HEIDENHAIN

54 + 9/2011

Klartext

News from the World of HEIDENHAIN Controls

Symbiosis of Efficiency and Convenience

New Combination: The TNC 640 for Turning and Milling

The Best of Two Worlds!

The New Generation of HEIDENHAIN Touch Probes ➔ **Page 8**

New Design Line of HEIDENHAIN Controls

A Comparison of the Controls ➔ **Page 10**

Editorial

Dear Klartext Reader,

Multi-operation machining is the hot item right now, and for good reason: if a workpiece can be both turned and milled in just one setup, then machining times and door-to-door times can be reduced significantly. The brand new TNC 640 milling/turning control is just the right thing for the machine operator to remain master of this combination of complicated machining operations. The new control with its elegant design will be introduced at the EMO trade show in Hanover, Germany. Read about the product features on page 4.

New features of the controls are the central topic in this issue of KLARTEXT. The iTNC 530 high-end control, the TNC 620 contouring control, and the MANUALplus 620 and CNC PILOT 620 lathe controls all have new and practical functions that the machine operators can look forward to.

Another highlight at EMO: all controls will be presented in their new stainless steel design.

Young entrepreneurs from Tyrol are setting a much different type of design trend. Their unconventional approach to CNC machining excited the KLARTEXT editorial staff. Read more about this on page 20.

Read and enjoy, with best wishes from The Klartext staff!

HEIDENHAIN at the EMO in Hanover – Hall 25, Booth E33

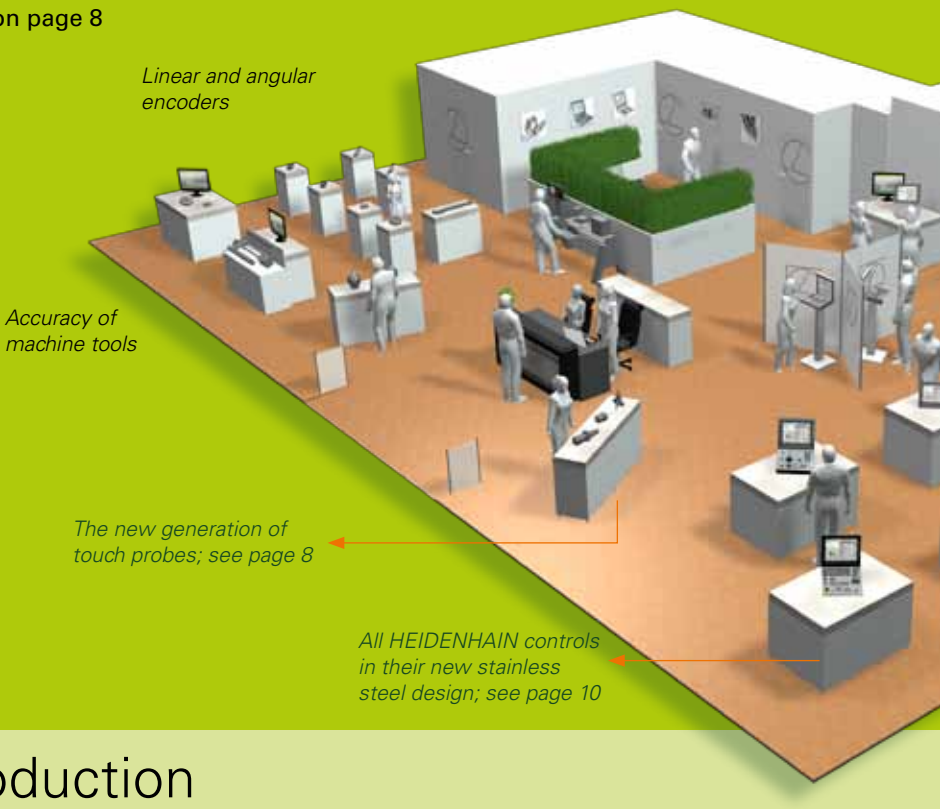
Witness the Debut of the New TNC 640 Control!

The time has come! HEIDENHAIN is presenting its brand new **TNC 640** control for combined milling and turning machining at its trade-show booth. Don't miss your opportunity to see the control's new design in its modern, stainless steel housing. **Read more on page 4**

There are also other new products: A completely new **generation of touch probes** with two methods of transmitting the probing signals will presented for the first time at the trade-show booth. **Read more on page 8**

The accuracy of machine tools is another emphasized topic. Using vivid examples, we will show how accuracy and productivity can be increased with **linear and angle encoders.**

We look forward to talking with you at our trade-show booth.



Production

Publisher
DR. JOHANNES HEIDENHAIN GmbH
Postfach 1260
83292 Traunreut, Germany
Tel: +49 (8669) 31-0
HEIDENHAIN website:
www.heidenhain.de

Responsible
Frank Muthmann
E-mail: info@heidenhain.de
Klartext on the Internet
www.heidenhain.de/klartext

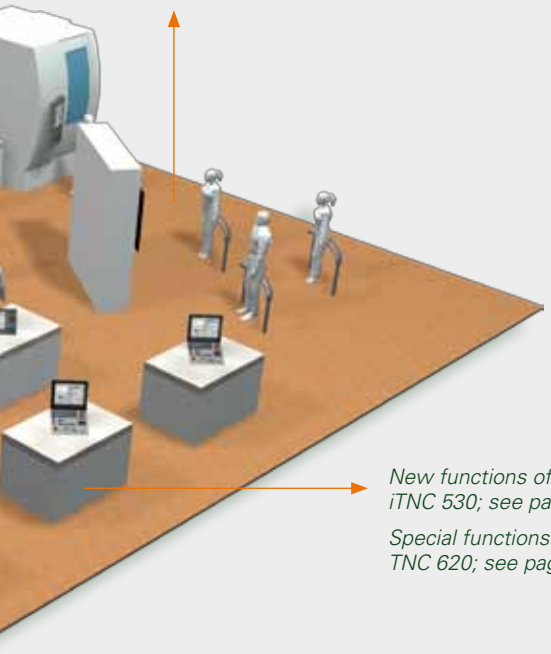
Also, check out our interactive KLARTEXT e-magazine, with even more background information, animations, and expert knowledge. Take a look at

www.heidenhain.de/klartext

Contents

EMO Hannover 19-24·9·2011

The new TNC 640 – the high-end milling/turning control from HEIDENHAIN on a machine tool; see page 4



New functions of the iTNC 530; see page 12

Special functions of the TNC 620; see page 16

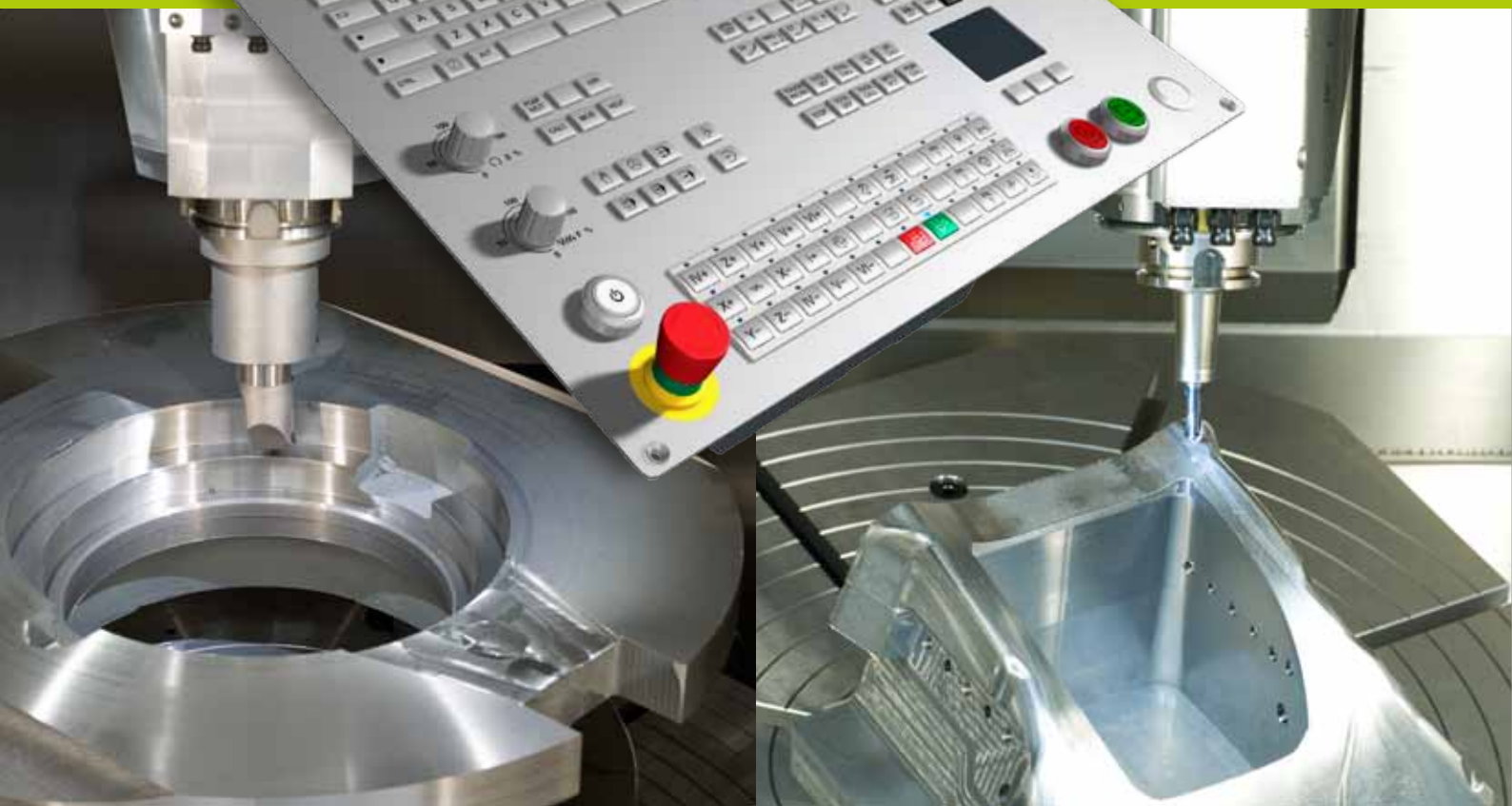
Editing and Layout
Expert Communication GmbH
Richard-Reitzner-Allee 1
85540 Haar, Germany
Tel: +49 (89) 666375-0
E-mail: info@expert-communication.de
www.expert-communication.de

