



HEIDENHAIN

Klartext

The TNC Newsletter



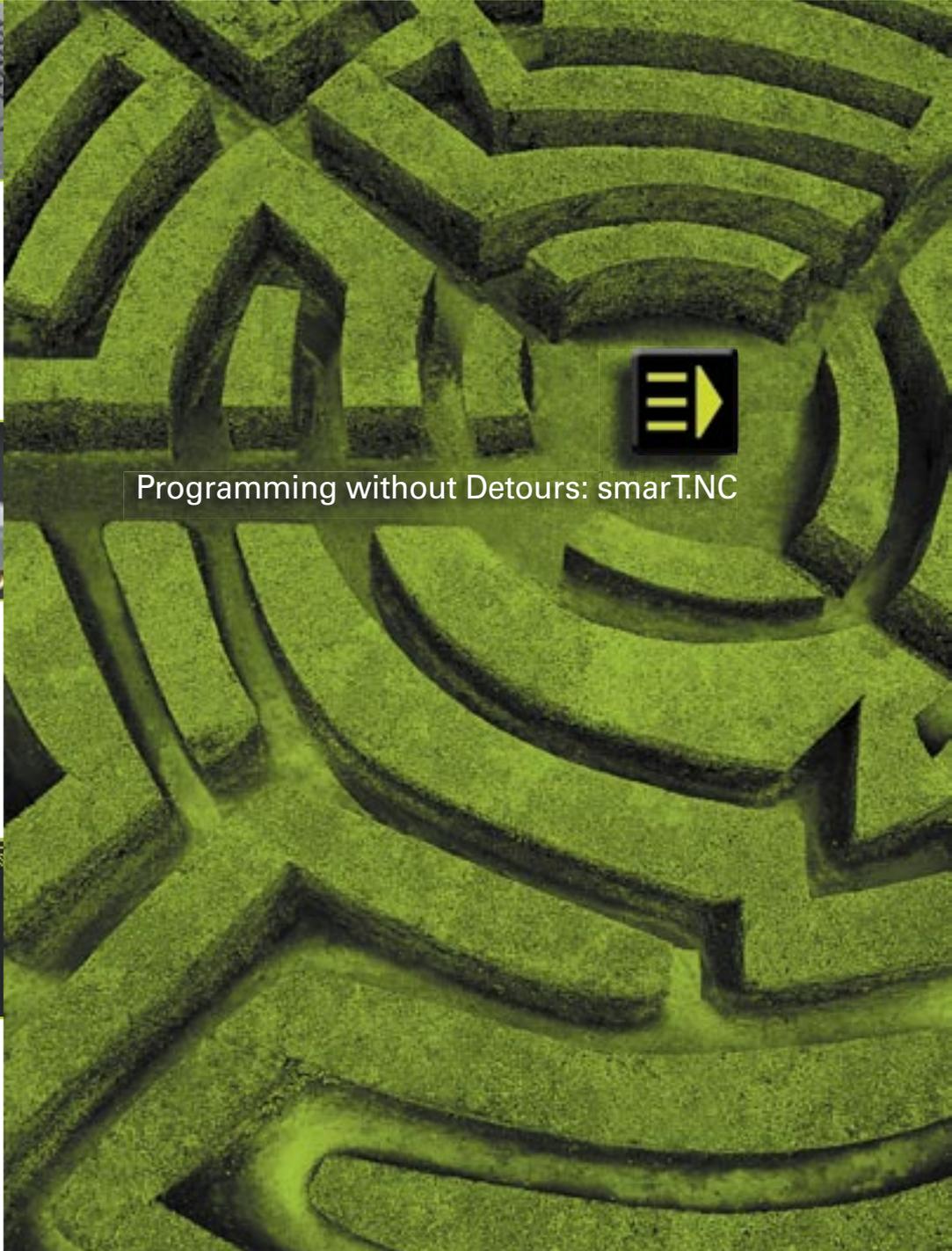
**New iTNC 530
Functions**



**Dynamic Collision
Monitoring**



DXF Import



Programming without Detours: smarT.NC

Issue 44 • 8/2005

Editorial

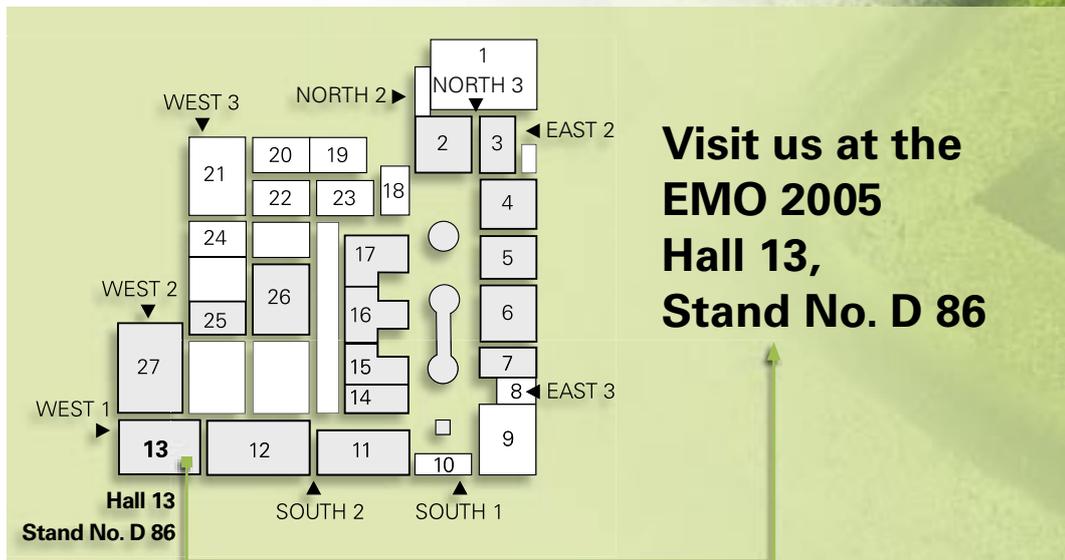
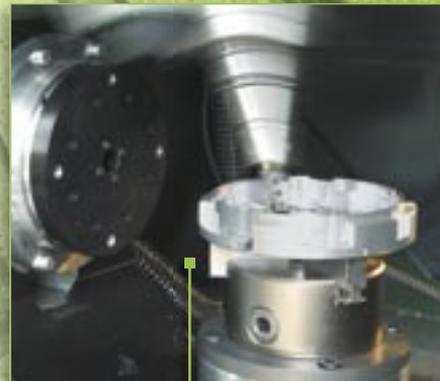
Dear Klartext Reader,

This issue of Klartext is completely dominated by the EMO! Having already introduced the smarT.NC operating mode, HEIDENHAIN is now setting off a burst of fireworks with innovative new features all around the TNC.

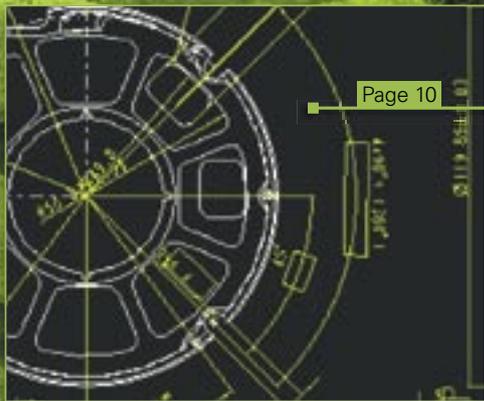
A special highlight for machine manufacturers and operators is the Dynamic Collision Monitoring (DCM) function for the iTNC 530. DCM protects the operator as well as the machine from impending collisions with machine components within the working space. DCM therefore increases the level of safety for both the operator and the machine. This helps to prevent machine damage, which would result in costly downtimes.

The importation of DXF files is another user-oriented function of the TNC. The DXF file is loaded from the network or a USB stick, and can then be opened in the file manager, just like an NC program. The