HEIDENHAIN

Klartext

News from the World of HEIDENHAIN Controls

Issue 47 + 9/2007

Editorial

Linear encoders improve the machining accuracy

Page 4

Dear Klartext Reader,

The motto for this year's stand at EMO is "HEIDENHAIN shows the way to precision." Demonstration units and presentations will show the advantages of machine tools equipped with linear encoders.

Real workpieces will give provide tangible evidence of the differences that occur on machines with and without linear encoders.

Of course, once again many product innovations will be presented. For example, HEIDENHAIN will in the future connect its control components using the purely digital real-time Ethernet bus HSCI. This makes the entire system thoroughly diagnosable and more available. The new TNC 620, which was conceived for the mid-range processing power segment, and the well-known, field-proven iTNC 530 control to tackle the high-end applications, with HSCI and EnDat 2.2 will be featured.

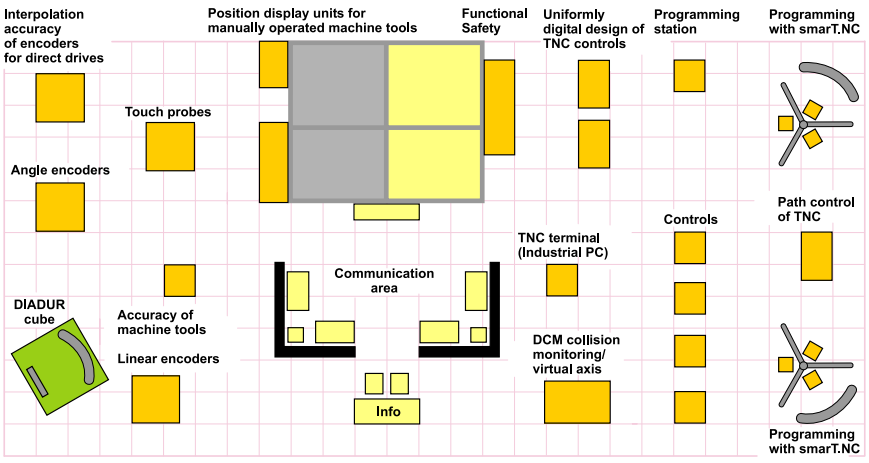
The MANUALplus 620 is a new control: it was conceived both for CNC and cycle lathes. The CNC PILOT 4290 lathe control with B axis makes it possible to drill, bore and mill in oblique planes.

With its TS 740, HEIDENHAIN will present a high-accuracy infrared touch probe for very demanding 3-D measuring tasks. The new TS 444 is the first battery-free infrared touch probe system.

So read and enjoy! We are convinced you'll be glad you did.

See us at the EMO 2007

Hannover
Sep. 17-22, 2007
Hall 25
Booth E18



The new TNC 620 adds a compact control with digital servo drive control to the range of products from HEIDENHAIN

Page 12

With the MANUALplus 620, HEIDENHAIN presents a new control, conceived for both CNC and cycle lathes

Page 18

Production

Publisher

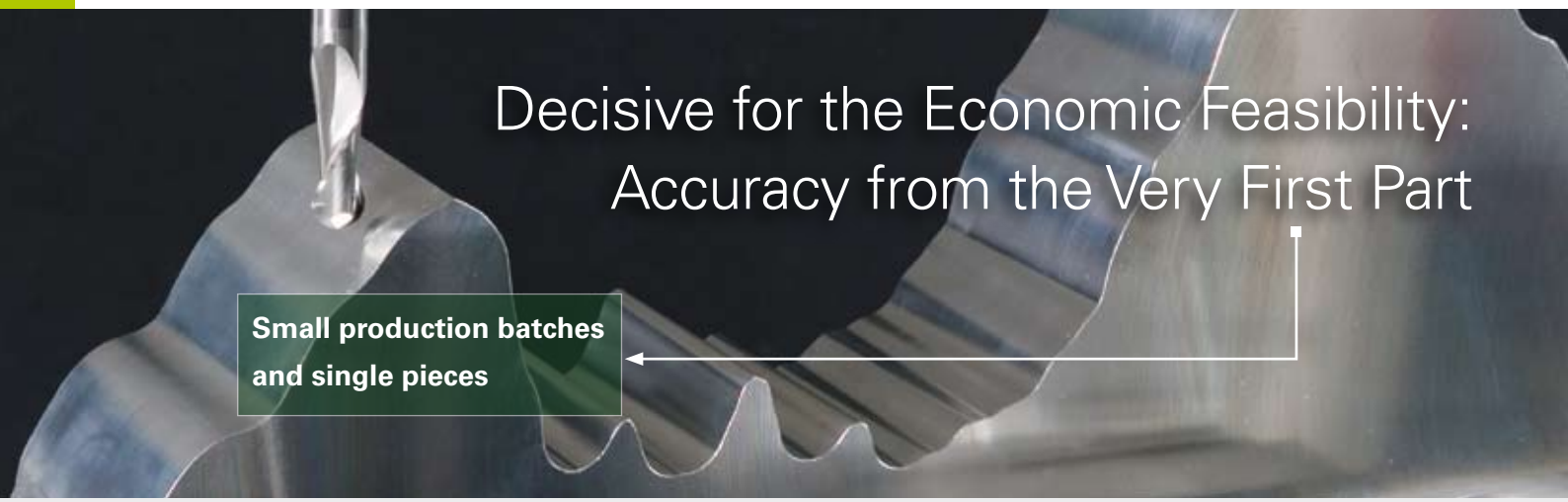
DR. JOHANNES HEIDENHAIN GmbH
Postfach 1260
83292 Traunreut, Germany
Tel: +49/86 69/31- 0
HEIDENHAIN on the Internet:
www.heidenhain.de

Editor

Frank Muthmann
Fax: +49/86 69/31-18 88
E-mail: info@heidenhain.de
Klartext on the Internet:
www.heidenhain.de

Layout and Typesetting

Expert Communication GmbH
Richard-Reitzner-Allee 1
85540 Haar, Germany
Tel: +49/89/66 63 75 0
E-mail: info@expert-communication.de
www.expert-communication.de



Decisive for the Economic Feasibility: Accuracy from the Very First Part

Small production batches and single pieces

The largest share of thermally-related inaccuracy on machine tools is produced in most cases in the feed drives. High speeds and acceleration levels heat up the ball screws and cause them to expand. Without suitable position measuring technology, this can lead to positioning errors of up to 100 µm within a few minutes. However, workpieces with tight tolerances can only be produced on machine tools that remain thermally stable despite very different machining operations.

Position measurement of feed drives

The position of a linear NC axis can be measured in principle through the pitch of the feed screw connected with a rotary encoder or through a linear encoder.

In the case of the feed screw/rotary encoder, the ball screw has a double function: as a drive system it must transfer high forces, but for position specification a high level of accuracy and repeatability of the screw pitch is required.

However, the control loop for position specification includes only the rotary encoder, which sends signals indicating the feed screw's rotational speed and subdividing each revolution. Wear and temperature-related changes in the drive mechanics are not factored into the position measurement in this case.

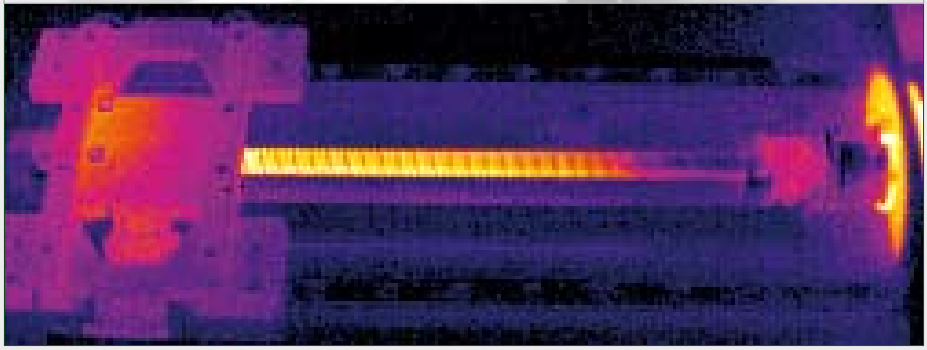
Positioning errors of the drives become unavoidable and can have a considerable influence on the quality of workpieces.

If a linear encoder is used for measurement of the slide position, the position control loop includes the complete feed mechanics. Play and inaccuracies in the transfer elements of the machine have no influence in this case on the accuracy of the position measurement. Measurement accuracy depends almost solely on the precision and installation location of the linear encoder.