Symbiosis of Efficiency and Convenience

New Combination: The TNC 640 for Milling and Turning

HEIDENHAIN is presenting the new TNC 640 at EMO 2011. For the first time, milling and turning are combined in one TNC. The new TNC 640 is designed for machine tools capable of multi-operation machining. Its exterior already captivates you with its classy design: the restyled keyboard, elegant stainless steel housing, and large 19" monitor. But as you are accustomed to from HEIDENHAIN, it is the intrinsic values that are supposed to convince you: the ones that make switching between milling and turning modes easy and practical for daily use.

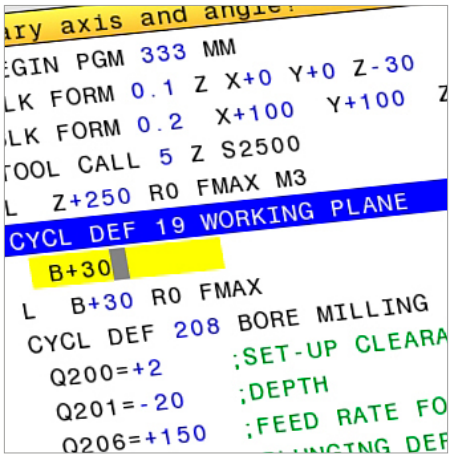
Machine tools for multi-operation machining are the hot item right now. However, the combination of milling and turning not only saves time and money, but also makes both the machine and the control more complex. In order for the machine operator to keep cool during this combination of operations, HEIDENHAIN has pulled out all the stops to ensure that operation is as easy and straightforward as possible: this starts from the new design, and continues to the functions that make switching between milling and turning especially easy, preventing errors from occurring.



The TNC 640 – the new, high-end control from HEIDENHAIN for milling and turning

Fresh new appearance

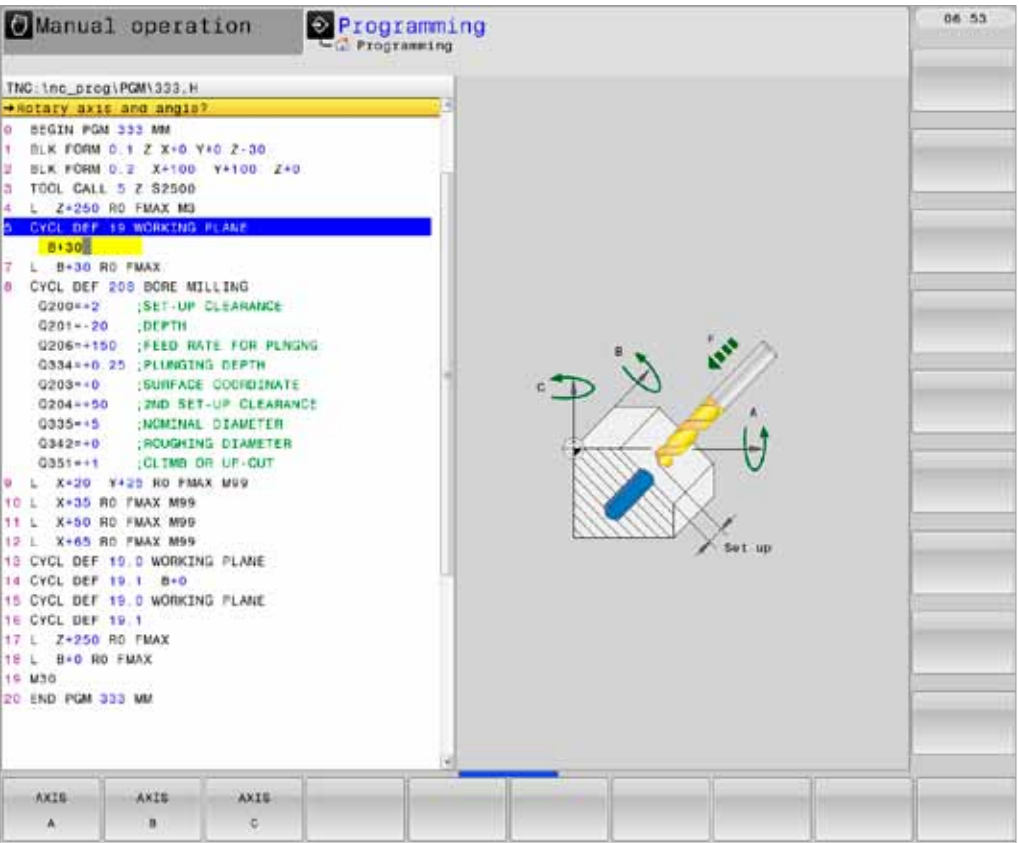
If you take a look at the new TNC 640 at EMO, you can be impressed by the modern design of the stainless steel housing around the 19" screen as well as by the new keyboard with its slightly convex keys. They are pleasant to the touch and reliable in operation. Their inscriptions do not wear off, even under extreme workshop conditions. The integrated machine operating panel now has LEDs that indicate the machine's current operating statuses.



Improved overview of NC programs through identification via various colors.

The rectangular, slightly rounded keys are pleasant to the touch and reliable in operation.

With smartSelect you can select functions quickly and easily under dialog guidance.



In the NC program, you can use very easy plain-language commands to change as desired between turning and milling.

Optimized editor for clear and simple operation

Operators of HEIDENHAIN controls just need one quick look at the editor to see the next development: tabs on the user interface indicate the various groups of operating modes. The programming dialog is now more like the window with the machining program, and various colors are used by syntax highlighting to differentiate between commands, values and comments. All this lets the machine operator see more easily and reliably what is important at the moment.

Quick access to soft-key functions with smartSelect

The growth in functions has naturally led to many more soft keys. The new smartSelect feature helps to "disentangle" the lower-level soft keys, and presents them in a tree view. Of course only those functions that can be defined in the current operating state are shown. Clicking the desired functions reveals detailed information in the right half of the window.

A quick change ...

Machining programs for multi-operation machining cover milling, drilling, boring and turning procedures. Since that would appear to make program creation quite difficult, the software developers at HEIDENHAIN made sure that switching between the operating modes would be very simple: standardized plain-language commands are used in the NC program to switch between milling and turning modes at will. There are almost no restrictions, since the switchover is completely independent of the momentary axis configuration.

During milling the TNC plays out its familiar strengths, but for turning it has learned a lot from the HEIDENHAIN lathe controls,

including the tried-and-true turning cycles. The turning contours are naturally programmed in HEIDENHAIN conversational language, and can be saved in contour subroutines, just as for milling. The familiar FK free contour programming feature can also be used for turning contours to complete drawing data not dimensioned for NC. The range of functions of course also includes the recess and undercut contour elements, with their appropriate support graphics.

The qualities of the new TNC 640 are intended to make it very easy for operators with milling experience on TNCs to quickly and easily machine turned contours.

... with great ease

The operator is not supposed to be encumbered with manual modifications and conversions when switching between the machining modes. That is why the TNC 640 carries out many of these changes itself: the control adapts all displays to the respective mode. Along with the programming graphics this also includes toggling between position and diameter display. The datum can automatically be set to the middle of the rotary table, and machine-dependent functions can be activated, such as clamping of the tool spindle.

Functions that are the same for milling and turning keep their TNC designation, even if the control must act differently.