new, compact TNC 320 will be presented at the EMO as the successor to the TNC 310. The compact design of its keyboard, display screen and modern technology are oriented on the upper range of contouring controls from HEIDENHAIN. Any machine operator familiar with the TNC 400 series and the iTNC 530 will soon feel at home with the TNC 320 as well.

Have a good read!



Content



Page 10

Page 12



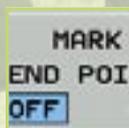
New General Programming Functions

4



New Programming Functions for smart.TNC

6



New Functions for Conversational Programming

7



New for the iTNC 530: DCM—Dynamic Collision Monitoring

8



The iTNC 530's DXF Import Function

10



New Programming Station Functions

12



Addition to the Digital Readouts—the New ND 780

13



TNC 320—The New and Compact Contouring Control

14

Imprint

Publisher

DR. JOHANNES HEIDENHAIN GmbH
Postfach 1260
D-83292 Traunreut
Tel: (0 86 69) 31-0
www.heidenhain.de
info@heidenhain.de

Editor

Frank Muthmann

Klartext on the Internet

www.heidenhain.de/e0.htm

Layout and Design

Expert Communication GmbH
Inselkammerstraße 10
82008 Unterhaching/München
Tel: (0 89) 66 63 75 0
info@expert-communication.de
www.expert-communication.de

