 F300 G64	N10 G1 X0 Y0 F300 G64	N10 G1 X0 Y0 F300 G64
N20 BSPLINE	N20 BSPLINE	N20 ;omitted
N30 X10 Y20	N30 X10 Y20 PW=2	N30 X10 Y20
N40 X20 Y40	N40 X20 Y40	N40 X20 Y40
N50 X30 Y30	N50 X30 Y30 PW=0.5	N50 X30 Y30
N60 X40 Y45	N60 X40 Y45	N60 X40 Y45
N70 X50 Y0	N70 X50 Y0	N70 X50 Y0

C-SPLINE



In contrast to the Akima spine, the cubic spline is continuously curved in the intermediate points. It tends to have unexpected fluctuations however. It can be used in cases where the interpolation points lie along an analytically calculated curve. C splines use third degree polynomials. The spline is not local, i.e. changes to an interpolation point can influence a large number of blocks (with gradually decreasing effect).



Question

1. Define the controlling points (nodes) for B-spline.

5. Fundamentals of workshop oriented programming - Sinumerik control system (ShopTurn) for modern CNC lathe centres

Time consumption: 3 hours

Explanation

Previously, the NC production normally involved complicated, abstract-coded NC programs. That work that could be performed by the specialists only. However, every skilled worker has learnt his craft and with his experience in the area of conventional cutting, is capable at any time of mastering even the most difficult tasks–even when the economics often suffered. These skilled workers needed to be given the possibility to use this knowledge efficiently with the help of CNC machine tools.

This is why with workshop oriented programming adopts a new path that saves the skilled worker from having to do any coding. WOP gives these skilled workers a new generation of SINUMERIK control:

- a work schedule rather than programming is the solution.
- tThe creation of this work schedule with easy-to-follow, skilled-worker-oriented handling sequences allows the ShopTurn user to make use of his actual knowledge, namely his know-how, for the cutting.
- ShopTurn, with its integrated, powerful traverse path creation, allows even the most complicated contours and workpieces to be produced without difficulty. Consequently:
- Simpler and faster from the drawing to the workpiece with filling-in texts according to the figures and charts – so the WOP - ShopTurn is very easy to learn indeed

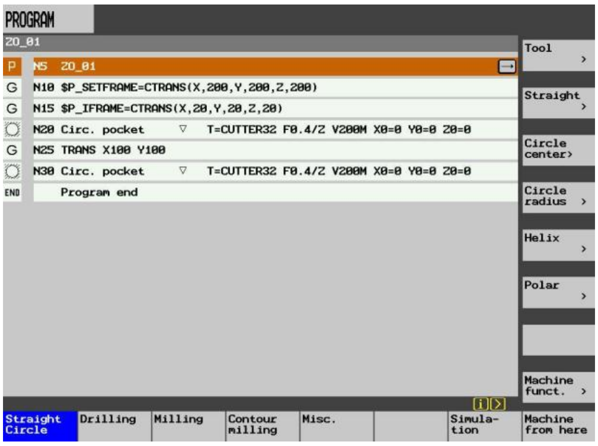


Fig. 5.1 ShopTurn Editor and workshop oriented NC programme.

Tasks

1. Can you explain the groove turning that is included in the programming editor Sinumerik?

2. Mark the motions of a cutting tool when turning with the technology of groove turning.

6. Description, technical data and controlling of CNC milling machines, fundamentals of 5-axis milling centre programming (MCV 1210)

Time consumption: 3 hours

Explanation

Vertical machining centre MCV 1210 (TAJMAC-ZPS, a.s., www.tajmac-zps.cz) is determined for production of moulds in the pressing, plastics, automotive and aeronautics industry with particular focus on machining of spatially complicated shapes in 3D, with utilization of both three-axis and five-axis machining. In the field of conventional production, the machine is capable of milling, drilling, counterboring and reaming of holes, thread cutting and thread milling with highly productive machining times.

The upper-gantry type portal, whose frame consists of two sidewalls fixed to the base, forms machine construction. Bolted down to the base upper faces are linear guide-ways for the cross rail travel. There also are the ball screws of the Y-axis driving mechanism and the measuring rulers mounted. One face of the cross rail is provided with the guide-ways for the travel of the protruding cross slide in the X-axis, in the middle of the face is the ball screw of driving mechanism having the minimum overhung.

Inside the cross slide are fastened two pairs of carriages of linear guide-ways for the slide ram shifting out in the Z-axis. Due to the high dynamics, high rigidity and damping capacity of its construction, the machine enables an effective use of the HSC/HFC technologies and other advantages.