Two examples:

- In both modes M144 stands for the necessary tilting of the axes with inclined tool. When turning with an inclined tool, the TNC 640 considers the angle of inclination in order to avoid damage to the contour.
- If the same tool is to be used for internal and external turning, it can be reversed with Cycle 800 without having to change the tool orientation in the tool table. The TNC 640 calculates the inclination of the tilting axes itself, and positions the milling spindle with the turning tool to the correct position.

New Turning Cycles

HEIDENHAIN controls have always been known for their comprehensive and technologically sophisticated package of cycles for frequently recurring operations. Naturally this also applies to the TNC 640. The package of cycles contains, along with the familiar TNC milling, drilling and boring cycles, many intelligent turning cycles, with which even complex turning operations can be programmed very easily at the machine. The existing know-how of the HEIDENHAIN lathe controls flowed into this package. The tool geometries defined in the turning tool table are considered by the turning cycles into order to avoid contour damage.

Along with the milling cycles the TNC 640 offers many turning cycles for

- Longitudinal turning
- Transverse turning
- Recessing
- Contour recessing
- Thread cutting

Summary: Solving complicated tasks as easily as possible

With the simple switch between machining modes, HEIDENHAIN has remained true to its consistent strategy regarding its users: the functions of a control must prove their worthiness in practice. The more complex the tasks are, the more important a clear interface and simple operation become.

After all, not only the multi-operation machining on powerful machine tools should be particularly efficient, but also the program creation and setup. That is why the new, high-end TNC 640 control innovatively uses its proven virtues to make it very easy for the machine operator to perform this combination of milling and turning.

Forecast

The TNC 640 has its debut at EMO 2011 in Hanover. The first TNC 640 controls will be shipped to machine tool builders at the beginning of 2012. The KLARTEXT editorial staff is already excited about the first reports from the field, and will of course keep its readers up to date.

Profile of TNC 640

The new and high-performance control is suited for HSC and 5-axis machining on machines with up to 18 axes. Equipped with the familiar TNC milling functions and the turning functions, the new TNC 640 is a powerful control for multi-operation machining.

The hardware of the new TNC 640 is based on a new and completely digital hardware platform. All components are connected to each other through digital interfaces. The uniformly digital design is the optimum precondition for combining highest accuracy and surface definition with high traversing speeds.

The new TNC 640 has powerful processor hardware. The block processing time is just 0.5 ms, and a hard disk with up to 138 GB for use by the operator is available for NC programs.

The New Generation of HEIDENHAIN Touch Probes

The Best of Two Worlds

There are always two possibilities, and most of the time you have to choose one or the other. Or can you have both? HEIDENHAIN has chosen the two best solutions: the new generation of touch probes features a brand new combination for signal transmission. The best method is chosen, depending on how far the signals must be transmitted or how exact they must be. This way the machine operator can be even more sure of receiving exact position results.

Touch probes have become indispensable in the precise machining of workpieces. Together with the control they exactly capture the position and orientation of workpieces. This information makes it easy to align the workpiece to be machined, and to set datums on it. The probing signals must be transmitted to the control quickly and reliably. But that doesn't always happen. If the signal must cover a long distance, such as if the touch probe plunges deep into a large workpiece, transmission can be interrupted, and no information is received.

The new generation of touch probes, in this case the **TS 460** (workpiece touch probe) and **TT 460** (tool touch probe) feature two transmission methods: infrared and radio waves. This doesn't sound spectacular, but advantages of combining the two in one probe are obvious. Because depending on the signal, the appropriate method can be used, with convincing results:

Energy saving

The amount of energy consumed sinks dramatically. Depending on the surrounding radio traffic, the operating time can be nearly doubled compared to the **TS 440.** The incremental standby mode also saves energy; the lowest energy mode is already reached after just half an hour. And should the battery ever die, then you simply exchange it without any tools: all you need is a coin.

Versatile

There are no limits to where the touch probes can be used: they are happy on milling, turning and grinding machines, in enclosed machining centers and on large open machines. New applications are also possible. For example, the new touch probes can plunge completely into a cylinder without the signal disappearing.

Innovative

The respective advantages of the two methods for signal transmission come to play depending on the signal. Signals where time is not of the essence, such as switch on and switch off, are generally transmitted via radio waves. The trigger signal is quickly and reliably transmitted via radio waves (for long distances) or infrared.

In addition to the hybrid touch probes, HEIDENHAIN is also offering two new cable-connected probes: the **TS 260** workpiece touch probe—with axial or radial cable outlet—and the **TT 160** tool touch probe. These two "small" touch probes supply trigger signals that do not need an interface or amplifier.

The new generation also includes the new SE 660 transmitter/receiver unit. Depending on the application, the transceiver is mounted either within the machine's work envelope (if radio waves and infrared are used) or outside of the machine (if only radio waves are used). The SE 660 can easily be unclipped from its mount, in order to easily select a free frequency channel after the surrounding radio traffic has been scanned.

The new touch probes don't just feature the new combination of signal transmission, but have many new improvements: they are smaller and more compact, resulting in much more freedom for swiveling and tilting. The lens system of the proven and wear-free optical sensor was revised, and a preamplifier has been integrated. As with all workpiece touch probes, the new probes also have an integrated cleaning blower/flusher: cooling liquid or compressed air is used to clean the probing point of loose particles, greatly contributing to process reliability.

Device protection for high accuracy

In cluttered work envelopes the touch probes can easily collide with the workpiece or fixture. A newly developed mechanical adapter between the touch probe and shank permits small collisions and sends a stop command to the control, which then automatically begins a new calibration cycle.

Practical touch probes

Infrared and radio transmission—now available in one device. This new development offers the user many advantages. And there are even more benefits: the new generation of touch probes not only features high repeatability, but they are also practical in every conceivable manner. The wear-free optical sensor pays off when there are many probing cycles, as frequently occurs during grinding. Also, the machine operator gets results faster when using touch probes from HEIDENHAIN: for example, no adjustment is necessary with the TT 140/TT160. You can start immediately once they have been installed and calibrated.

Benefits of radio transmission
At a frequency of 2.4 GHz the large range for radio transmission is usually 15 m, but much larger ranges are possible in practice under ideal circumstances. The standard transmission protocol, such as as WLAN, does not require a license. Unique addressing of the signals makes it almost impossible to mistake them for one another. The transmitter does not have to be in the line of sight, making it possible to even use the touch probe inside of large workpieces.

Benefits of infrared transmission
The reliable infrared transmission supplies exact measured values, since the trigger signal is transmitted rapidly and consistently. The new infrared protocol is resistant to light interferences.

Overview of the New Touch Probes

Workpiece touch probes:
- TS 460 with infrared/radio wave transmission to the SE 660 transceiver unit for automatic tool change
- TS 260 with cable for manual tool change

Tool touch probes:
- TT 460 with infrared/radio wave transmission to the SE 660 transceiver unit
- TT 160 with cable

New Design Line of HEIDENHAIN Controls

Humans Are the Standard

HEIDENHAIN's goal of making everything as easy and practical as possible for the machine operator continues to be valid. One aspect is the design. It plays an important role in the operability of a control, since ergonomics and aesthetics must be in harmony as part of the man-machine interface. Good design enhances confidence during operation, and can lead to an increase in productivity. That is why HEIDENHAIN is presenting a completely new design line for all TNC controls at EMO 2011.

Classy design with restyled keyboard and elegant stainless steel housing

The current **iTNC 530** and **TNC 620** controls, the new **TNC 640** milling/turning control and the **CNC PILOT 620** lathe control will exhibit their new, shiny and high-quality stainless steel housing at EMO. The completely plane surface features a special protective coating, making it extremely insensitive to contamination in the harsh daily environment on the shop floor.

The keys have also been redesigned: rectangular, slightly rounded and slightly convex. They are pleasant to the touch, and reliable in operation. Furthermore, each key in the machine operating panel section has an LED indicating its status. So the machine operator can see at a glance which functions are currently active. With the handy control knobs, now slightly larger, you can individually adjust the feed-rate, rapid traverse and spindle speed. Also, there is a now a USB 2.0 port integrated in the keyboard unit, for direct and uncomplicated connection of storage media or pointing devices.

A Comparison of the TNC Controls

Production requirements can be quite different, as well as the demands of machine operators. HEIDENHAIN offers controls for every type of application, in order to satisfy each customer's demands regarding functions and quality. There's something for everyone, from compact versions to the premium version. What do all controls have in common? They increase efficiency and reduce costs in the company, and grant the assurance that you can rely on the control.

TNC 320 and TNC 620 – compact but powerful

The **TNC 320** and **TNC 620** are compact but versatile controls. These contouring controls for universal milling, drilling and boring machines offer the user-friendliness you are accustomed to from HEIDENHAIN. Applications include series and single-part production, research and development, as well as prototypes and pilot plants.

Because of their flexible operational design, the customer can select between workshop-oriented program creation with the familiar HEIDENHAIN plain-language programming language and off-line programming.

Because of its analog nominal speed command output, the **TNC 320** is well suited for the retrofitting of machine tools.

Thanks to its **digital design,** the **TNC 620** has control over the machine's entire drive system.