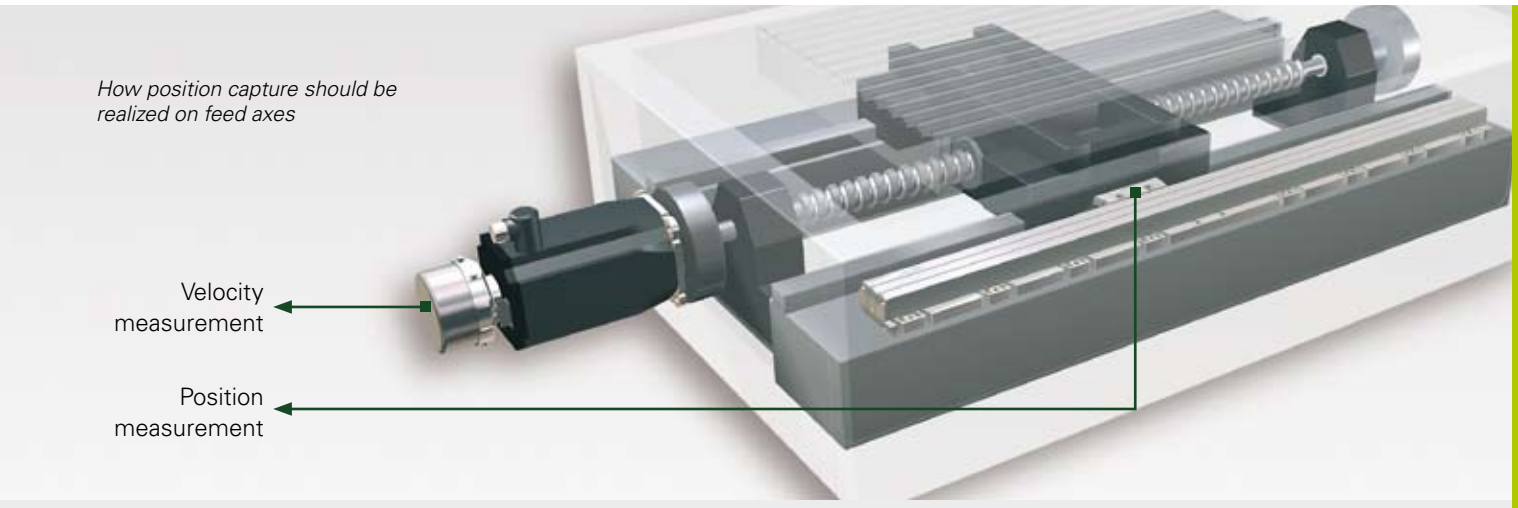
Example of machining an integral component

Typical integral components are machined on high-performance HSC machine tools in conjunction with high feed rates and high cutting speeds. Different feed rates during roughing and finishing lead to continually changing thermal expansion factors of the ball screws. If the feed drives are operated without linear encoders, part dimensions differ for each manufactured single component in small production quantities with short door-to-door times. The danger therefore exists that thermal expansion will prevent the specified manufacturing tolerances from being achieved.

Linear encoders from HEIDENHAIN improve the machining accuracy



The recirculating ball screw heats up when face milling at 10 m/min. At left is the table, at right the servo motor. The thermographic image shows temperatures of 25 °C (dark blue) to 40 °C (yellow).



How position capture should be realized on feed axes

Such sources of errors can be prevented through the use of linear encoders, which enable thermal expansion of the ball screws to be fully compensated.

The test shown in the figures clearly illustrates the thermal errors of machining without linear encoders.

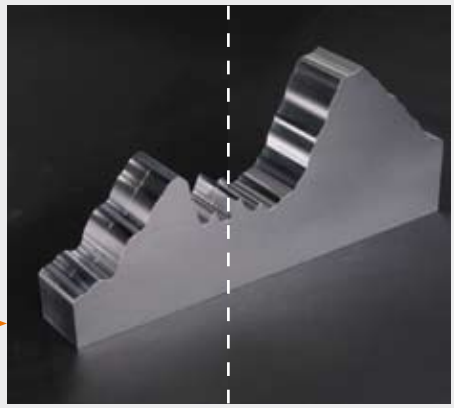
A coupling lever from aviation technology is milled from aluminum to a depth of 10 mm. After 20 air cycles above the workpiece, the lower part of the lever is milled. The thermal drift of the feed axis is visible by an edge on the side of the workpiece. If the machine has linear encoders, then no edge results in this test.

A reproducible accuracy from the very first unit is guaranteed!

Coupling lever, processed twice from the same blank form
Without linear encoders (left): *Thermal drift is visible by the offset on the edge*
With linear encoders (right): *No thermal drift visible*

Effects on mold and die making

Mold and die making for milling places high demands on the form accuracy. At the same time, high feed rates are necessary in order to shorten machining times. The first and last milling paths must match, otherwise the time previously gained would be lost due to extensive reworking. The example shows a form machined to represent the profile of the Watzmann mountain. In order to visualize the linear deviation resulting on this mold component from operation without linear encoders, machining was deliberately begun in the middle of the workpiece. Start and end paths therefore lie side by side, and the edge clearly shows the thermal drift. If a machine tool with linear encoders is used, then the Watzmann profile does not show this edge.



Watzmann profile with free-formed surfaces: machined on the left without linear encoders, on the right with linear encoders

Conclusion:

Production orders are completed especially successfully if the machine tools used feature a high degree of thermal stability. In this case the feed axes must achieve the required accuracy over the complete traverse range even with strongly varying speeds and machining forces. These targets can be met by using linear encoders on machine tools.

At the **HEIDENHAIN stand in Hall 25, Booth E18,** you can see and feel the effects of such machining on the actual workpieces described in this article.

New Innovative Functions for the iTNC 530

Dynamic Collision Monitoring
Adaptive Feed Control
DXF Converter
KinematicsOpt
KinematicsDesign

The NC software 340 49x-04 for the iTNC 530 includes a series of new functions for machine manufacturers and users. These functions make it even easier to work with the control, and they also make operation of the machine more safe.

Dynamic Collision Monitoring (DCM)

Dynamic **C**ollison **M**onitoring (DCM) helps prevent damage to the machine and workpieces. Although NC programs created from CAD/CAM systems avoid collisions between the tool and workpiece, the machine components in the work envelope are not taken into consideration.

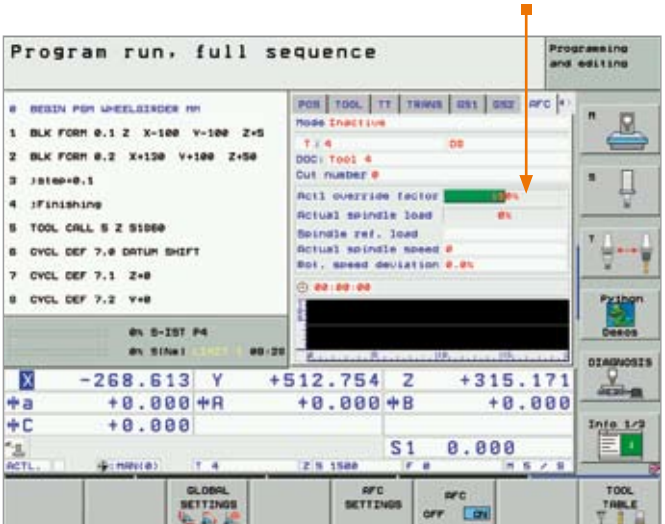
This is where HEIDENHAIN comes into play, and makes the work envelope defined by the machine manufacturer visible to the control. The operator can see machine components in danger of collision on the screen, and then move them out of the collision area. The adaptable settings for the split screen layout are new. For example, the program blocks can be displayed in one window, and the work envelope in the other window.

If a collision is imminent, the control interrupts the automated machining.

Adaptive Feed Control (AFC)

Adaptive **F**eed **C**ontrol (AFC) optimizes the contouring feed rate depending on the performance of the tool spindle and other process data.

The dynamic line diagram in the status window is new. It shows how the contouring feed rate and spindle performance relate to another.



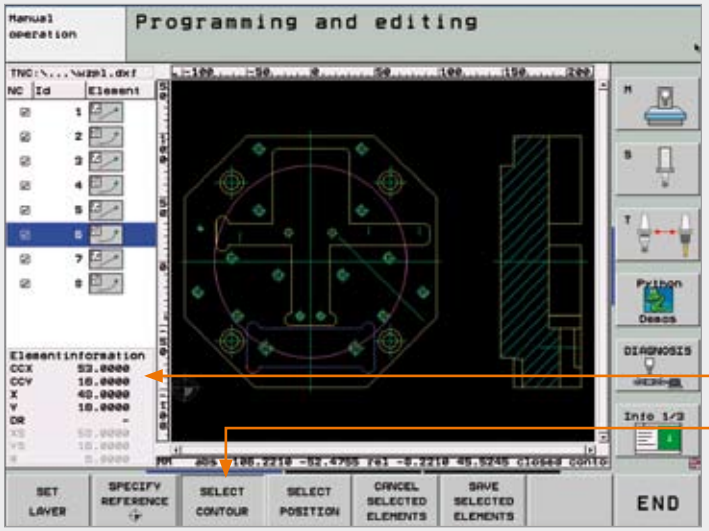
During the learning phase, the TNC shows the currently saved reference power in a pop-up window. If required, you can use a soft key to reset the reference power measured up to that point and restart the learning process.

DXF converter (option)

You can use the DXF converter to open CAD data in DXF format directly on the iTNC 530 in order to extract contours. Convenient extraction of the contour not only saves time otherwise spent on programming and testing, but you can also be sure that the finished contour is exactly according to the designer's specifications.

Handling has been improved significantly in the new version:

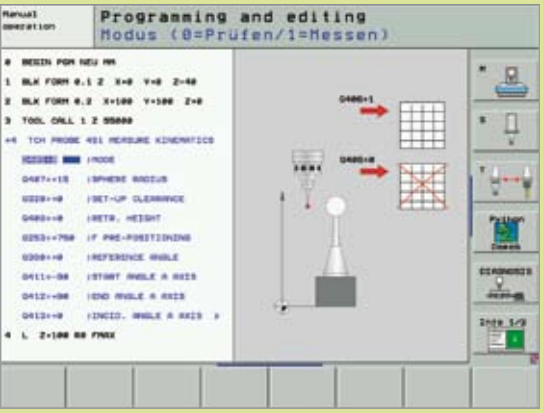
- Zoom settings of the last selected DXF file are saved.
- Datum set for the the last selected DXF file is saved.
- Circle center points can now be assumed directly.



The new info box, showing all data of the selected element, is very helpful. For machining positions you see the X/Y coordinates, for contour elements the start and end points, and for the circles the center point and direction of rotation are shown as well.