Introduction

The iTNC 530 from HEIDENHAIN has proven itself for years as a versatile contouring control for milling, drilling and boring machines as well as machining centers. Along with HEIDENHAIN's plain-language conversational programming for the shop-floor, the iTNC 530 is characterized by many **useful functions** and **innovative features**. To name just a few, these include:

- Exact tool guidance with five-axis machining
- Simple tilting of the working plane
- Praxis-oriented setup functions
- Highest contour accuracy for HSC milling
- Extensive machining cycles
- Useful programming support through unambiguous function keys, free contour programming and support graphics
- Upwardly compatible machining programs
- External programming and fast data transfer

The success story of the iTNC 530 also includes **smarT.NC**—the new operating mode from HEIDENHAIN. It represents another step toward a user-friendly interface for shop-floor programming. Well-structured input forms, straightforward graphic support, and comprehensive help texts combine with the easy-to-use pattern generator to form a compelling programming environment.

New functions for the iTNC 530

Of course there is always room for further development, improvement and simplification. The NC software 340 49x-02 for the iTNC 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. The most important functions are:

- New programming functions
 - General (page 5)
 - smarT.NC (page 6)
 - Plainlanguage (page 7)
- DCM (Dynamic Collision Monitoring) (page 8)
- Import of DXF files (page 10)

Please note:

The new NC software 340 49x-02 only supports the BF 150 flat-panel display, and only runs on the MC 422B and MC 420 main computers (on the MC 420 without DCM).

Function upgrades: Separation of error fixes from improvements in function

Until now, each NC software update always contained all new functions along with the error fixes. As of NC software 34049x-02, error fixes and software improvements are separated. In the future, updates of NC-software will contain only error fixes.

New functions certainly offer added value in user-friendliness and operational reliability. Of course you also have the opportunity to purchase these new functions after a software update: These improvements will be offered as "feature upgrades," and are enabled via the Feature Content Level option.



New General Programming Functions

Numerous new programming functions of the new software assist you at the machine or at the programming station.

Contour pockets: Subcontours can have different depths (upgrade function)

The powerful contour pocket function is available for machining complex contours. In smarT.NC you can define up to 9 subcontours (pockets or islands), and in plain-language programming up to 12 subcontours, for machining a contour pocket. These subcontours can contain up to 8192 contour elements. Until now the depth of a pocket was globally specified. Now you can assign a separate depth to each subcontour. If the subcontour is an island, the iTNC interprets the "depth" entered as the height of the island.

Separate machining units are now available in smarT.NC for fine roughing, side finishing and floor finishing, thereby complementing the machining of a contour pocket.

Cutting speed *vc* as alternate entry to the spindle shaft speed *S*

When selecting a tool, you can now enter a cutting speed *vc* in m/min instead of the spindle shaft speed *S* in rpm.

Freely definable tables in form view

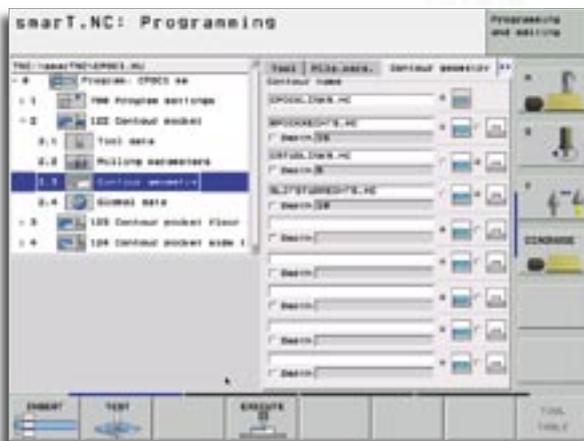
The iTNC can now show all freely definable tables, meaning all tables of file type .TAB, in a form view. This view is especially useful for large amounts of data.

Support of USB block devices (upgrade function)

The single-processor version of the iTNC now supports memory sticks, hard disks and CD-ROM drives via the USB port. This makes data transfer easier if your TNC is not connected to your company network via the Ethernet interface, also a standard feature.

Expanded look-ahead

In order to machine contours rapidly, the control must calculate the geometry ahead of time. This is necessary in order for the iTNC 530 to detect directional changes in time, so that the affected axes can be braked and accelerated optimally. Usually, calculating 256 blocks ahead suffices completely. Today's computing power of NC controls, however, makes it possible to calculate further in advance. This fact is taken advantage of in the new NC software 340 49x-02. The machine manufacturer can now parameterize the look-ahead via a machine parameter. The maximum value is 1024 blocks, i.e. four times as many blocks are calculated as previously. This will, for example only have an effect in NC programs with extremely short traverse blocks.