TNC 640 – the new milling/turning control

The **TNC 640** will be introduced at EMO 2011. The key point: it has been conceived for machines that are capable of performing combined milling and turning operations! It offers powerful functions that enable you to switch between milling and turning mode in the NC program.

You can create turning contours as usual in the HEIDENHAIN plain-language dialog for subsequent machining. Typical contour elements for turning (recesses, undercuts, thread undercuts) are provided. With the aid of turning cycles, you can very easily program even complex turning operations at the machine.

HEIDENHAIN offers the **MANUALplus 620** and **CNC PILOT 620** lathe controls for cycle and CNC lathes.

The iTNC 530 is now available in a new, stainless steel design

iTNC 530 – A 10-Year-Old Success Story

iTNC 530 – milling in perfection

The **iTNC 530** from HEIDENHAIN is the head of the class when it comes to milling controls. It is the optimum solution for companies with particularly demanding production processes. Numerous functions are available, such as for universal milling machines, HSC (high speed cutting), 5-axis machining with swivel head and rotary table as well as 5-axis machining on large machines and machining centers.

Accuracy is of great importance here: perfect surfaces and details are the proof of highest precision. The **iTNC 530** achieves this with optimized motion control, short block processing times and special control strategies.

The **iTNC 530** scores points for process reliability and machining speeds with powerful functions. Examples: the DCM dynamic collision monitoring (option) cyclically monitors the working space of the machine for possible collisions with machine components and fixtures. With the AFC adaptive feed control (option), the iTNC 530 regulates the feed rate automatically—depending only on the respective cutting conditions. This optimizes the machining time and tool life, and reduces machine wear.

The new hardware design was introduced at EMO 10 years ago, and since then numerous successful applications have come about.

A quick look back: when the iTNC 530 was presented in 2001 it marked the first time that the MC main computer of the NC was separated from the CC controller unit and equipped with a powerful Intel processor architecture (iTNC = TNC with Intel processor architecture). The operation and programming were designed to be user friendly and compatible to previous controls, so that the machine operator would not have to relearn the basics of operating the control. Naturally there were also some new functions, and the iTNC completed its tasks on the shop floor faster and with more precision than its predecessors. This provides the operator with several advantages: long programs can be transmitted and edited rapidly, and complicated programs can be tested quickly and easily. Of course all machining processes benefit from the increased power and speed. Whether talking about high speed cutting, demanding tasks for tool and die making, 5-axis machining of 3-D shapes or machining in a tilted working plane, there are no limits to where the iTNC can be used.

The versatility and flexibility of the iTNC during operation is matched by how different and unusual its applications can be. Some examples from recent issues of Klartext illustrate the range of purposes it is used for:

- The Swiss company Robert Ott AG manufacturers high-precision surgical instruments with the iTNC. They have particularly complex shapes and are used in complicated operations. (Klartext issue 52)
- BMW, the car manufacturer, uses the iTNC 530 for large toolmaking. (Klartext issue 51)
- The new company Rolf Spectacles from Tyrol, Austria, produces unmistakable designer eyeglasses with the iTNC. (Klartext issue 54)

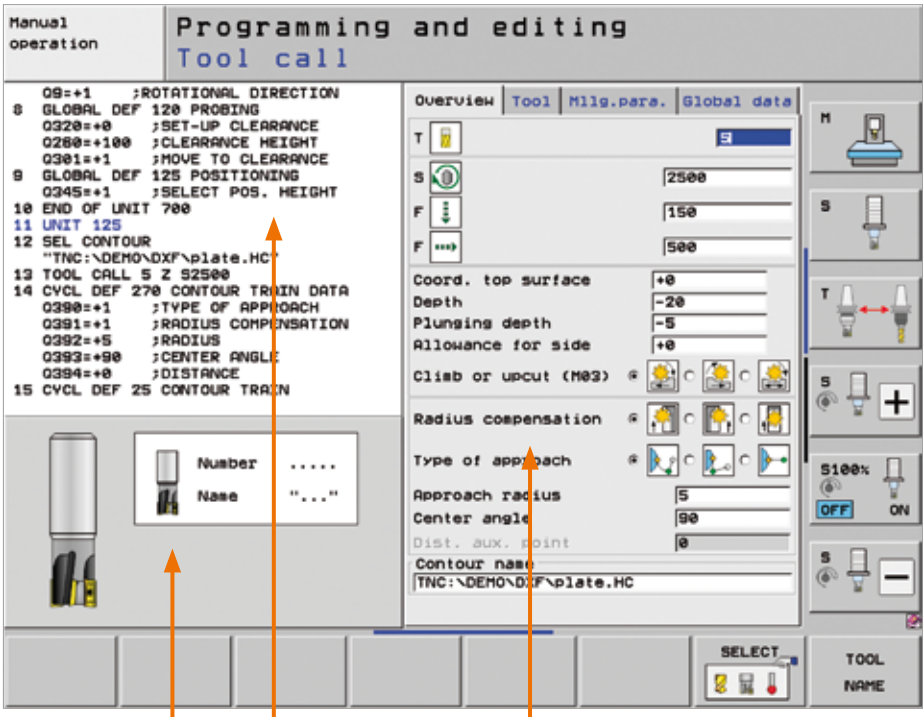
New NC Software 340 49x-07

smart Goes Plain Language and Other Great New Functions of the iTNC 530

Two worlds have come together!

You no longer need to decide whether you want to program in **plain language** or in the **smarT.NC** operating mode. HEIDENHAIN has combined the strengths of both types of programming in a single user interface. Combine the full flexibility of conversational programming, based on NC blocks, with the fast, form-based workstep programming method of smarT.NC. You particularly profit, for example, when you use the machining patterns from smarT.NC, which are more quickly defined as a result of their graphic support. All other machining units available in smarT.NC can also by inserted in conversational programs in the blink of an eye. Handling of DXF files using the DXF converter is also very easy in the smarT.NC mode.

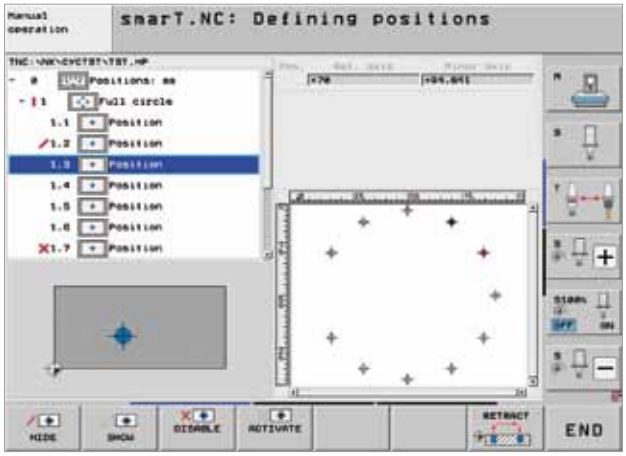
And how does it all come together? Very simple: when the form is saved, the iTNC 530 inserts all conversational blocks necessary for the defined machining operation. You can then edit the program any way you wish, either in the NC block or in the smarT.NC form.



smarT.NC form with the corresponding, unit-specific input fields

Plain-language program

Support graphic for the respective input field



Definition of machining patterns is child's play with the smarT.NC pattern generator

Now it's the iTNC's turn (interpolation turning option)

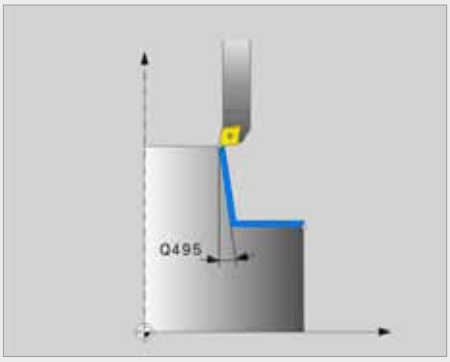
Interpolation turning is a real alternative for certain tasks, such the machining of flanges or shoulders. Here are some of the advantages: alignment of a rotationally asymmetric workpiece for turning is not necessary, nor are any exotic special tools, setup times are reduced, and more.

How does interpolation turning even work?
In interpolation turning the cutting edge of the tool moves on a circle, with the cutting edge always oriented to the center of the circle. This way you can produce rotationally symmetric objects in any working plane.

The new **interpolation turning cycle 290,** which is suited for finishing, does it like this:

- The starting and end point of a rotationally symmetric shoulder is defined.
- The center of rotation is the tool location in the working plane at the time the cycle is called.
- The rotational surfaces can be inclined or rounded relative to each other.

The machining strategy can be chosen flexibly: from the outside in or vice versa, and also from top to bottom or vice versa.