KinematicsOpt (option)

Accuracy demands are always a hot topic, especially for 5-axis machining. Machining of complex workpieces requires complex tool movements, which must be performed with a high degree of precision. The new **KinematicsOpt** feature ensures reproducible accuracy, even over long periods of time. This guarantees a high level of quality for series production.



New KinematicsOpt function: Automated measurement of kinematics and automatic recalibration of machine kinematics

The principle:

- The rotary axes are measured completely automatedly with a 3-D touch probe:

  A 3-D touch probe cycle measures the rotary axes on your machine completely automatedly, regardless of whether the rotary axes are present as tables or heads. A calibration ball is fixed at any position on the machine table, and measured with a resolution that you define. You define for each rotary axis the area that you want to measure.

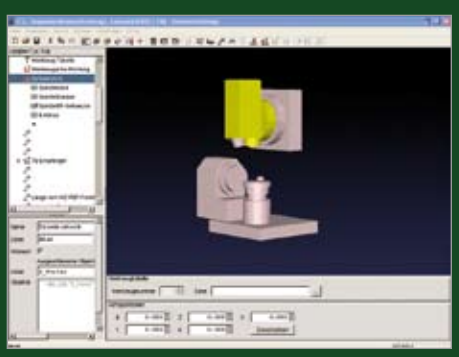
- The iTNC 530 determines the statistical tilting accuracy from this.
- The spatial error resulting from the tilting motions is minimized.
- The machine geometry is saved in a kinematics table.

Of course, a comprehensive log file is also saved with the actual measured values and the measured and optimized dispersion (measure for the statistical tilting accuracy), as well as the actual compensation values.

PC software KinematicsDesign

Now the machine manufacturers, too, can develop kinematics tables more quickly. With **KinematicsDesign,** the user now has a PC tool that gives him graphic support when defining kinematics tables.

KinematicsDesign makes it possible to simulate critical axis positions during the conception phase and avoid them by setting limit switches at the right positions on the machine.



New KinematicsDesign PC tool: Develop and manage kinematics tables

Global program settings (option)

User-friendly improvements have been made to the global program settings function.

What to do if large, externally created NC programs need to be modified?

You define coordinate transformations and settings that have a global effect and are superimposed on the selected part program. This way the actual NC program does not need to be changed.

Along with datum shifts, rotations and mirror images, axes can be switched or disabled and handwheel superimpositions can be set.

A new feature is the activation of the virtual axis (VT)

For example, if you want to run an entire part program with a constant oversize, you can use the handwheel to move the tool in the currently active tool axis direction (if Tool Center Point Management (TCPM) is active).

Use the HR 420 handwheel: then you can select the virtual axis (VT) directly via the handwheel soft keys. You can see the value of the distance moved in the virtual axis direction in the handwheel's display.

For handwheels without integrated position display: here you select the virtual axis with a machine key defined by the machine manufacturer. The distance moved is shown in a separate position display (and also in the global program settings form). The value remains stored until you change the tool or switch the function off.

The global program settings come into play especially in large-scale mold making.

3-D basic rotation, machine-specific (upgrade function)

This function can be used to correct any workpiece misalignment in three dimensions (3-D set-up compensation).

Prerequisites:

- Your machine must have at least two rotary axes.
- Your machine manufacturer must adapt this function specifically to your machine.

New function: Generation of service files

Good error logs are often needed in case of errors or uncertainties. Now there is a function that collects all important data in a ZIP file.

The ZIP file contains:

- the active NC program
- the tool table TOOL.T
- any active datum tables
- important system files

Download the ZIP file through one of the data interfaces, and e-mail it to your machine manufacturer or the HEIDENHAIN service department. Rapid assistance will be available that much sooner.

New conversational languages (option)

Starting immediately, Turkish and Romanian are available as conversational languages.

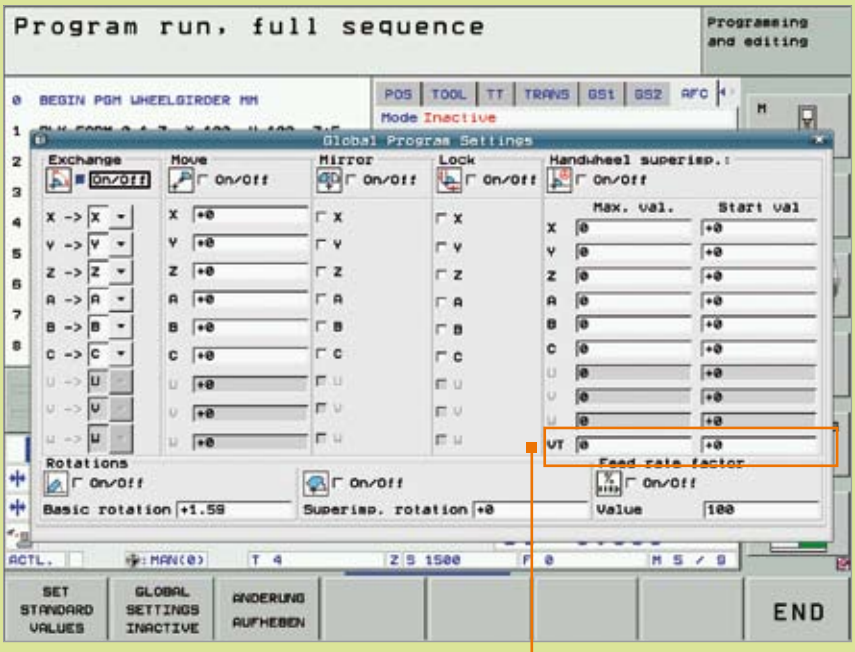
New file management

Are you already familiar with the file management in **smarT.NC,** which can now be operated entirely with the mouse, as well as by soft key?

File management for conversational programming now works identically.

Further highlights:

- Files can now be sorted by name, type, size, date of change and status.
- Favorites can be managed.
- Files are selected when the first letter of the file name is entered on the keyboard.
- The display of file information can be configured now.
- The date format can be configured now.



Display of values in the virtual axis (VT)

Hardware requirements for the new 340 49x-04 software for the iTNC 530

- MC 422 B or C main computer
- 512 MB RAM

New Functions for Plain-Language Programming

New: Pattern definition

The point-pattern generator function already known from **smarT.NC** is now also available for plain-language programming.

New PATTERN DEF function:

Definition of the machining pattern:

- Points (up to 9 individual positions)
- Row
- Frame
- Area
- Circular arc
- Full circle

The machining patterns defined in this way can be called with the familiar CYCL CALL PATTERN function.

New: Globally effective cycle parameters

GLOBAL DEF cycles: You can define a wide variety of cycle parameters at the beginning of the program with global effect.
The following groups are available:

- *General cycle parameters such as safety clearance or retraction feed rate*
- *Drilling-specific cycle parameters, such as dwell times*
- *Milling-specific cycle parameters, such as the plunging behavior*
- *Touch-probe-specific cycle parameters, such as clearance height*

In the cycle definition you simply link to the defined values via soft key.

The TNC then enters the word PREDEF (for predefined) in the cycle definition. Any change in the GLOBAL DEF cycle affects all cycles that refer to the PREDEF entry in the respective GLOBAL DEF cycle.

File functions

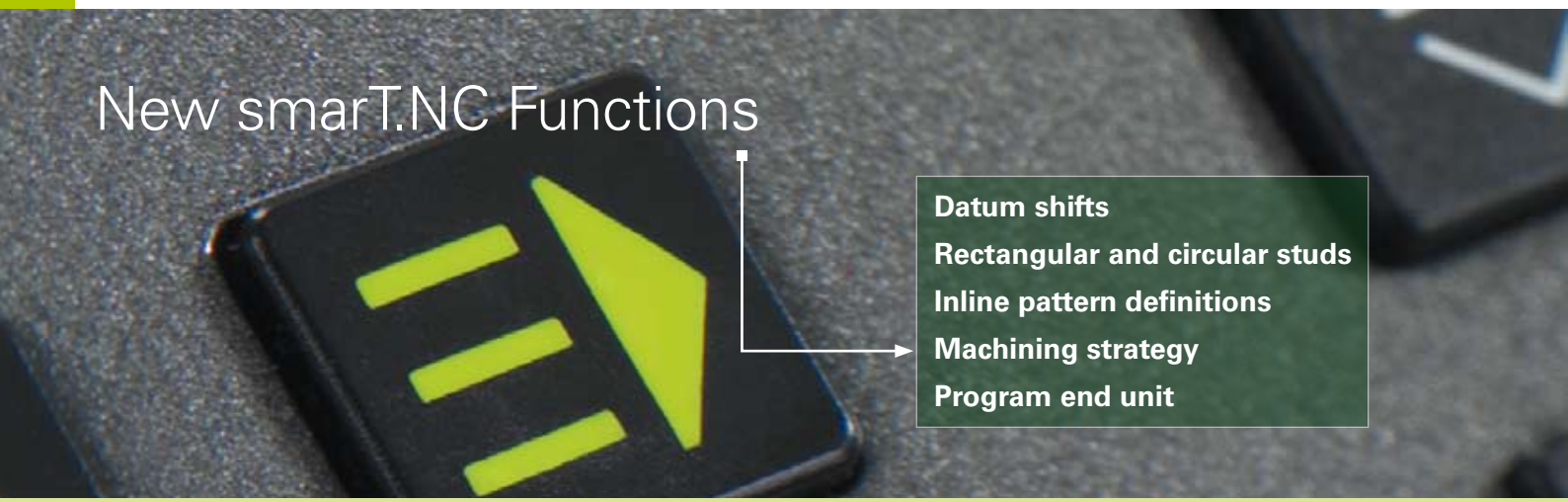
With the FILE FUNCTION feature you can copy, move and delete files from within the part program. This enables you, for example, to copy and start part programs that you have saved on an external drive.