New Programming Functions for smarT.NC

The new software for the iTNC also contains new features and expanded functions for smarT.NC.

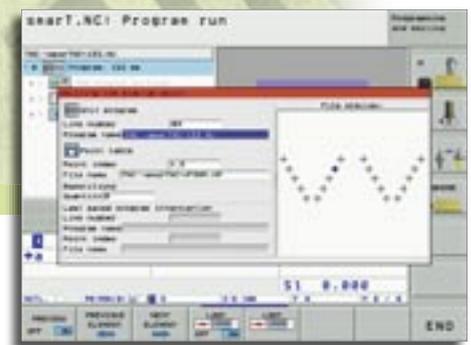
- **Coordinate transformation (upgrade function)**
Forms are now available for the mirroring, rotation, scaling and datum-shift coordinate transformations.
- **PLANE function (upgrade function)**
A form is available for the PLANE function for tilting the working plane.
- **Block scan with graphic support (upgrade function)**
Since the introduction of the pattern generator in smarT.NC, a powerful function offering flexibility and graphic support of machining positions has been available. Machining positions can be entered blockwise in point tables, and can therefore easily be used at various locations for separate machining operations.

Now the re-entry into a machining program has been adapted to this flexible function: smarT.NC detects your entry into a unit in which machining positions are defined, and symbolically displays the positions in a preview window. You can now select via soft key at which position you would like to begin machining. smarT.NC then automatically inserts the appropriate tool and traverses to the selected position – of course only after you have confirmed these actions.

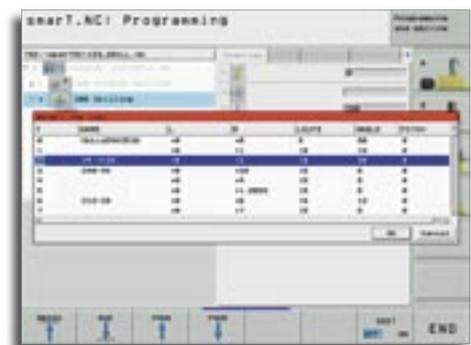
- **COPY/PASTE one or more units**
Using the CTRL+C and CTRL+V shortcuts you are familiar with from Windows, you can copy an entire unit and insert it at any other location in the program. Naturally the SELECT BLOCK function you know from conversational

programming is available for the copying and pasting of multiple units.

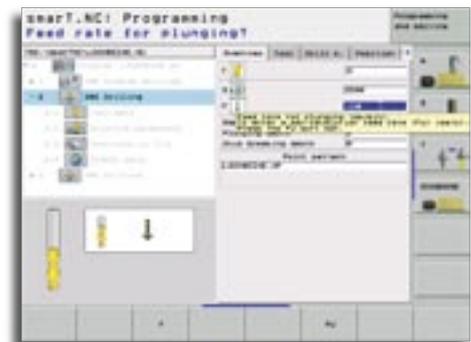
- **Feed rate can also be entered as $F_z/F_u/F_{MAX}$**
When specifying the feed rate, you also enter a revolution feed rate F_u in mm/rev or a cutting feed rate F_z in mm/tooth rather than the normal feed rate in mm/min.
- **Tool data can be edited during tool selection**
In the pop-up window for tool selection you can now also change the tool data shown there. The TNC automatically saves these changes in the tool table TOOL.T.
- **Axis keys can also position the cursor within the forms**
Experienced conversational dialog programmers were unhappy about not being able to use the orange axis keys for contour programming in smarT.NC. This possibility is now available in the TNC forms, so that the operation is analogous to conversational programming. The orange I key (switch to incremental) and P key (switch to polar coordinates) keys can now also be used correspondingly.
- **Automatic transfer of the workpiece blank into the contour program**
When a new contour program is created, smarT.NC automatically transfers the workpiece-blank definition into the unit program. This transfer can be updated via soft key at any time.
- **Incremental entry of machining positions**
When defining the positions directly in the form for each machining unit, the positions can now also be entered incrementally.



Mid-program startup with graphic support

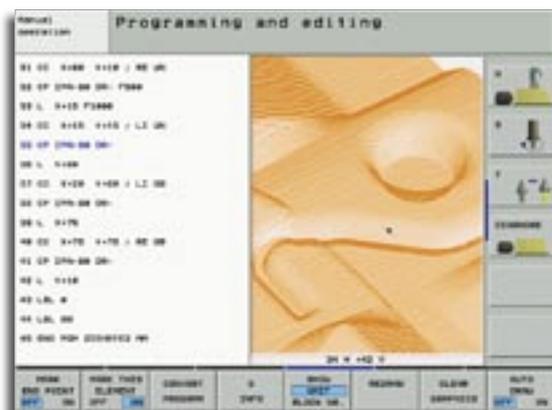


Editing of tool data



Tooltips on mouse over

- **Tooltips appear when working with the mouse**
The TNC displays a tooltip if you hold the mouse over an entry field or selection box for more than one second. Tooltips contain information or important notes about the respective function.