Fig. 6.1 5-axis machining centre MCV 1210 with the Sinumerik 840D pl

Questions

1. What kind of the 5-axis machining is applied to the vertical milling centre MCV1210? (Where are the rotational axes?)

2. Can the machine MCV 1210 be used for HSC milling? (What is the extent of rotations and power?)

7. Fundamentals of ISO programming – milling technology

Time consumption: 3 hours

Explanation

The technology of milling emerges from all manufacturing technologies as the most important one. The application to the CNC machines was so intensive because of high versality of the cutting with multiple-axis for hole production, cavities, surfaces, etc that used to be machined by turning, shaping or boring (e.g. rotational molds). The main milling technologies:
:

- face milling,
- shoulder milling,
- copying (contour) milling,
- cavity milling,
- circular milling,
- rotational milling,
- milling of threads,
- cutting-off,
- high feed milling,
- plunge milling,
- counter sinking,
- linear and circular interpolation,
- hobbing,
- etc.,

This distribution is based on the effect of cutting and force loading of tool or cutting paths along workpiece. A milling is a machine tool used to machine solid materials. Milling machines are often classed in two basic forms, horizontal and vertical, which refers to the orientation of the main spindle. Both types range in size from small, bench-mounted devices to room-sized machines. Unlike a drill press, which holds the workpiece stationary as the drill moves axially to penetrate the material, milling machines also move the workpiece radially against the rotating milling cutter, which cuts on its sides as well as its tip. Workpiece and cutter movement are precisely controlled to less than 0.001 in (0.025 mm), usually by means of precision ground slides and lead-screws or analogous technology. Milling machines may be manually operated, mechanically automated, or digitally automated via computer numerical control (CNC).

Milling machines can perform a vast number of operations, from simple (e.g., slot and keyway cutting, planing, drilling) to complex (e.g., contouring, diesinking). Cutting fluid is often pumped to the cutting site to cool and lubricate the cut and to wash away the resulting swarf.

Questions

1. Is it possible to use sub-programmes for CNC?

Tasks

1. Explain the programming of function G18 for turning and milling.

8. Fundamentals of ISO programming – milling technology (Sinumerik 840D, Heidenhain iTNC530)

Time consumption: 3 hours

Explanation

The ISO programming is regarded as the „gold standard“ for NC and even today can be seen as fundamental and basic for reliable programming. A very effective help can be found in the use of special cycles that simplifies the programming and cuts down the time for calculations. A selection of typical canned cycles is mentioned bellow (systems Sinumerik 840D and Heidenhain iTNC530) and will be analyzed lately in the subchapters.

Cycles for milling – Sinumerik 840D

Oblong holes placed on circle arc	LONGHOLE
Grooves on circle arc	SLOT1
Circumferential grooves	SLOT2
Milling of rectangular cavity (pocket milling)	POCKET1
Milling of circumferential cavity (pocket milling)	POCKET2
Milling of threads	CYCLE90
Pocket milling	POCKET3
Milling of circumferential pocket	POCKET4
Plane milling	CYCLE71
Contour milling	CYCLE72
Pocket milling with island	CYCLE73
Forwarded edge contour of a pocket	CYCLE74
Forwarded island contour	CYCLE75
Rectangular pigeon milling	CYCLE76
Circular pigeon milling	CYCLE77

Cycles for milling – Heidenhain iTNC530

RECTANGULAR POCKET	Cycle 251
CIRCULAR POCKET	Cycle 252
SLOT MILLING	Cycle 253
CIRCULAR SLOT	Cycle 254
POCKET FINISHING	Cycle 212
ISLAND FINISHING	Cycle 213
CIRCULAR POCKET FINISHING	Cycle 214
CIRCULAR STUD FINISHING	Cycle 215
SLOT (oblong hole) with reciprocating plungecut	Cycle 210
CIRCULAR SLOT (oblong hole) with reciprocating plunge-cut	Cycle 211
Special cycles	Cycle 32 (TOLERANCE)

9. Fundamentals of workshop oriented programming for modern CNC milling centres (Sinumerik, ShopMill)

Time consumption: 3 hours

Explanation

Modern milling centers are used dominantly for the continuous 5-axis machining (with 3 translation axes, 2 rotational axes), but very frequently are equipped with so-called workshop oriented programming also. The main advantages lies in an integration of the manual programming and import of external big files for machining of shaped parts that had been generated with external computers and advanced SW in the vector form (cinematically independent).