New: Machining rectangular and circular studs

Rectangular and circular studs can now be machined even more easily with the new Cycles 256 and 257. The constant cut distribution is particularly helpful when the difference between the dimensions of the blank and the finished part is greater than the tool radius. Of course, the distribution of cuts can be modified by an overlap factor.

The new cycles are structured similarly to the already existing milling cycles 251 and 254.

New smarT.NC Functions



Datum shifts
Rectangular and circular studs
Inline pattern definitions
Machining strategy
Program end unit

New datum shift

Being able to shift the datum solely via datum tables is now a thing of the past. You can now also define the shifts for specific axes in a form. And resetting is even easier: just press a soft key, and it's done!

New: Machining rectangular and circular studs

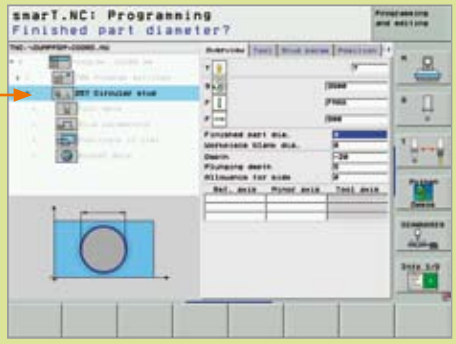
The new Cycles 256 and 257 for plain-language programming are mirrored in the smarT.NC operating mode, with machining units 256 and 257.

Inline pattern definitions revised

Being able to define machining patterns without a pattern generator: that is new, and is done directly in the overview form of a machining unit. Available patterns:

- Points (up to 9 individual positions)
- Row
- Frame
- Area
- Circular arc
- Full circle

smarT.NC operating mode



Loading values from previous units

Repetitions occur frequently, and machining units often only differ in small details. For example, when different tools or oversizes are used for roughing and finishing definitions.

This is now made quite simple with **smarT.NC:** you can use the values defined in the earlier machining unit as default values for the new machining unit (in the same smarT.NC program). This saves you much programming time.

New: Setting the number of probing points for a circle

Measuring circles as previously with 4 probing points, or with just 3?

You have the choice in touch-probe machining units 412, 413, 421 and 422.

New: Defining a machining strategy for clearing

How should the TNC move the tool during clearing?

Choose in machining unit 22:

- Retracing the entire contour
  The TNC moves at constant height to the areas to be cleared without removing the tool from the finished part contour. This strategy works well when the distance between the areas to be cleared is small and the fine roughing tool is large enough to machine the remaining material in one step.

- or machining individual areas separately
  After fine roughing each area to be cleared, the TNC moves the tool at rapid traverse to the safety clearance. This strategy is helpful when there is a large distance between the areas.

Fast retraction during tapping

There are still possibilities for reducing machining time: for example, you can retract at a shaft speed greater by a factor X from a drill hole. You specify this factor in machining unit 209 for tapping.

Also new in smarT.NC: KinematicsOpt (Option)

This new function, the measurement of machine kinematics, already described for plain-language programming, is now available in smarT.NC, via machining units 450 and 451.

New: Program end unit

The following settings can be made in the program end unit:

- Definition of M functions, e.g. M5, M30
- Approach to a safe position in the tool axis
  (either in the workpiece or machine coordinate system)
- Approach to a safe position in the working plane
  (either in the workpiece or machine coordinate system)

200,000 NC Controls from HEIDENHAIN

In the middle of 2007 HEIDENHAIN shipped its 200,000th NC control, and celebrated a new record in its 30-year history of success. With over 30,000 units, the iTNC 530 has succeeded the TNC 426 as HEIDENHAIN's most successful of over 50 models. Over 10,000 units of this model are expected to ship from Traunreut in 2007—numbers that speak for its great acceptance in the NC control market.

These TNC controls have their roots in the workshop, where they banished machinists' reluctance toward the new technology. Today, their user-friendly dialog guidance ensures HEIDENHAIN a high market share in applications that require shop-floor programming.

Plain-language, conversational programming has won its place in the metalworking industry with HEIDENHAIN controls guiding the user with questions and prompts and automatically generating the program. Since 2004, the "smarT.NC" operating mode has offered even more convenience with straightforward fillable forms, interactive graphics and quickly understood user aids.

HEIDENHAIN controls are widely used on machines capable of manufacturing parts of high quality. At the high end—on machining centers and complex milling machine for 5-axis machining—the iTNC 530 stands for short machining times, high contour accuracy and best surface quality. In the mid-range, the TNC 320 provides precision and efficiency in manufacturing on machines with up to four controlled axes. The TNC 124, a well-proven straight-cut control for simple machine tools, meets customers' needs for less demanding control tasks. Controls for lathes such as the MANUALplus 4110 complete the product range.

The TNC 620 – The New Contouring Control from HEIDENHAIN

Workshop oriented, compact and trend-settingly digital

The new TNC 620 adds a compact control with digital servo drive control to the range of products from HEIDENHAIN. HEIDENHAIN already introduced the TNC 320, an analog control for simple 3-axis machines, at the EMO 2005, and it has been proving itself in daily operation ever since. Both controls are based on a new, trend-setting software concept from HEIDENHAIN, and use the same software platform.

Keep learning instead of re-learning

HEIDENHAIN controls undergo continuous improvement, but the basic operational technique remains the same. The motto "keep learning instead of re-learning" is as valid today as it has always been. This basic premise was naturally taken into account for the TNC 620: an experienced TNC programmer will have absolutely no problems with the TNC 620.

Thanks to the workshop oriented programming style with helpful dialogs and graphic support, novices will quickly feel at home when using the new control. The clear structure of the soft keys constantly gives you an overview of the necessary functions, so that you can find them immediately. The keys for initiating typical TNC dialogs are included on the compact keyboard, so that you can quickly access all TNC functions. In addition, the machine manufacturer can make machine-specific functions available in the vertical soft-key row.

Cycles simplify program creation

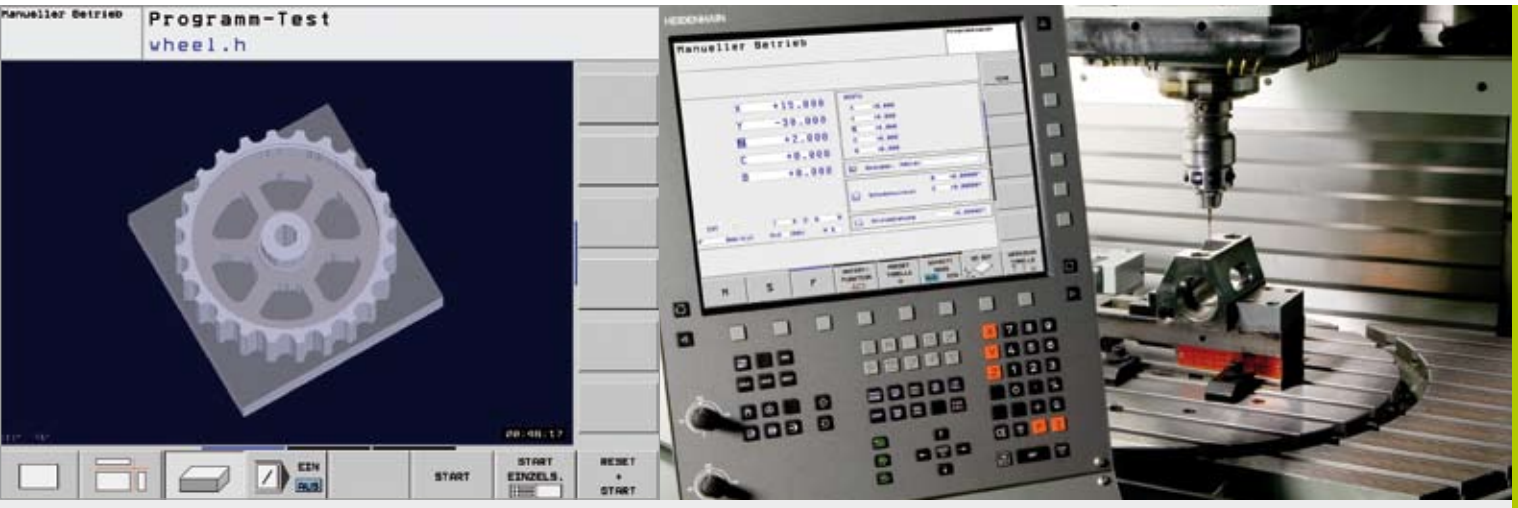
The TNC 620 features numerous cycles designed for most machining tasks that occur on the shop floor. Along with the **machining cycles** for drilling, tapping (with or without floating tap holder), thread cutting, reaming, and boring, there are also cycles for hole patterns (circular and linear). Milling cycles include facing of flat surfaces, as well as roughing and finishing pockets, slots and studs.

Touch-probe cycles, which can easily be integrated into the machining program, are available for automatic measurement and inspection of workpieces. The TNC 620 provides meaningful support graphics and dialog texts when the machine operator enters information for machining or touch-probe cycles.

When creating workshop-oriented programs using the tried-and-true conversational programming language from HEIDENHAIN, the programming graphics interactively show step-by-step what is currently being programmed. This is especially helpful when using the powerful free-contour programming feature to create parts not dimensioned for NC.

Knowing ahead of time— thanks to sophisticated graphics

After completion of an NC program, the test run graphics can give a realistic impression of the finished part before machining begins. The TNC also performs an internal test run to check the NC program for logical errors, even



before the workpiece is placed in the machine. This makes it easy to avoid downtimes. Tips about the cause of error, as well as possibilities for troubleshooting, simplify the search for errors.