New 3-D line graphic with powerful zoom function

New Functions for Conversational Programming

Conversational programming is the soul of the TNC. That is why software developers from HEIDENHAIN invest so much effort in the continual improvement and development of this type of programming.

- **Touch-probe cycle for global setting of touch-probe parameters (upgrade function)**
In order to more quickly optimize programs containing multiple probing cycles, a cycle that globally overwrites the following probing parameters is now available:
 - Positioning feed rate
 - Positioning feed rate = Set rapid traverse
 - Angle updating ON/OFF
 - Program interrupt on error ON/OFF
- **Point filter for externally created NC programs (upgrade function)**
With this function you can filter contours that were created using external programming systems. The filter function creates a copy of the original program, and then adds any points required by the filter parameters that you set. This smoothes the contour, which results in the program running more quickly and with less jerk.
- **3-D line graphics, with three-dimensional display of the tool center point path (upgrade function)**
Use the 3-D line graphics to have the TNC show the programmed traverse paths in three dimensions. A powerful zoom function is available for recognizing details quickly. You should especial-

ly use the 3-D line graphics to inspect programs created externally for irregularities before machining, in order to avoid undesirable traces of the machining process on the workpiece. Such traces of machining can occur when points are output incorrectly by the postprocessor. In order to find the error location quickly, the currently active block of the 3-D line graphics appears in a different color in the left window.

- **Manual traverse in the active tool axis system (upgrade function)**
Use this function in the Manual and Electronic Handwheel operating modes, as well as during a program interruption in the Program Run, Full Sequence operating mode, to move the tool with the external direction keys or the handwheel in the direction in which the tool axis is currently pointed. This function is especially useful if you need to retract the tool in five-axis programs, and do not have detailed information about the active working plane.
- **Simplified management of the preset table**
As the result of many customer requests, the handling of the preset table was simplified. You access it directly now with the PRESET TABLE soft key in the Manual operating mode. There are also new possibilities for saving values in the preset table:
 - Directly assume the actual position of the tool (dial gauge) as the new preset
 - Assign any value to the actual position of the tool (dial gauge)

- Incrementally shift a preset already saved in the table
- Directly enter the new preset without calculation of the kinematics
- **Taking an active basic rotation into account in the manual probing cycles**
The probing cycles for setting up the machine—available in the Manual operating mode—now take any active basic rotation into account during the probing procedure. The touch probe then moves towards the workpiece at an angle, if you have already determined a basic rotation before reference-point setting. This reduces inaccuracies.
- **Measuring log can now also be displayed on the TNC screen**
With the measuring cycles for workpiece measurement (Cycles 420 to 431), you can now have the measuring log also be displayed on the screen. The TNC interrupts program run until you exit the display by pressing the NC start button.

MARK END POINT OFF ON	MARK THIS ELEMENT OFF ON	CONVERT PROGRAM	Q INFO	SHOW OMIT BLOCK NR.	REDRAW
-----------------------------	--------------------------------	--------------------	-----------	---------------------------	--------

DCM – Dynamic Collision Monitoring

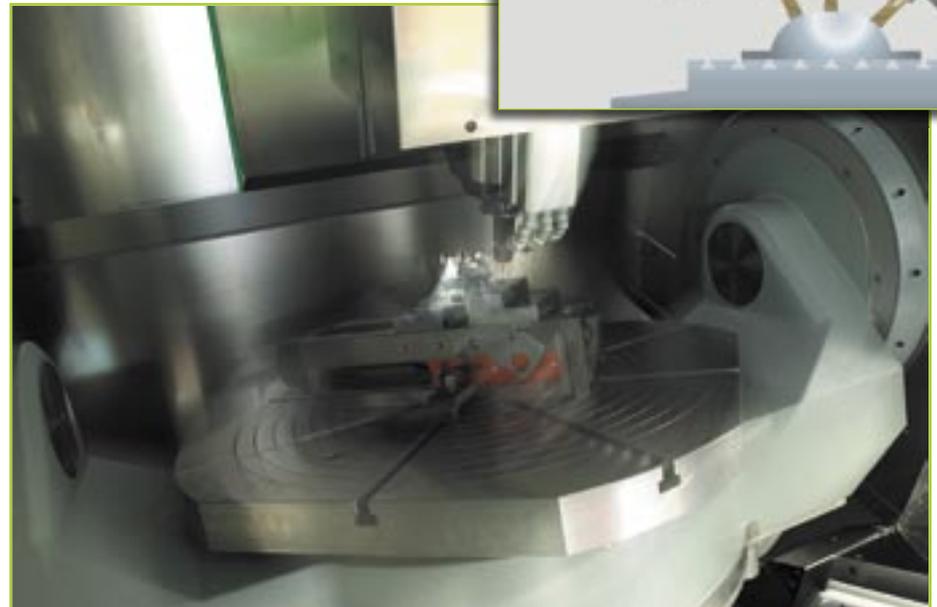
At the EMO 2005 in Hanover, HEIDENHAIN is presenting a particularly attractive new feature for machine tool builders and machine operators: DCM – Dynamic Collision Monitoring (only with MC 422 B). The iTNC 530 cyclically monitors the working space of the machine for possible collisions with machine components. Only the most recent generation of CNCs like the iTNC 530 have had the raw processing power needed to realize this function.