Another advantage can be seen in the special cycles that ensure rather complicated cutting tool motions (e.g. trochoid milling that can work effectively even in the noisy workshops (Fig. 9.1).

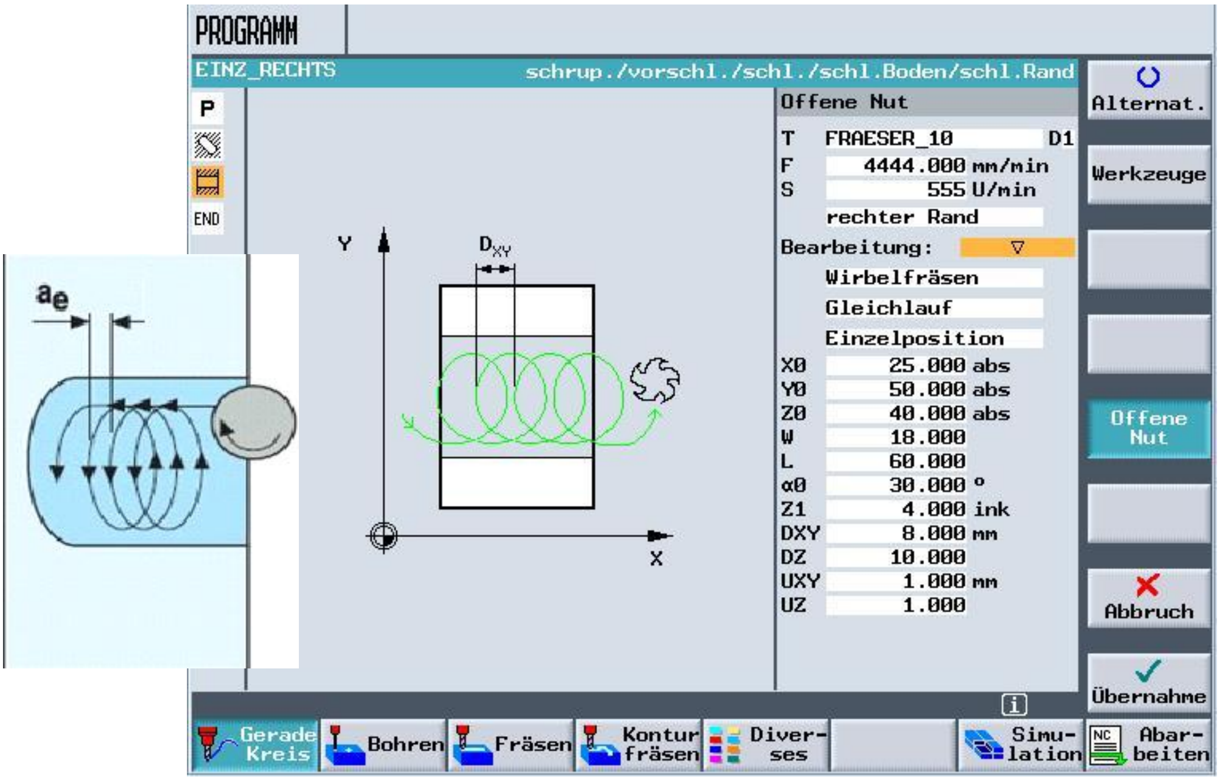


Fig. 9.1 ShopMill system for production of „deep“ grooves.

Questions

1. Is it possible to program so-called *plunge milling* directly from a machine operational panel?

10. Measurement with contact and contact-less probes for milling operations

Time consumption: 3 hours

Explanation

Apart of a tool calibration also a calibration of the touch probe OMP 400 (work-piece probe, calibration of a diameter or length) can be done - in the regime JOG (manual regime of a machining centre). Typically, measurement of all workpiece surfaces, coordinates, angles is very effective directly at the machine. A very common application is the setting of the zero-point (beginning of the coordinate system – at the workpiece corner - Fig. 10.1). The cycles are based on measurement of a point, straight line, plane or even semi-automatic measurement in the holes (rectangular pockets), cylindrical pigeons, etc. The inspected surfaces can be measured in one point or in two points and it is very useful for setting of origins at the corner without perpendicular walls. Kinematics of a machine can be measured also, but some measures should be accepted carefully