Simple handling of complex operations

The TNC 620 is equipped for handling very complicated tasks, even those that also include the use of swivel and rotary axes. For example, the working plane can be tilted around one or more rotary axes. The machining program is then simply created in the main plane (usually X/Y). Special cycles are even available for machining contours, slots or ridges on cylinders as if they were in just two axes.

The TNC 620 also features special functions for simultaneous machining with up to five axes: dynamic advance calculation of the contour, algorithms for jerk limiting, and intelligent path control fulfill the high demands placed on the surface quality of the workpiece.

Select your features

You can determine the scope of your TNC 620's functions to meet your needs and desires. You select various options to configure your control in a manner useful to you, and for what it will need to do in daily operation. But of course this configuration is not carved in stone. If it turns out in the future that you need a function not initially chosen, you can have your machine tool builder activate the function for you.

Hardware design: compact and modern

In the past, the MC main computer and CC controller unit of digital HEIDENHAIN controls were always installed in the electrical cabinet. Now the main computer is housed inside the operating panel, directly behind the TNC keyboard and the large, clear, 15-inch TFT flat-panel display with XGA resolution (1024 x 768 pixels). Complicated wiring is now a thing of the past.

The CC controller unit is still in the electrical cabinet, and is connected with the power stages via the well-proven PWM interface.

HSCI – the new modular hardware concept

The new hardware concept of the TNC 620 also ensures that connecting the individual components of the control will be child's play in the future.

The main computer, controller unit, and other components of the HEIDENHAIN control system feature a new, powerful interface: **HSCI.** The outstanding properties of the uniformly digital design of the TNC 620 entire system guarantee not just very high accuracy and surface quality, but also rapid traverse speeds as well as high availability of the entire system *(see page 14 for more information).*

And what's inside? Plenty of processing power and memory space!

The TNC 620 is equipped with a powerful Intel processor with a clock frequency of 400 MHz. In addition, the 512 MB of RAM ensure that complex graphic simulations are processed quickly.

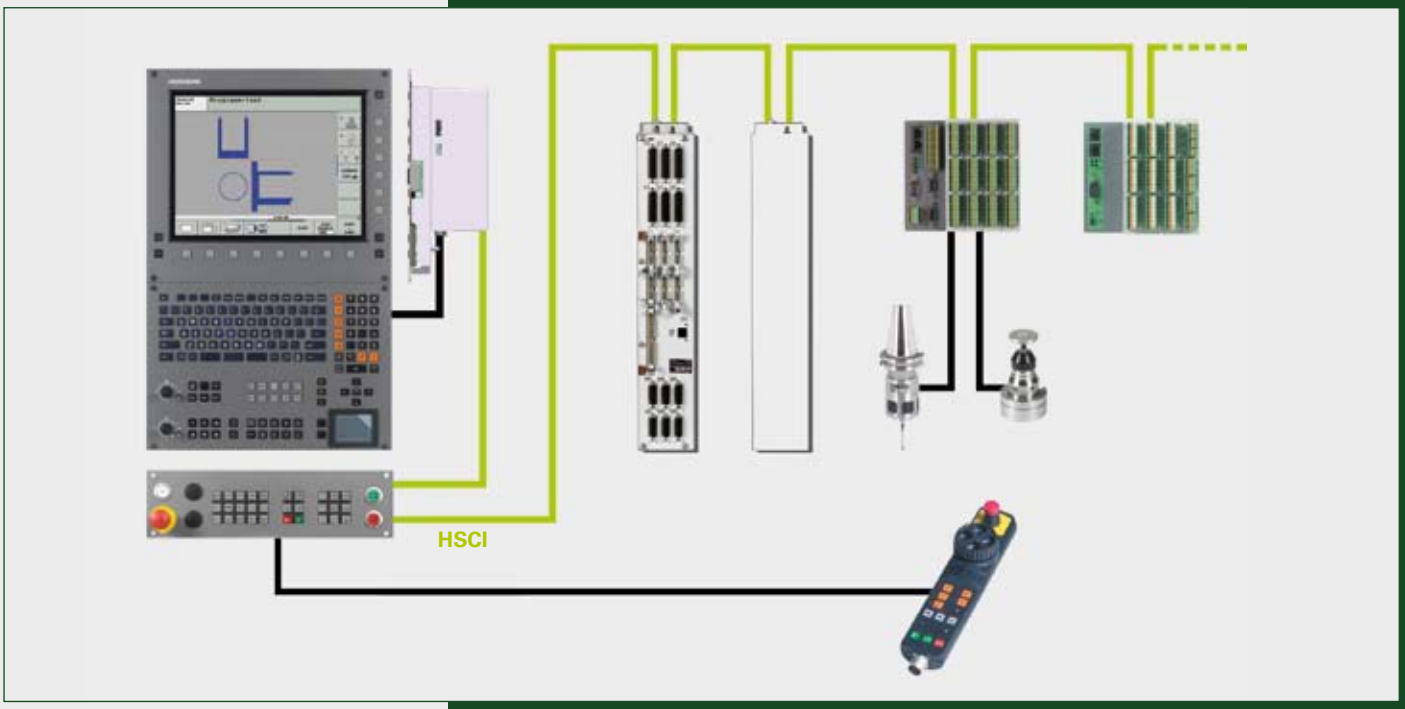
A CompactFlash memory card is used for both NC and PLC programs. The memory card is immune to mechanical shock, thereby offering optimum safety for your data.

The TNC 620 is a reliable partner in the workshop when it comes to data transmission. This is taken care of by the Fast Ethernet interface, integrated as a standard feature, which makes it possible to connect the TNC to your company network with very little effort.

Input and pointing devices, as well as external media (such as USB sticks or external hard drives), are connected to the TNC 620 via the USB port.

The TNC 620 is offered with three controlled axes and a controlled spindle, and as an option, two more controlled axes can be added. (Software options can be used to adapt the scope of function of the NC software to the respective needs and applications.)

Uniformly Digital – The New Hardware Design for Controls from HEIDENHAIN



The hardware design of the future: Uniformly digital connection of the various control components

HSCI: *HEIDENHAIN Serial Controller Interface*

Uniformly digital is more than just a buzzword: all components are connected to each other via purely digital interfaces. The control components are connected via HSCI (HEIDENHAIN Serial Controller Interface), the new real-time protocol from HEIDENHAIN for Fast Ethernet, and the encoders are connected via EnDat 2.2, the bidirectional interface from HEIDENHAIN.

Both the machine manufacturer and the end user profit from the advantages: the entire system becomes less susceptible to noise, is thoroughly diagnosable, and so ensures a high degree of availability.

The previous, proven hardware design

The MC main computer and CC controller unit are both contained in the electrical cabinet. The operating panel only includes the screen and keyboard. The components of the operating panel are connected to the MC main computer via several cables.

The new hardware design

The MC and CC are connected via a real-time Ethernet cable, specifically a 100BaseT Ethernet Physical Layer device. The protocol was developed by HEIDENHAIN, and carries the designation HSCI. Together with the purely digital EnDat 2.2 encoder interface, there is a uniformly digital design from the main computer to the encoder.

The main advantages of this new design:

- Simpler wiring
- Simpler commissioning
- Extensive possibilities for diagnostics
- Improved noise immunity

This new technology ensures highest accuracy and surface quality at high traverse speeds.

Safety-Related Control Technology for Machine Tools

Safety is becoming increasingly important in machine and plant construction. These measures mainly serve to protect human beings, but material assets and the environment are also receiving more consideration.

The goal of functional safety is to minimize or at least reduce the risks that can occur during normal or impaired operation of machines or facilities. The first step of this is achieved with redundant systems. For example, axes that are moved in safety-oriented applications require redundant position information in order to perform the corresponding safety functions.

Basic principle

The controls and position encoders from HEIDENHAIN with functional safety meet safety integrity level 2 (SIL 2) as per the IEC 61 508 standard, as well as the performance level "d" as per ISO 13 849-1 (which replaced ISO 954-1). These standards describe the assessment of safety-oriented systems, for example based on the failure probabilities of integrated components and subsystems. This modular approach helps manufacturers of safety-oriented systems to implement their systems, because they can begin with prequalified subsystems. Position encoders with functional safety and the iTNC 530 with HSCI control accommodate this concept.

Functional safety on machine tools

HEIDENHAIN is planning on offering HSCI controls with functional safety starting in the middle of 2008. Two redundant safety channels that work independently of each other are the foundation for controls with functional safety. All safety-relevant signals are captured, processed and output via two channels. Errors are detected by mutual comparison of the states and data in the two channels. This way, the occurrence of just one error in the control does not lead to the safety functions being incapacitated.

The goal is to make actions by the machine operator at machining centers possible during automated production runs, even when protective measures are not in effect (such as protective doors being open), without danger to the operator:

- Setup
- Manual intervention
- Process monitoring

Safety-related operating modes

HEIDENHAIN controls with functional safety offer four safety-related operating modes as per the EN 12 417 standard (Machine Tools–Safety–Machining Centers).