Dynamic Collision Monitoring (DCM)

The need for and benefit from such a monitoring feature are obvious. State-of-the-art machining centers are attaining ever higher acceleration values and travel speeds. At a rapid traverse of 60 m/min, for example, the axis covers one meter in only one second! Even with the fastest reactions, a machinist can hardly be expected to reach the emergency stop button in time to prevent a threatening collision. Also, the five-axis traverse needed for manufacturing complex molded parts is far too unpredictable. While the required NC programs are usually generated in CAM systems that avoid tool or tool holder collision with the work, it does not normally consider the machine components within the working space.

An external machine simulation also cannot guarantee that machine conditions, such as fixture positions, will be identical to those of the simulation. In the worst case, a collision will remain undetected until the damage is done.

Here is where the iTNC 530's Dynamic Collision Monitoring can help the machinist. Whenever a threat of collision occurs, the control interrupts machining and



thereby increases safety for the machine and its operator. This helps to prevent machine damage, which can result in costly downtimes. Unattended shifts become safer and more reliable.

The working space monitoring feature imparts critical advantages even beyond the iTNC's automatic mode, in which the part program is executed without interruption. It is also available in the manual operating modes: for example, if during setup the machine operator takes a collision course to a component in the working space, the iTNC 530 detects it, stops axis movement, and issues a warning or error message.

To give the machine operator a certain leeway in his work, there are three warning stages: advance warning, warning, and error – depending on the distance between the two components. Of course the iTNC 530 also shows the machine operator which machine components are endangered.

- An advance warning is released if two machine components come closer than 14 mm to each other.
- A warning is released when they are closer than 8 mm.
- An error message is released when they are closer than 2 mm.

The machine operator can acknowledge an advance warning or warning and then continue manual or controlled operation of the axes. If an error message appears, DCM must be switched off. Only then can the danger of collision be removed and the axes taken out of danger.

To enable this convenient working space monitoring feature, the machine manufacturer must save the geometry and kinematic data for all components in the control memory. The working space and the collision objects are described using geometric bodies such as planes, cubes and cylinders. Of course, complex machine components can be modeled with

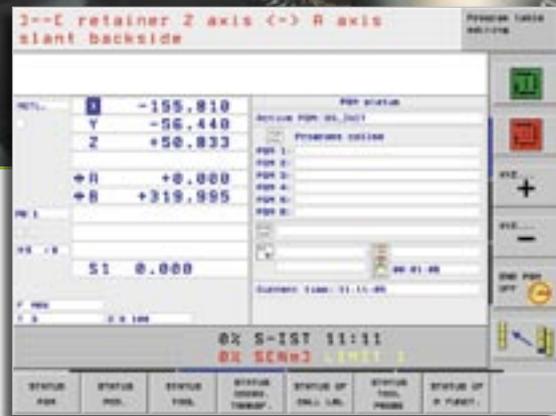
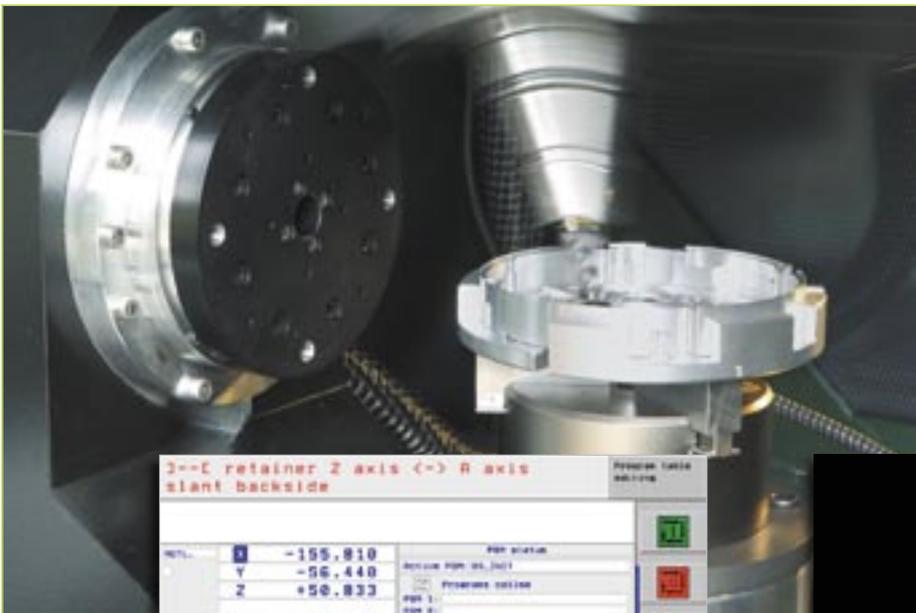
multiple geometric bodies. The tool is automatically considered a cylinder of the radius defined in the tool table.