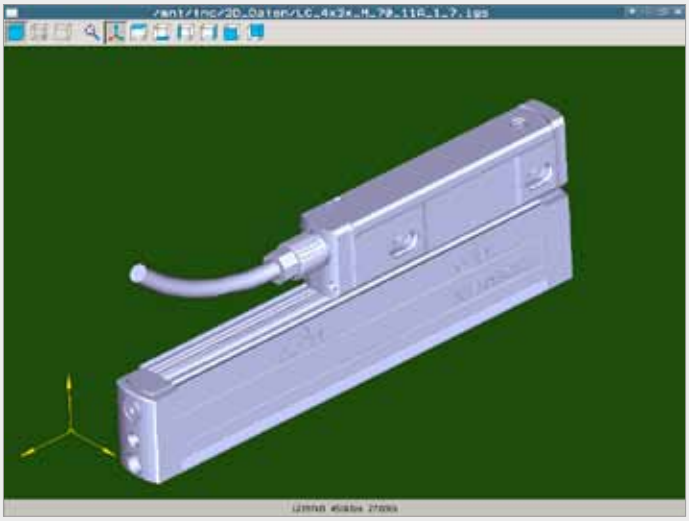


After you have defined the contour start and end, and other parameters, enter the angles of the face surface and circumferential surface (Q495 in figure), as well as the radius of the contour corner, and interpolation turning is ready to go!

"Show me the CAD drawing!" (option)

Sometimes it's not that easy. All you can do then is head for the CAM office. A faster way of seeing it is with the new 3-D CAD viewer for the iTNC 530. Now you can view the CAD drawings on your control's screen. It does not matter whether the file is on the iTNC's hard disk or on a connected drive. This way you can easily check any uncertainties in 3-D models or tooling sketches.

- Select a CAD data file (STEP, IGS or IGES) from the iTNC's file manager, and the 3-D CAD viewer opens automatically.
- Use the shift, rotate and zoom functions for detailed views of the features in question.

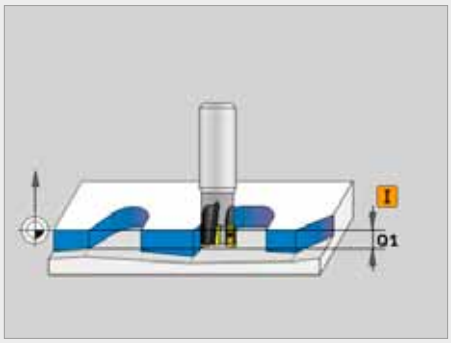


New for the iTNC: the 3-D CAD viewer (only possible with the HSCI version of the iTNC 530 and the new software version 606 42x-02)

New contour train cycle—of particular interest for large-scale mold making in the automotive industry!

New cycles are constantly being added to the iTNC 530 in order to satisfy new demands from industrial customers. Peripheral milling of cutting and bending tools is no exception. The distinguishing feature of these peripheral contours is that tool also needs to advance in the Z axis, i.e. in the tool axis, since the cutting or bending edge does not have a constant Z height, but can vary greatly. The new Cycle 276 CONTOUR TRAIN 3D can create these 3-D contours very easily, particularly if a smaller tool must be used to rework specific areas.

The DXF converter has also been prepared for this: contours or contour sections can now be loaded from plain-language programs created by CAM systems.



- Similar to Cycle 25 CONTOUR TRAIN, which is two-dimensional, the contour to be machined must be defined in a subprogram.
- Specify the approach behavior, machining mode and radius compensation with Cycle 270 CONTOUR TRAIN DATA.
- Call Cycle 276 CONTOUR TRAIN 3D.
- The 3-D contour train can be machined with or without a defined infeed.

Graphic selection of contours (option)

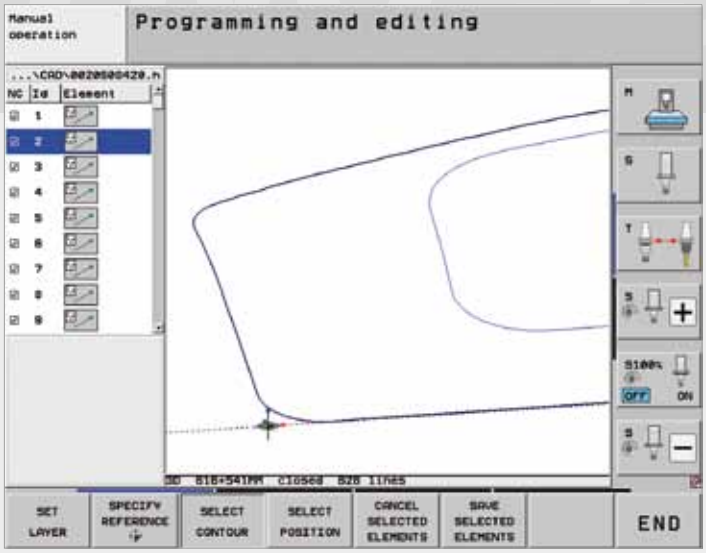
In the DXF converter you want to view a plain-language program that was created off-line in a CAM system? You don't need to head for the CAM office here anymore, either. Because now you can select on you iTNC the tool paths generated by the CAM system.

For several years now the iTNC 530 has been able to select contours from DXF files: you can select sections of the contour and save them as separate NC programs. You can then machine this newly created NC program directly or in combination with the TNC's contour train cycles.

For what? For example, if you need to rework parts of contours with a smaller tool, or even if you just want to rework some parts of a 3-D shape, this feature is simply unbeatable.

What else is new with the DXF converter?

- Quicker capture: Contours and machining positions can be selected faster and with better accuracy with a capture function.
- Faster overview: You can tell which tool can machine the contour in any case. The status bar shows the smallest contained contour radius.
- Rapid selection: You can now choose preselected contours in the tree view.



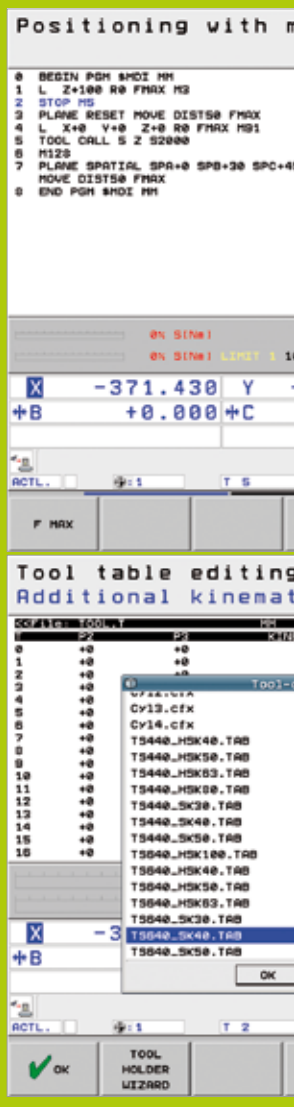
New in the DXF converter: graphic selection of contour trains from the CAD system, with subsequent saving as separate NC program.

DCM is not a luxury, but rather an important contribution to process safety (option)

Since the introduction of DCM dynamic collision monitoring in 2005, this function has become an increasingly important instrument in reducing costly machine downtimes and relieving the machine operator, since particularly in manual traverse the wrong axis-direction key can accidentally be pressed in the daily hustle and bustle at work. In this case DCM behaves admirably: if a collision is imminent, the iTNC first reduces the feed rate, and then stops the axis if the distance becomes too small. The TNC not only monitors the permanent machine components defined by the machine tool builder, but also fixtures, tools and tool carriers.

There have also been interesting further developments for this function:

- Automatic activation and deactivation of fixture situations
- With the two new NC functions SEL FIXTURE and FIXTURE SELECTION RESET you further increase safety in automated production, by activating or deactivating previously saved fixture situations in automatic operation. You can also activate specific fixtures from pallet tables.
- Tool carrier management
- You can now also monitor tool holders: Simply assign an appropriate collision object to the tool. The matching preview image is displayed in the tool table, making the assignment that much easier.
- Collision monitoring for multiple task tools
- You work with multiple task tools? Each stage of such a tool is monitored, and you can see this in the kinematics view.



New functions for enhanced tool management (option)

Do you sometimes exchange data via CSV files? Reading and exporting of CSV files now also works with enhanced tool management. CSV is a text-file format (CSV stands for comma separated values), and can also be opened and saved by MS Excel. Data exchange with this function is especially easy if you measure and calibrate your tools with external presetters.

Don't delete too much! Tool data can now be deleted more quickly and with greater care. The tool data to be deleted is shown in a pop-up window, giving you the opportunity to make sure that no important data is deleted by accident.

New cycles

New engraving cycle 225
Texts and serial numbers can be produced easily. You enter the text via a text parameter in the cycle. The text can be engraved in a straight line or along an arc.

Thread milling cycles 262, 263, 264 and 267
A separate feed rate is now available for tangential entry into the thread. Particularly with small thread sizes this makes it possible to select a higher subsequent cutting feed rate, which reduces production time.

KinematicsOpt cycles 451 and 452 (option)
Faster optimization algorithms reduce the time for measurement, as does the fact that position optimization is now performed simultaneously with angle optimization. Furthermore, the ascertained offset errors are now available as result parameters, permitting subsequent program-controlled evaluation.