Operating mode 1
Automated or production mode

- Operation only with closed protective door
- No machine motion possible if protective door is open

Operating mode 2
Set-up mode

- Operation with open protective doors
- Axis motions of 2 m/min at most
- Spindle stop within 2 revolutions
- Only one axis can be moved at a time (no interpolating motions)
- Spindle rotation only possible with permissive button

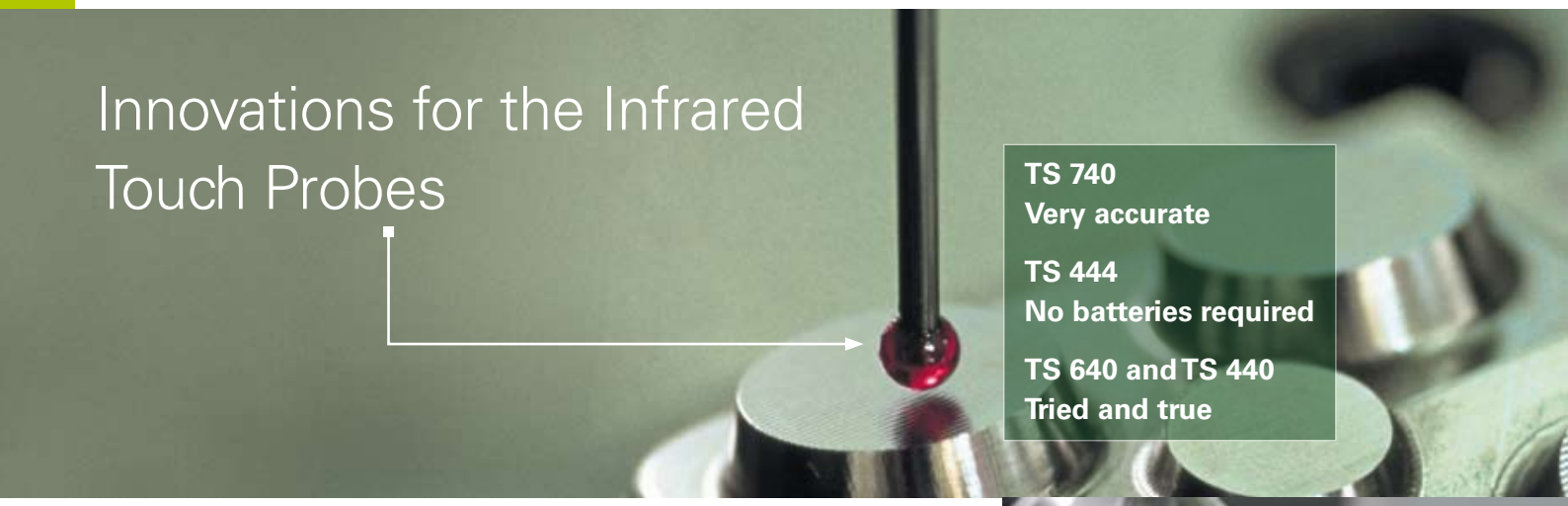
Operating mode 3
Manual intervention

- Operation with open protective doors
- Axis motions of 5 m/min at most
- Spindle stop within 5 revolutions
- More than one axis can be moved at a time (interpolating motions)
- Spindle rotation only possible with permissive button

Operating mode 4
Advanced manual intervention, process monitoring

- Operation with open protective doors
- Axis motions of 5 m/min at most
- Spindle stop within 5 revolutions
- More than one axis can be moved at a time (interpolating motions)
- Permissive button must only be pressed to start spindle rotation

Innovations for the Infrared Touch Probes



TS 740
Very accurate

TS 444
No batteries required

TS 640 and TS 440
Tried and true

The TS 740 and TS 444 workpiece touch probes are two newly developed products from HEIDENHAIN.

TS 740 –
The high-accuracy touch probe

The TS 740 is the right touch probe for measurement tasks with especially high demands regarding probing accuracy and repeatability. In spite of its very low probing forces, the TS 740 does not generate an uncontrolled trigger signal at high accelerations and rapid probing.

Probing process:
Contact with a workpiece deflects the stylus, applying a force to the pressure elements, which are the core of our newly developed sensor. The difference in forces is calculated by the electronics, and the trigger signal is generated.

Touch probe	Probe accuracy	Probe repeatability (repeated probing from one direction)
TS 440 / TS 640	≤ ± 5 µm (when using a standard stylus)	2 σ ≤ 1 µm at a probing velocity of 3 m/min
TS 740	≤ ± 1 µm	2 σ ≤ 0.25 µm at a probing velocity of 0.25 m/min

HEIDENHAIN TS touch probes

A frequent requirement is the reduction of setup times. Use our workpiece touch probes to perform setup, measuring and inspection functions directly on the machine tool.

With touch probes from HEIDENHAIN you can:

- measure workpieces,
- align workpieces,
- set datums,
- and digitize 3-D forms.

HEIDENHAIN offers touch probes that transmit the trigger signal either by cable or by infrared transmission.

TS 444 – The touch probe without batteries

The TS 444 is an innovative and smart alternative to touch probes with batteries. Handling, storage and disposal of batteries is completely done away with. The only requirement is the supply of compressed air through the spindle.

The action of cleaning the workpiece position to be measured simultaneously charges the touch probe.

The principle behind the energy supply:
Compressed air is pushed through the taper shank into the touch probe in order to clean the measurement point before it is probed. A turbine wheel is driven there. Changes in the magnetic field generate electrical energy, which is stored in high-power capacitors. The exit air is used for cleaning the probing point. The compressed air does not need to be specially cleaned.

The charging time varies depending on the pressure: The higher the pressure, the shorter is the charging time. In order to ensure that charging takes place within a reasonable time, the supply pressure should be at least 5 bars.

For example, at 5.5 bars it takes around 3 seconds to completely charge a touch probe. This suffices for a two-minute measuring cycle.

TS 640 and TS 440 – The tried and true touch probes

Our well-proven TS 640 and TS 440 touch probes also have new features.

New:
Longer operating time

We were able to more than double the operating time for each set of batteries. By revising the electronics, we were able to increase the operating times to about 800 hours for the TS 640, and 200 hours for the TS 440.
Example:
If a touch probe is active 5% of the operating time, the batteries must be exchanged after three years on the TS 640, and after nine months on the TS 440 (3-shift operation, 220 workdays/year, lithium batteries).

New:
Different possible types of batteries

Alkaline batteries or rechargeable batteries can now be used as well. Please keep in mind that the lifetime of high-quality lithium batteries is somewhat longer.

New:
Optical indicators

Knowledge at a glance: the operator can now see if the touch probe is switched on or off. The new indicators also signalize a deflection of the stylus.

MANUALplus 620, the Contouring Control for CNC and Cycle Lathes

The screen: clearly structured and user-friendly

The operating panel: few keys, identifiable functions

For years now the MANUALplus 4110 has proven itself on action-oriented lathes. HEIDENHAIN now introduces a much improved version of the MANUALplus. The MANUALplus 620 features improved cycle programming, and also the brand new smartTurn programming mode of operation. This is HEIDENHAIN's newest control: conceived both for CNC and cycle lathes.

MANUALplus 620, the control for CNC and cycle lathes

The MANUALplus 620 is designed for lathes with spindle, one slide (X and Z axis), C axis or positionable spindle and driven tool. It is suited for horizontal and vertical lathes with simple tool holders or with tool turrets. Cycle lathes are usually used for smaller and mid-size production batches. The operator of a MANUALplus 620 profits from the quickly learned cycle programming, with which workpieces can be machined quickly and efficiently. And when requirements increase and you machine complex tasks with your lathe, you then create your NC programs with the new smartTurn programming mode. The smartTurn programming mode of operation is the basis for NC programming on CNC lathes.

This new type of NC programming is also mastered quickly, since the machinist does not have to deal with G or M functions, or with structuring a machining block. smartTurn uses the easy-to-learn form entry method of programming.

Cycle machining

The writing and testing of a "real" NC program, with G and M functions, takes too much time for small and mid-size production batches. The cycle programming of the MANUALplus is the ideal solution here, since a cycle is a pre-programmed machining step, and therefore only requires few entries.

The machinist can concentrate on the machining of his workpiece. He determines the tool for the machining step, selects the cycle, defines the required parameters, monitors machining with the graphic simulation function, and performs the cycle. This way, in an action-oriented manner, the first workpiece and the cycle program are created at the same time. This cycle program is then saved. The machinist can now let the program run automatically, and each additional part machined saves time and cost.

You define simple lathe and milling contours directly in the cycle on the MANUALplus. And what happens if the contours become more complex? No problem! Even complex workpieces can be described quickly and without additional calculations when using ICP contour programming.

smartTurn - the new programming mode of operation

Has the safety clearance been correctly entered, is the speed limit taken into account, how are oversizes defined? All this needs not only be considered by the beginner, but also by the experienced NC programmer when creating conventional DIN/ISO programs. smartTurn makes many things much easier: **the working block known as a unit** plays the central role in smartTurn programs. A unit describes a machining step completely and unambiguously. *The unit includes the tool call, the technology data, the cycle call, the approach and departure strategies, as well as global data, such as safety clearance, etc.* All these parameters are summarized in one form—simply and clearly.

For simple operations, you need only enter a few parameters. You use smartTurn to define such a machining step quickly in a single overview window. If required you can define additional machining options. These options are available in subforms in which, with a few additional keystrokes, you can enter the data for machining options.

The principle of smartTurn gives you the reassurance that the working block is defined correctly and completely. In the NC program, smartTurn lists the DIN PLUS commands of the unit. This not only gives you an overview of all working-block details, but you also have a clearly legible and well-structured NC program.

Describing contours with ICP

You describe workpieces or contours with the interactive graphics of the ICP contour editor. You create the contour by entering the elements step-by-step. When selecting the contour elements, you already specify the direction of the line or the direction of rotation of the circular arc. This way the MANUALplus needs very little information about the contour element. The MANUALplus calculates missing coordinates, intersections, circle center points, etc. In most cases you can describe the workpiece with the dimensions given in the production drawing. If there are multiple solutions, ICP displays the mathematically possible variants from which you can select the correct solution.