Cyclic monitoring of all machine components within the interpolator's cycle time of 1.8 ms would require enormous computing power. Fortunately, the mechanical design of the machine itself prevents some component combinations from ever colliding. For example, a tool touch probe like the HEIDENHAIN TT 130 clamped on the machine table can never come into contact with the machine cabin. Such imaginary collisions of machine components can be ruled out from the start—and that saves computing power.

But there are also constraints to DCM:

- While DCM can help reduce the danger of collision, it cannot eliminate it.
- Only the machine manufacturer can define collision objects. The machine operator cannot define his own components, such as special jigs and fixtures.
- Collisions between machine components (such as swivel heads) and the workpiece cannot be detected.
- Handwheel superimposition (M118) is not possible.
- DCM cannot be used during operation in following error mode (without feed-forward).
- It is not yet possible to monitor for collision before machining.

There is one „collision,“ however, that DCM cannot prevent—that of the tool with the workpiece! After all, we do want to machine a workpiece.



The iTNC 530's DXF Import Function

The iTNC 530 can now also process DXF data

You can now open DXF files created in a CAD system directly on the iTNC, in order to extract contours via mouse click and save them as plain-language programs.

The DXF format is very widespread, and is supported by all usual CAD and graphics programs. This especially applies to the R12 DXF format (also known as AC1009), which is a good enough reason for HEIDENHAIN to support exactly this format.

After the DXF file has been loaded onto the iTNC from the network or a USB stick, you can open the file in the file manager, just like an NC program. The iTNC takes into consideration the operating mode from which you opened the DXF converter:

- If you call it from smarT.NC, then after you save the contour, a contour program with the file extension .HC (HEIDENHAIN CONTOUR) is generated, which you can immediately use in a contour machining unit.
- If you call it from the Programming and Editing mode, then after you save the contour, a plain-language program with the file extension .H is generated, which you can use without any further effort via the SELECT CONTOUR function.

As a rule, DXF files contain multiple layers, with which the designer structures and organizes the drawing. The designer uses the layers to create groups of various types of elements, such as the actual workpiece contour, dimensions, auxiliary and design lines, shadings,



and texts. So that as little unnecessary information as possible appears on the screen during selection of the contours, you can hide via mouse click all excessive layers contained in the DXF file. This requires the keyboard with touchpad or an external pointing device. And here's the show-stopper when you select the contours later: The iTNC can select a contour train even if the designer has saved its parts on different layers.

And it gets really convenient when it's time to define the workpiece reference point.

The datum of the drawing for a DXF file is not always located in manner that lets you use it directly as reference point for the workpiece, especially when the drawing contains multiple views. Therefore, the

iTNC has a function with which you can shift the drawing datum to a suitable location simply by clicking an element.

- At the beginning, end or center of a straight line.
- At the beginning, end or center of a circular arc.
- At the transition between quadrants or at the center of a complete circle.
- At the intersection of two straight lines/paths, even if the intersection is actually on the extension of one of the lines.
- At the intersection of a straight line and a circular arc.
- At the intersection of a straight line and a complete circle.