Even more improvements

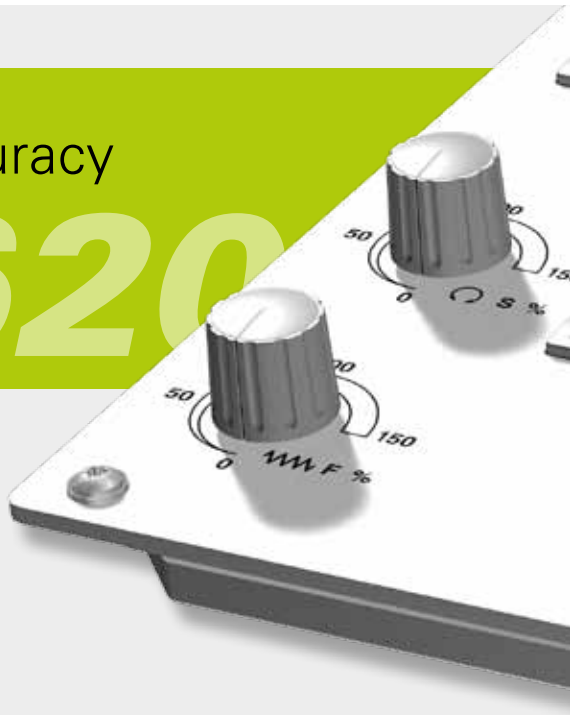
Global Program Settings (GS) (option)
In the global program settings (GS) form there is now an additional switch with which you can specify whether the values traversed in the virtual axis direction should be reset upon a tool change.

File management: support for ZIP archives
In the iTNC 530's file manager you can now create ZIP archives, in order to archive files of completed jobs, for example. Of course you can also open existing archives and extract files from there.

Working with pallets
Selected workpieces can be hidden during tool-oriented machining in combination with the pallet table. A new keyword is available for this.

TNC 620 – Paths to the Higher Accuracy

TNC 620



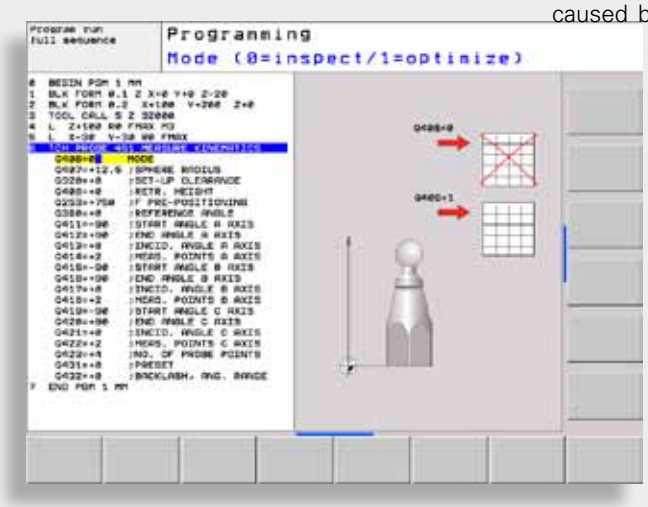
The flexible operational strategy and the workshop-oriented programming in the HEIDENHAIN plain-language format or over offline programming qualify the TNC 620 for use on universal milling, drilling and boring machines. The enormous performance range of the TNC 620 make it suitable in many different areas, such as series and single-part production, tool making, pilot plants as well as training and education facilities. The KinematicsOpt function now also makes higher machining accuracies possible with the TNC 620.

The TNC 620 is now available in a new, stainless steel design

KinematicsOpt – also for the TNC 620 (option)

The optional software function **KinematicsOpt** known from the iTNC 530 is now also available for the TNC 620. The OEM simply needs to prepare the machine for it. KinematicsOpt can now ensure continuously high accuracy of workpiece machining for the TNC 620. A 3-D touch probe cycle fully automatically measures the rotary axes present on your machine.

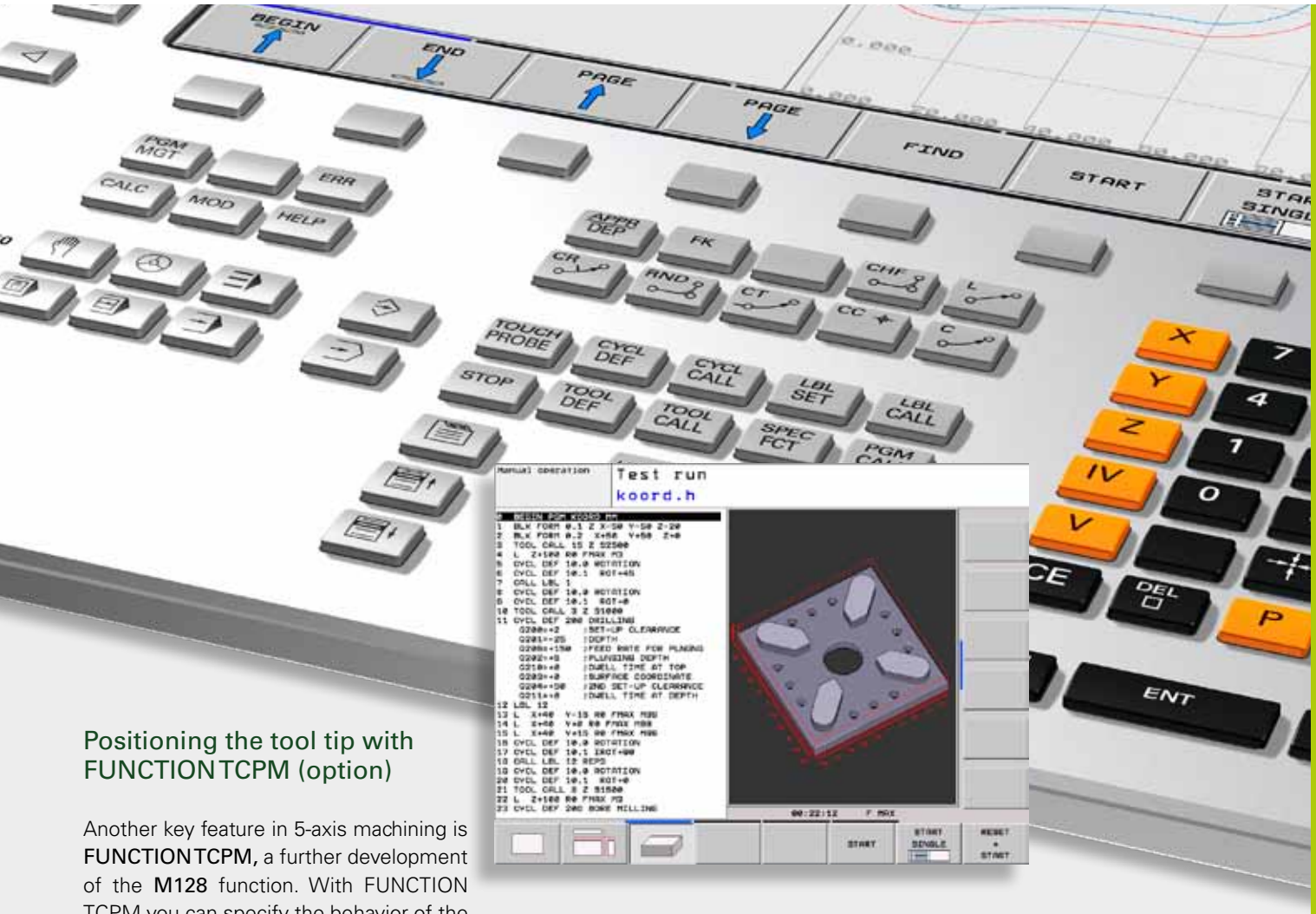
Deviations in the kinematics are a frequent problem in production, and are often caused by mechanical loads or temperature variations. Also, changes in the kinematics are quickly detected and compensated with the aid of a high-accuracy HEIDENHAIN touch probe and the highly precise and very rigidly mounted HEIDENHAIN calibration sphere. The machine can thus move the tool more accurately along a programmed contour. The result: increased efficiency and continuous precision.



And here's how it works: regardless of whether the rotary axes are mechanically in the form of a table or head, a calibration sphere is mounted anywhere on the machine table. During cycle definition you simply define for each rotary axis the area to be measured and the resolution during measurement.

The TNC then handles all the rest for you: the measuring sequence, the calculation of the statistical tilting accuracy averaged over the smallest spatial error, as well as entry of these values in the appropriate machine constants.

In the next calibration process you can immediately use the determined compensation values again. You only need to define a code number with which you protect your data. The data can then only be overwritten if you confirm this action by entering the code number. A comprehensive log file also stores your data. That way you can access both the entered and the measured values, the optimized scatter (the measure for the statistical tilting accuracy) and the actual compensation values at any time.