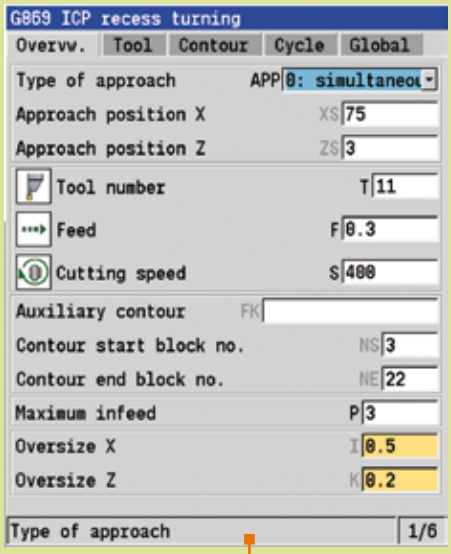
Importing DXF:
It's even easier when the workpiece drawing is already in DXF format, since with ICP you can import contours available in DXF format. Not only does this save time otherwise spent on testing, but you can also be sure that the finished contour is exactly according to the designer's specifications.

Contour follow-up:
Another highlight of the MANUALplus 620 is the contour follow-up feature. If you define the workpiece blank at the beginning of your smartTurn program, the control then computes the new blank for each following cycle. The machining cycles are adapted automatically to the current workpiece blank. They are so intelligent as to avoid air cuts and to optimize approach paths, even if the workpiece material has been previously removed.

You can use the smartTurn units to realize the entire lathe machining operation, as well as drilling and milling operations with the C axis. However, if you want to control special machine components or use variable programming, then DIN PLUS programming is the answer. You use DIN PLUS to realize functions that are not provided by smartTurn. A significant advantage of the MANUALplus 620 is that it enables you to switch between the smartTurn and DIN PLUS programming modes within an NC program.

Unit selection:
smartTurn also provides special machining units to go along with those for turning, drilling and milling machining. You define global program parameters, such as oversizes, safety clearances, coolants, etc., in the start unit. smartTurn then transfers these parameters to the other units.

Unit as form:



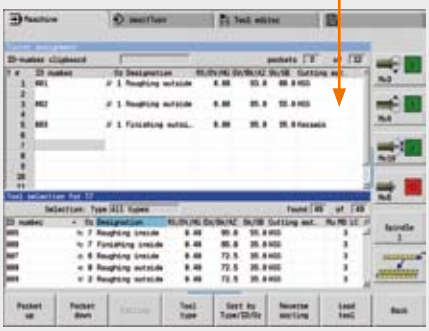
Unit as NC program:

```
N 400 UNIT ID"G869_ICP"[G869 ICP recess turning]
N 402   T11
N 403   G96 S400 G95 F0.3 M3
N 404   M8
N 405   G0 X75 Z3
N 406   G47 P3
N 407   G869 NS3 NE22 P3 I0.8 K1 Q0 U0 H0 V0
N 408   G14 Q0
N 409   G47 M9
N 410 END_OF_UNIT
```

The tool and technology database

Tried-and-true features of the MANUALplus include the saving of tool and cutting data, as well as simple determination of setting dimensions.

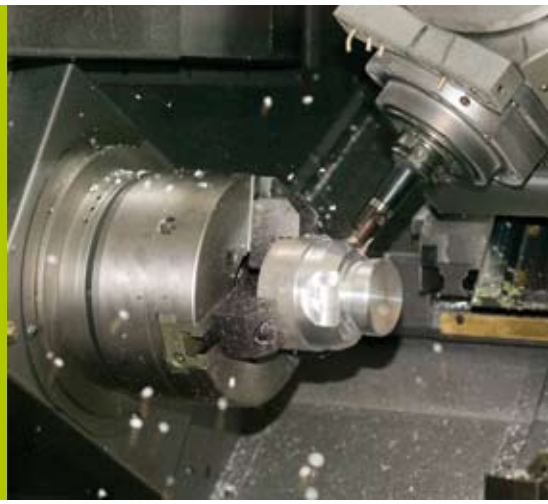
The tool database of the MANUALplus 620 not only provides greater capacities and easily understood dialogs for data entry, but the MANUALplus 620 also supports placement of tools in the turret.



If you want to change the tool assignment or the tools in the turret, you can additionally display the tools currently in the turret in the upper window, and the entries of the tool database in the lower window. Now you need only select the turret pocket and choose the correct tool from the database. You can transfer the tool data to the turret assignment entry with a simple keystroke.

With the MANUALplus 620, you also need enter the cutting data only once. The technology database saves the cutting data according to the criteria of workpiece material, cutting material and machining mode. Thanks to this three-dimensional table, the control always knows the appropriate feed rate and the correct cutting speed. The MANUALplus 620 determines the machining mode from the cycle or with smartTurn from the unit. The cutting material is defined in the tool description. You need only define the workpiece material at the beginning of the cycle program or the smartTurn program, and the technology database will propose the correct values for your machining operation. You can use the suggested cutting parameters or adjust them if required.

CNC PILOT 4290 with B Axis



At the EMO 2007 HEIDENHAIN will present the CNC PILOT 4290 with B axis for the first time. The B axis makes it possible to drill, bore and mill in oblique planes. But lathe machining also profits significantly from the B axis: by tilting the axis and rotating the tool you can bring it into positions that enable you to use a single tool to machine in the longitudinal and transverse directions on the main and opposing spindles.

The CNC PILOT 4290

The CNC PILOT 4290 lathe control was conceived both for compact and complex CNC lathes. The control can be used for machines with up to six slides, four spindles, and two C axes (up to a total of 12 control loops). You do not just profit from the flexibility of the control, but the **programming itself is incredibly easy, even for complex machines with multiple slides.** When creating the program with TURN PLUS, you describe the workpiece with interactive graphics, and then generate the NC program automatically with a keystroke. The automatic working plan generation feature of TURN PLUS creates the working plan, selects the working strategy, determines the tools and cutting data, and generates the NC blocks, even for NC programs for machining with opposing spindles.

The CNC PILOT assists you in the creation of clearly structured and legible NC programs using DIN PLUS. In DIN PLUS you first describe the contour of the workpiece, and then program the machining steps. Workpiece machining on machines with multiple slides is already programmed into the CNC PILOT 4290's command set. Specialized four-axis cycles, synchronization commands, etc. facilitate creation of programs for machines with multiple slides.

The B axis

At first glance the programming of drilling, boring and milling operations in oblique planes appears very complicated and time consuming. On the CNC PILOT, however, the programming is preceded by a coordinate transformation that makes it as easy as working in a main plane.

The usual separation of contour description and machining on the CNC PILOT also applies to milling, drilling and boring operations on a tilted plane. First you rotate and shift the coordinate system so that it lies in the tilted plane. Then you describe the hole pattern or the milling contour as you would in the Y/Z plane. Here you can use the hole pattern and figure definitions of the CNC PILOT. This means that, for linear or circular patterns and simple figures (circles, rectangles, regular polygons, etc.), you only need a few more entries to describe the pattern or figure on the tilted plane.

For boring and milling you move the tool to a position perpendicular to the tilted plane. Then you start cutting with the cycles, using the same drilling, boring and milling cycles as for the Y/Z plane. The CNC PILOT already has the required parameters of the tilted plane from the contour description.

The simulation feature of the CNC PILOT 4290 shows the hole pattern and milling contour for the B axis perpendicular to the tilted plane—without distortion. This ensures simple verification of programmed patterns and contours. In the position display of the simulation feature the CNC PILOT displays the angle of the tilted plane and the tilt angle in the B axis.

Flexible use of tools with the B axis

If your machine is equipped with a B axis, you can use your tools much more efficiently than before. On conventional lathes you need four different tools for longitudinal and transverse turning on opposing spindles. With a B axis, you can do it with a single tool.

You simply tilt the B axis and rotate the tool to the normal position or for machining from behind the workpiece—whichever is required for longitudinal or transverse turning on the main or opposing spindle. All you need is a single call. The CNC PILOT calculates the tool lengths, the tool angle and the other tool data for you.

Tool-use flexibility is increased significantly when several tools are mounted in one holder. For example, with a roughing, finishing and recessing tool you can perform considerable parts of turning and recessing operations on a main and opposing spindle—without changing the tool. And programming is very easy. You simply indicate which tooth of the tool to use and then define the tilting angle and the tool position. And that's it, because the CNC PILOT already has the rest position and the data of each tool tooth in its database.

This type of flexibility lowers the number of tools, and you save valuable machining time by reducing the number of tool changes.

Lathes with multiples slides and B axis

The B axis not only significantly increases the range of parts that can be produced on a lathe, but the machine productivity is also improved due to the flexible use of tools. Since a B axis is usually used on lathes with multiple slides and spindles, the NC programmer is faced with the challenge of optimally distributing the machining tasks over the various available slides and spindles.

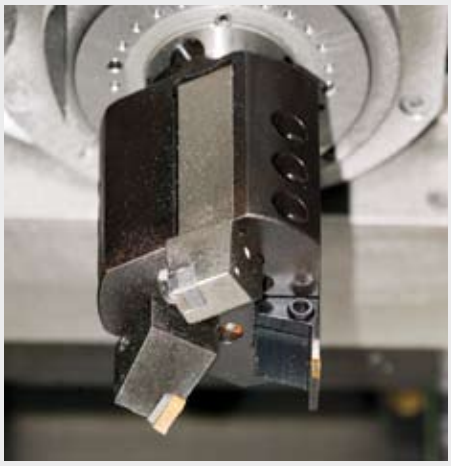
The CNC PILOT offers support with various programming and monitoring functions. For example, creating a program for machining the rear side is made easy by mirroring and/or shifting the contour, as well as by converting specific NC commands for machining with the opposing spindle. In addition, the assignment of program sections to slides or spindles also helps to give the part program a clear structure.