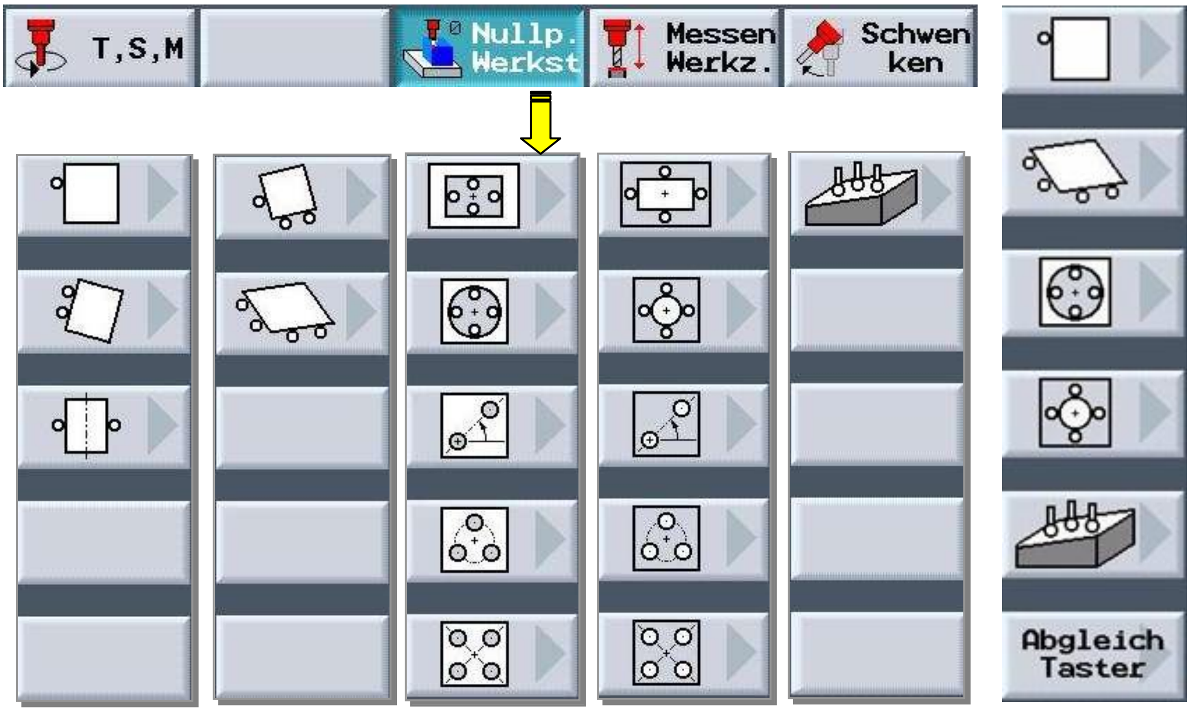


Fig. 10.1 Horizontal and vertical icon menu of ShopMill editor (measurement).

Questions

1. What kind of contact-less communication can be used with the probe OMP400?
(wire-less, inductive, optical)

11. Fundamentals of CAD/CAM technologies (3D modelling, programming in CAM environment, postprocessing, remote diagnostics, simulation in the control system Sinumerik

Time consumption: 3 hours

Explanation

The CAD/CAM technologies includes the CAD (Computer Aided Design) softwares (e.g. SolidWorks, PowerSHAPE, atd.) for a design (3D models of work-pieces), and the CAM (Computer Aided Machining, e.g. PowerMILL, FeatureCAM, atd.) for cutting paths generations.
CAD environment can include many tools for kinematics, stress-strain analyses and graphical visualization or mathematical verification of cutting. The CAD output is a model, the CAM output is a code that should be transformed into CNC code (post-processing) and format.
A transport of the NC programme can be assured with flash disc or special interchange SW. The system Heidenhain uses the software TNCRemo, Sinumerik uses the pcAnywhere with other remote diagnostics. The simulation of the NC programme follows in the standard or cinematically independent (vector) form in the control system or a PC. Another facility is the direct measurement – e.g. software Productivity+ that presents a product independent to the CNC system or CAM.
A special postprocessor allows programming of the measurement cycles with probes with data outputs.

Software Productivity+ can compare the imported graphical 3D model designed in the CAD system (and add the measured points. Furthermore, a zero offset or a transformation, measurement of tool compensations and their full- inspection can be measured, stored, copied and used in many ways as text-files. All the handling is very easy with a mouse-click only. A sequence of data measurement can be optimized.

Questions

1. Is it feasible to import a 3D graphical model with the software Productivity+? Is it possible to enter positional coordinates from the keyboard?

2. Can be a point, circle or curve drawn with the CAM software PowerMILL for following machining?