Positioning the tool tip with FUNCTION TCPM (option)

Another key feature in 5-axis machining is **FUNCTION TCPM,** a further development of the **M128** function. With FUNCTION TCPM you can specify the behavior of the TNC 620 when positioning rotary axes, and in contrast to M128 you can even define the operating methods of various functions yourself:

- Whether the programmed feed rate refers to the tool tip or to the contour: F TCP / F CONT.
- The interpretation of whether the rotary axis coordinates programmed in the NC program are axis angles or spatial angles: AXIS POS / AXIS SPAT.
- Whether axis or vector interpolation is used between the start and end position: PATHCTRL AXIS / PATHCTRL VECTOR. Axis interpolation moves the tool point on a straight line. Vector interpolation moves the tool point on a straight line and also the tool circumference along a flat plane that connects the start and end positions.

They make working even easier: even more new functions of the TNC 620

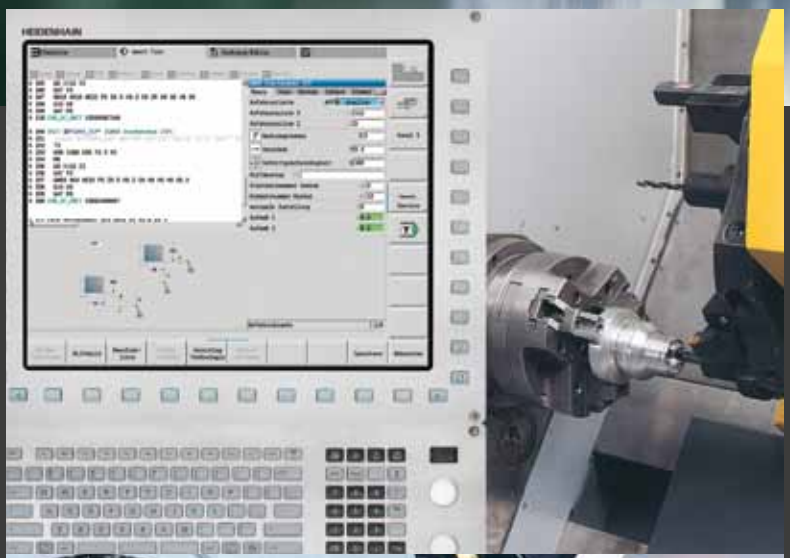
- **Verification graphics**
  In the verification graphics you can set the simulation speed via soft keys. You can then reduce the speed at tricky locations, in order to check the finished piece better. Not only can the speed be adjusted now, but the entire simulation was improved, and a mouse can now be used for viewing (tilting, panning, zooming). Also, tool movements can be displayed. and a new scale is displayed around the workpiece blank.
- **Touch probe cycles**
  Touch probe cycles can now also be used in the tilted working plane.
- **Tool tables**
  There is now an import function for importing the tool tables of the iTNC 530 into the TNC 620 and converting them.
- **Q-parameter programming**
  You can now work with local QL and non-volatile QR parameters.
  - Local QL parameters are effective only in the program in which they were defined.
  - Non-volatile QR parameters are effective until they are reset, even if the power is interrupted.
- **Automatic tool change if the tool life expires**
  With the new M101 miscellaneous function a replacement tool can now be inserted automatically after the programmed tool life expires in order to continue machining.

Additional hardware version

Starting from EMO 2011, there will be an additional hardware version of the TNC 620, with a full-fledged keyboard. This makes the control more attractive for DIN/ISO programmers as well.

Whether Simple or Complicated: Manufacturing Turned Parts More Easily and Economically

Of course there are also new things to report about the HEIDENHAIN controls for lathes at EMO 2011. The MANUALplus 620 and CNC PILOT 620 offer many new features that support complicated machining procedures and the functions of new machines. As is HEIDENHAIN's custom, operation and program creation are to remain simple. What's new is the possibility of using functions for rear-face machining for full-surface machining of turned parts in a single setup.



Keeping a good overview

The new design line from HEIDENHAIN is also apparent on the MANUALplus 620 and CNC PILOT 620 lathe controls—for a better overview and improved safety in operation.

You are working on your lathe's control and want more detailed information about a particular function?

With **TURNguide,** the new, integrated, context-sensitive aid, you are quickly presented with information from the HEIDENHAIN User's Manual—directly on the control's screen.

You want a better overview of all data for a particular project?

Not a problem with **project directories!** In separate folders you can easily store all NC programs, cycle programs, DXF contours and ICP contours. Organizing and sorting of the data goes very quickly with the optimized functions for cutting, copying, pasting and deleting.

The lathe controls from HEIDENHAIN are now available in a new, stainless steel design

New possibilities for easy manufacture of complicated workpieces

How can you turn an extrusion shaft or spiral conveyor with variable thread pitch?

You can now specify a variable thread pitch in the **expanded thread cycle.** This can be done in the Teach-in and smart.Turn modes, as well as in DIN/ISO.

You want to produce key flats on your lathe through polygonal turning?

Then use the new **G270 spindle synchronization** option. The function synchronizes the shaft speeds of two or more spindles so that they rotate synchronously. Additionally, a transmission ratio or specific offset can be defined. This option permits many further complicated production sequences without manual rechucking, since the synchronism can be used to transfer the workpiece between the main and counterspindles. When combined with the hobbing cycle you can also mill external teeth and profiles.

Do you tap "by touch" or with defined chip breaking?

The control handles chip breaking with the new **G73 and G36 tapping cycles.** With the "P" parameter you define the chip breaking depth, and with "I" the retracting length.

What else would you like to teach the control?

Two very practical functions have been added to the Teach-in operating mode: deburring cycles are now available for the C axis, and the engraving cycles can now also be used during Teach-in.

Avoid obstacles easily

You need more tools for machining than your turret has room for?

Starting immediately, the lathe controls can also manage **manual change systems** for tool turrets. These special tool holders have clamps for tool inserts. The tools can be exchanged during NC machining with just a few simple steps. This way more tools are available for machining than the turret has pockets.

What can you do if resonance oscillations lead to chatter marks?

The new **G924 fluctuating speed** function modifies the shaft speed within a specified range and in a defined interval. Resonance oscillations can be suppressed with the correct settings.

How can you easily correct the widths of keyways?

With the new G976 function you can program misalignment compensation in the X, Y and Z axes. This function can also be used for the production of taper shafts.

Rear-face machining

The MANUALplus 620 and CNC PILOT 620 lathe controls now make it possible to fully machine workpieces on the front and rear faces. Numerous functions are available for counter-spindle and rear-face machining. For example, the "WP" parameter for the active tool spindle was added to the cycles. The current tool spindle is then entered in the cycles as a proposed value when making a new entry. The set-up functions also take the current tool spindle into account, and can be used for the main spindle and counterspindle.

Naturally the ICP editor and the simulation also support rear-face contours now. Other functions, such as for conversion, mirroring of traverse paths and tool dimensions, mirroring and shifting of workpiece blank and finished contours, and G and M functions, facilitate program creation.

The new "traverse to a dead stop" function is used for transfer of the workpiece by the second traveling spindle or for pressing the tailstock against the workpiece.

Through the sleeve monitoring you can specify the maximum contact force for a selected axis. For example, this lets the counterspindle be used as an intelligent tailstock.

Creativity Meets Precision – How the iTNC gave a Start-Up Greater Productivity

Would you dare to present a new product on a market filled with many established brands? You would need a good idea, lots of courage, and no compromises in your demands. This particularly applies if the object—in the middle of your face—strongly emphasizes your individuality: we're talking about eyeglasses here. An everyday product with the potential to become something exotic!

The material itself is already unusual: wood, with a figure that is as unique as each human fingerprint. This results in glasses frames that radiate individuality. But success can also be quite demanding: basic prerequisites are excellent design and quality without compromise. The top models must rise above the other eyewear, mostly produced in Asia, through their combination of low weight, no screws, unusual surfaces and perfect functionality.

The courage for a new start

Roland Wolf, his partner Marija Iljazovic, and her brother Martin Iljazovic proved their courage in 2008 when they founded their own brand, "Rolf Spectacles." Driven by the idea of authentic wooden glasses, they began their R&D in the basement of their parents' house. Producing precisely manufactured wooden glasses required equipment that was either financially far beyond these young entrepreneurs' means or that was not even available at all. However, they were able to rely on unconditional self-confidence and friends with expert knowledge. New production equipment, with at least as much individuality as the glasses they produced was conjured with components from used machines and "misused" machines.

Success "Made in Tyrol"

And then the time had come! Their ambitious goal of individuality and quality without compromise, "Made in Tyrol," was ready for the market. "When you hold the glasses in your hand, and open and close the temple arms, you then know all you need to know," Wolf laughs today. "Everything has to come together in this frame ..." he adds, and could enthusiastically expound for hours about all the innovations.

So the team headed for Paris in 2009, together with plenty of enthusiasm. They presented their new product to the market at SILMO, an international optics trade show. In the competition for innovative design, they spontaneously won the SILMO d'or, the equivalent of an Oscar trophy for opticians. In 2010 they received a Red Dot Design Award, a coveted award for outstanding design. The product became famous practically overnight, and so economic success soon came their way.

"The machine and control definitely fulfilled our demands."