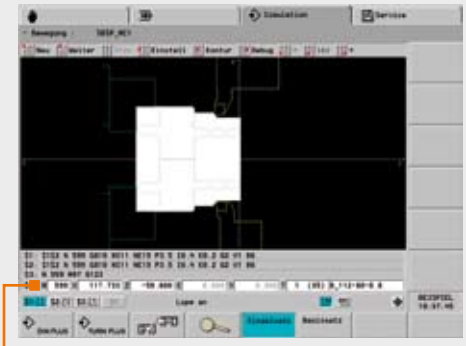
Simulation

The graphic simulation feature of the CNC PILOT 4290 supports the monitoring of complex machines and facilities. The control shows all workpieces and the tool movements of all slides in the simulation window. The CNC PILOT 4290 takes the entire machining zone into consideration. Tools and chucking equipment are shown to scale.

The graphic simulation is characterized by a high degree of flexibility. You can define what is shown in the simulation window. You can specify whether the window for lathe machining is shown, for the end face or the lateral surface, the side view with B-axis machining, or a combination of these windows.

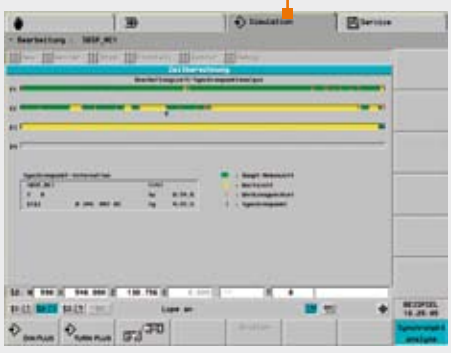
With the help of this support, you can effectively and comprehensively program and check complex, multi-slide programs—even before making the first cut.





Synchronous point analysis

During the simulation, the CNC PILOT 4290 saves the productive, non-productive and idle times, as well as all tool changes and synchronization points. Based on this information the synchronous point analysis shows the chronological sequence of the machining, and the interdependency of the slides. This serves to make the sequence of the workpiece machining steps more transparent, which is a good basis for the NC programmer to analyze and optimize the machining of the workpiece.



E-Learning for CNC Specialists and for Vocational Training

Fundamentals of CNC programming
Fundamentals of machining in tilted working planes
New: fundamentals of touch-probe applications

MITS project

As part of the **"Leonardo da Vinci"*** program, and in cooperation with partners in Belgium, Luxembourg, Hungary and Spain, members of our Technical Training department have been developing a concept since 2004 for a **M**odular **I**nteractive **T**raining **S**ystem **(MITS)** for mechatronics engineers, and have implemented a CNC fundamentals training course for practical use.

Modular

The e-learning system consists of individual units, each with its own goal.

The modular structure makes it possible to design courses adapted to the specific needs of the students.

Interactive

Interactive Flash simulations, which permit the students to control the animations, are used for sequences and animated graphics.

Practice-oriented e-learning content

Real-world situations are described with interactive scenarios (graphics, videos, animations). They include "hands on" learning experiences, since they also permit the students to make mistakes, and to learn from them.

*"Leonardo da Vinci" is a program of the European Union to promote vocational training.

From e-learning units to an e-learning course

A course is assembled from an e-library or repository that contains all e-learning units. The resulting course can then be used in any learning environment.

TNC Training, Version 3

The new, third edition of the HEIDENHAIN e-learning TNC Training program now includes the fundamentals of touch-probe applications in the world of CNC.

The complete program now contains the following units:

Fundamentals of NC programming:

- Coordinate systems
- NC axes
- Tools
- The TNC
- Programming basics
- Frequently-used functions

Fundamentals of machining in tilted working planes:

- Programming basics
- Tool compensation
- Use in tool and mold making

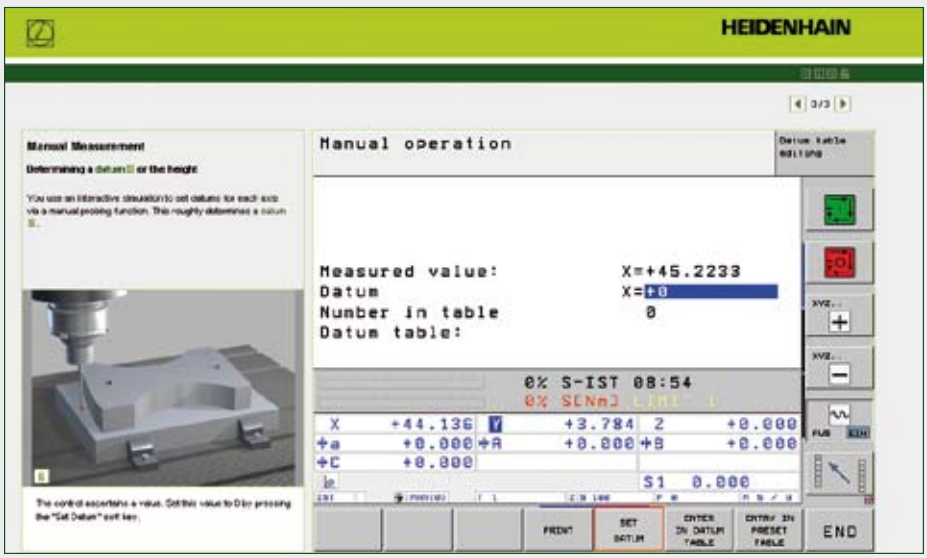
New: Touch-probe applications

- Measuring the workpiece
- Tool measurement

The user can choose between the following languages: German, English, French, Italian, Spanish, Dutch, Czech, Hungarian, and Chinese.

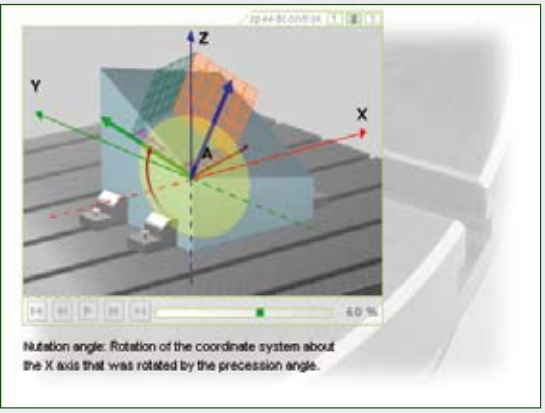
The e-learning program is also available over the Internet in the Services -> Training area.

Of course you can also request a free DVD from HEIDENHAIN.



The TTC Varelerhafen

The Technology Transfer Center Varelerhafen has been the northernmost authorized training partner of HEIDENHAIN in Germany since 2004. Today participants come from over 500 km away to take part in the TNC programming courses at the TTC, mainly because of the practice-oriented qualification measures offered.



The DVD is automatically sent to participants in the iTNC 530 Basics, Tilted Machining and Touch-Probe Cycle courses when their registration is confirmed, so that each participant can prepare himself individually for the respective course.

The success of the TTC is based on its training concept

The TTC was born from the idea of the Deharde Maschinenbau company to efficiently provide basic and advanced training for both apprentices and experts, in practice and in theory, that would suit their needs. Since its founding in December 2003, the Technology Transfer Center Varelerhafen, a privately-owned training center attended by many companies, has become a mainstay for practice-oriented courses in Germany. The TTC Varelerhafen is one of the most active of HEIDENHAIN's authorized training partners.

Experts increase their know-how

"We know that motivated and competent employees are the most important resource, especially in production and processing trades," states Holger Hoffmann, managing director of the TTC.

The TTC offers a wide range of courses, from one-day seminars to week-long courses, either in Varelerhafen or on-site at the customer. All important areas, from CNC milling to CAD/CAM to control technology and pneumatics are covered. The relationship to the real world of the shopfloor is of utmost importance at the TTC. The trainers have many years of practical experience. State-of-the-art machines and equipment from renowned manufacturers are used for training. The comprehensive assortment of machines owned by the Deharde Maschinenbau company is available for this.

In summary, the training concept of the TTC is as follows:

- Flexible course times – individual dates can be agreed to on short notice
- Quality instead of quantity – Small groups with up to only 4 participants
- The training site is completely up to you: "Either you come to us, or we come to you"
- Experts teach experts
- The professional trainers have been master craftsmen and engineers with many years of experience
- Many machines on site, e.g. three 5-axis CNC centers from DMG
- Consolidated knowledge – participants receive easy-to-understand documents for each course

Further information and an overview of the courses can be found on the Internet at www.tectransfer.de. Of course the team at the TTC will gladly answer your questions at +49/4451/9133550.

Warning! Machine tools without linear encoders may be inaccurate.

HEIDENHAIN shows the way to precision.

Machine tools without linear encoders use the pitch of the ball screw as the measuring standard. But at the same time, the ball screw transfers enormous forces at high traverse speeds and deforms due to thermal changes. Result: the position values become inaccurate. Machine tools with linear encoders are statically, dynamically and thermally more precise—advantages that we symbolize with a sign. Most linear encoders installed on machine tools have it: our sign of precision. For more information, visit: www.heidenhain-shows-the-way.eu

angle encoders + **linear encoders** + contouring controls + position displays + length gauges + rotary encoders

Visit us at the EMO in Hannover from Sep. 17 to 22, 2007, Hall 25, Booth E18.

This invitation is our gift to you. Simply cut out the mark of precision and trade it in at the Laatzen train station for an EMO admission ticket.