Martin Iljazovic, Production Manager

Roland Wolf (right) and Martin Iljazovic (left)

The iTNC 530 machines exotic wood

The unique machines did their job well. But the time-consuming manual steps got in the way of the growing unit numbers. What could be done? The decision was quickly made to find a powerful NC solution that was suited for the precise machining of exotic woods.

Thomas Dobler from Deckel Maho immediately became excited about the unusual request, and presented several interesting proposals to the young entrepreneurs. In the end, a DMU 40 mono BLOCK with HEIDENHAIN iTNC 530 got the nod to manufacture the high-quality products.

The iTNC 530: greater productivity and higher precision.

This was a good decision! Nobody in the Rolf Spectacle team had any idea about 5-axis machining, not to mention CNC programming. But as pragmatically and enthusiastically as they approached everything else, Martin Iljazovic delved into the subject. The workshop-oriented programming in the HEIDENHAIN plain-language format made it easy. Now he navigates through the program with ease, and praises the structure function with which he quickly finds his way in long programs.

But of course it is somewhat unusual that the machine tool is being used here, since after all the wooden structures being machined are very delicate. Isn't that like cracking a nut with a sledgehammer? Martin Iljazovic is convinced that they made exactly the right decision: "We must work very precisely, particularly on the high-quality hinge. We can optimize use of the machine regarding capacity and machining speed, and the previously complex handling with various fixtures is now covered by just one setup."

The high accuracy of the HEIDENHAIN control is well suited for wooden products, which do not accept any compromises.

Automation meets uniqueness

Each wooden frame is unique – even the automated machining of time-consuming production steps can't change that. But it helps to bring about large unit numbers while fulfilling the high quality expectations.

Optimum productivity at Rolf Spectacles begins with the creation of the machining programs for the various eyeglasses models. As soon as the design for a pair of glasses has been decided upon, a CAD/CAM system is used to prepare it for machining.

One intention of the automated machining is to completely remove humans as a source of error. Before machining begins, the iTNC 530 uses a HEIDENHAIN touch probe and the appropriate cycles to check the correct position and orientation of the blanks, which are manually placed on a suction fixture. This idea also shows these youngsters' creativity: the flexible wooden blanks are held in place in the fixture by means of lower air pressure. This way the wood sits in an exactly defined position, which is a prerequisite for precise machining.

Having the automated tool measurement detect which model is to be machined was also a great idea: only a single program is needed for the around 30 different models of eyeglasses. The HEIDENHAIN touch probe simply measures the distance to a defined hole on the fixture being used, and the control then automatically selects the correct machining program.

Many steps of the production process were easily solved with features such as the preset table for managing datums, Q-parameter programming and the integrated arithmetic functions.

Plain-language programming made it easy to learn NC machining.

A HEIDENHAIN touch probe on a HEIDENHAIN control – a good pair for automated processes.

The glasses frames are milled from the workpiece blanks using 5-axis machining. Since very short tools are used for the cutting of the fine structures, dangerously close encounters can occur in the machine's work envelope. Therefore, Rolf Spectacles makes sure to use the all-encompassing dynamic collision monitoring (DCM option) of the iTNC for certain positioning movements.

The delicate wood also presents the machine tool with another challenge: The cutters being used have very small radii, and no cutting data for this atypical material can be found. For this reason, and to increase reliability of the production process, the tools are constantly monitored for breakage.

Another disruptive factor is the fine wood dust, which can impair the machine's functions. Rolf Spectacles has modified all of its machines, and in this case the DMU 40 from Deckel Maho was equipped with a suction bell, which artfully solved this problem.

Hitting the bull's-eye

The decision for a 5-axis model was a large step for the authentic newcomers from Tyrol, who had to finance their success with their own means. At the time, the DECKEL MAHO machine was considered to be an extremely large investment, Martin Iljazovic explains.

In the meantime, everyone is convinced that this decision was correct. They are particularly excited about the technology they purchased: the dialog-guided operation of the iTNC 530 facilitates adaptations and optimizations for a material for which there is nearly no empirical data. The high accuracy of the HEIDENHAIN control is well suited for wooden products, which do not accept any compromises.

The company is doing quite well today, with significantly higher unit numbers and fewer scrap pieces. The new productivity has also given rise to new knowledge. But that is not enough: constantly striving for new ideas, the team is already working on new solutions for the future, since there still seems to be a lot of potential in the new machine, controlled by the iTNC 530.

In any case, Wolf and Iljazovic are sure that they hit the bull's-eye, regarding quality and productivity, by acquiring the DMU 40 controlled by an iTNC 530, because Rolf Spectacles does not make any compromises!

ROLF – Roland Wolf KG

A start-up company, nestled in the mountains of Tyrol, producing high-quality and very light wooden frames for eyeglasses.

www.rolf-spectacles.com

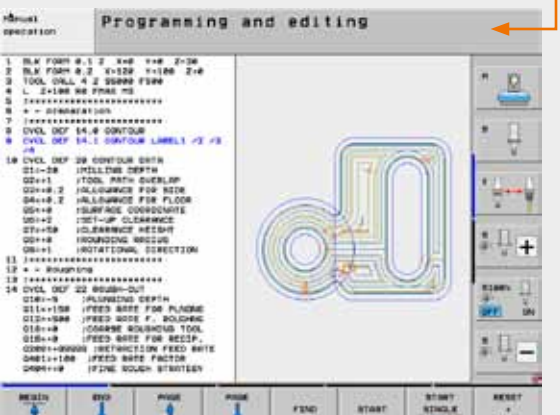
HEIDENHAIN Training for the Real World

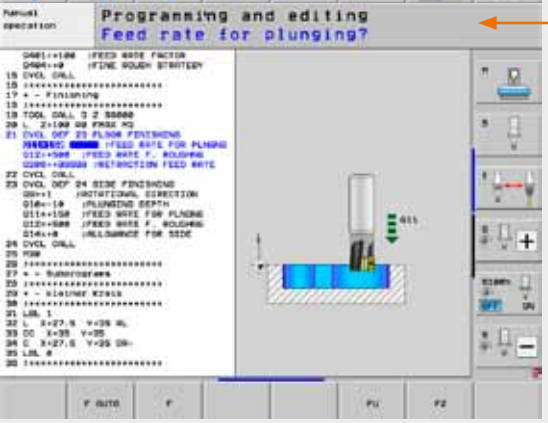
Easy Programming of Pockets and Islands with SL Cycles

The practical skills of the TNC users are at the center of the popular programming training courses; that is why the course rooms are perfectly equipped with a programming station for each participant. Whenever possible, the trainers gladly answer the participants' individual questions, which often arise from their own practical experience. In a new series called "HEIDENHAIN Training for the Real World," Klartext will use selected features to demonstrate how the practical knowledge is taught in the course, including with an interactive screencast on the web!

The Klartext staff wanted to experience this recipe for success live, and simply visited a course in progress. SL cycles, taught in the basic course, were the current topic.

There are lots participants in the course room this morning, and Christina Lohmayer, course leader at HEIDENHAIN, is the only woman in the room. From her own experiences on the shop floor she knows the best way to approach the topic: using a workpiece drawing of a contour pocket, she first explains the terms and the task. Before the participants get to work, each part of the machining program's structure is explained on the screen via an overhead beamer. Then the task itself is presented:





The participants are to program a new contour pocket themselves:

- First all closed contours must be defined in their own subroutines.
- For each subroutine the radius compensation must be used to specify whether the tool is to move within or outside of the contour.
- At the beginning of the program, in Cycle 14, all subroutines needed for the contour pocket are listed.
- Then, in Cycle 20 Contour Data, all geometric information is specified for the contour pocket. To improve understanding, the TNC offers a support graphic for each parameter.
- Cycle 22 Rough-out is used to define the infeed and feed rate for the currently inserted tool.
- If a finishing allowance was defined in Cycle 20, then in the current example Cycles 23 Floor Finishing and 24 Side Finishing must follow, with information about the infeeds and feed rates for the finishing operations.

Each participant can test his machining program using the simulation. First the Test Run operating mode is switched to. Then the written program is selected and loaded via the file manager. After the RESET + START softkey has been pressed, the TNC begins to simulate the machining steps with detailed graphics.

If the result doesn't quite match the specifications of the workpiece drawing, or if other questions arise, Ms. Lohmayer quickly provides assistance. With practical tips every participant soon ends up with the right result, and can subsequently put the new knowledge to work back at his job.

The Klartext staff is sure that the participants remember the courses well. Anyone who hasn't taken the basic course in Traunreut yet might want to after checking out the Klartext eMagazine. In a brief training unit, Christina Lohmayer demonstrates how to create machining programs with SL cycles – of course at no charge!

www.heidenhain.de/klartext

So what are SL cycles?

SL cycles enable you to form complex contours by combining up to 12 subcontours (pockets or islands). You define the individual subcontours in subprograms. The TNC calculates the total contour from the subcontours (subprogram numbers) that you enter in Cycle 14 *CONTOUR*.

New in Time for EMO

www.tnc640.com