SET
LAYER

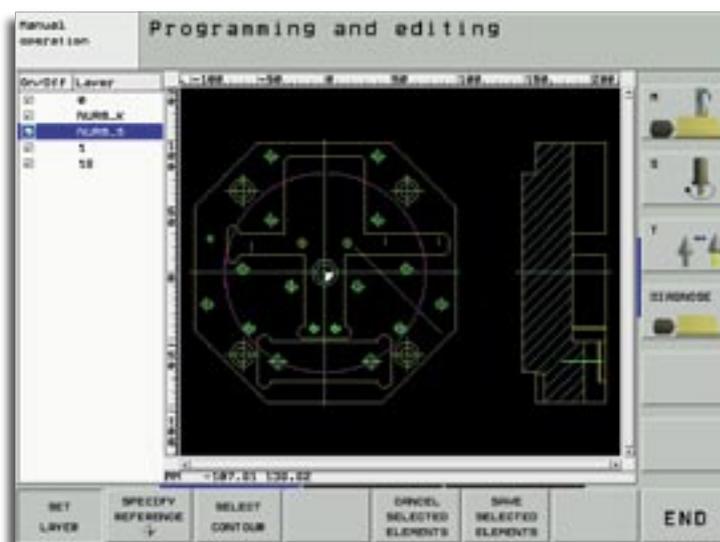
SPECIFY
REFERENCE



SELECT
CONTOUR

CANCEL
SELECTED
ELEMENTS

SAVE
SELECTED
ELEMENTS



Even if multiple intersections can result between two elements (e.g., between a straight line and a circle), the iTNC doesn't let you down: Use the mouse to decide which intersection to use.

After having set the reference point and which layers are to be shown, you begin selecting the contours. Select any element by clicking it with the mouse. As soon as you select a second element, the iTNC detects your desired direction of machining, and starts the **automatic contour detection**. The iTNC automatically

selects all clearly identifiable contour elements until the contour closes open itself or branches out. If it branches out, then you must click the next contour element. In this way you can select extensive contours with just a few clicks of the mouse.

All that's left to do is save the selected contour: Simply enter and confirm a file name, and you already have a contour program that you can use immediately. You just need to make a few technology data entries, which is very easy to do in smarT.NC.

A powerful zoom function and various possibilities for settings round out the functionality of the DXF converter. For example, you can define the resolution of the contour program to be output, in case you want to use it on older TNC controls, or a transition tolerance, if once again the elements do not quite adjoin with each other.

All in all, it is quite a shapely addition to the iTNC's many features.



New Programming Station Functions

Since the beginning of this year, the iTNC programming station has been delivered with a new keyboard. The keyboard is now included in a much smaller housing, and of course also features the new smarT.NC keys. The new features of the programming station version 340 494-02 are described below.

- **Complete availability of software options and FCL functions**

A special treat of the iTNC programming station is that all premium software options and FCL functions (only when retrofitting) on the control are available free of charge. Anyone using the freely available demo version, or who has already purchased a programming station, can extensively test all functions, and determine if an update on the machine makes economic sense. At present, the following NC functions can be purchased with a control software upgrade:

- DXF converter (software option)
- Support of USB block devices (FCL 2 function)
- 3-D line graphics (FCL 2 function)
- Touch-probe cycle for global setting of touch-probe parameters (FCL 2 function)
- Point filter for smoothing externally created NC programs (FCL 2 function)
- Manual traverse in the active tool axis system (FCL 2 function)
- smarT.NC: Coordinate transformations and PLANE function (FCL 2 function)
- smarT.NC: Define different depths for each subcontour within the contour pocket (FCL 2 function)
- smarT.NC: Block scan with graphic support (FCL 2 function)



Programming station iTNC 530: Virtual keyboard



New programming station keyboard

- **Virtual keyboard**

Especially the users of the demo version of the programming station can get an even better idea of how easy it is to operate the TNC with the keyboard. Use the iTNC Control Panel to open a virtual TNC keyboard on the screen, showing the most important dialog keys of the TNC. So that you may see the control screen as well as the virtual keyboard at the same time, you need to set your screen resolution to at least 1280 x 1024 pixels.

- **PLC basic program can be installed as option**

When installing the programming station software, you can now also install the PLC basic program as an option. This way you can perform a "dry run" in the Program Run modes with the programs created on the programming station.

Addition to the Digital Readouts – the New ND 780

At this year's EMO, HEIDENHAIN will present the new ND 780 digital readout. With this readout, HEIDENHAIN sets new standards regarding the user-friendly interface as well as the workshop-oriented design. The ND 780 was even decorated in the **iF International Forum Industry Design Competition 2005** for its successful ergonomics and its high standard of quality.

With digital readouts from HEIDENHAIN, you increase the productivity of your manually operated machine. The readouts save time, increase the dimensional accuracy of machined parts, and enhance

operating ease. The new ND 780 is especially suited for use on milling, drilling and boring machines and lathes, and can be configured for incremental encoders on up to three axes. The monochrome flat screen is an innovation for digital readouts in this range. It offers the user convenient possibilities for navigating through the various menus, and it clearly and concisely displays position values, dialog and input information, graphic functions and the graphic positioning aid. In addition, extensive online help topics are available at all times.

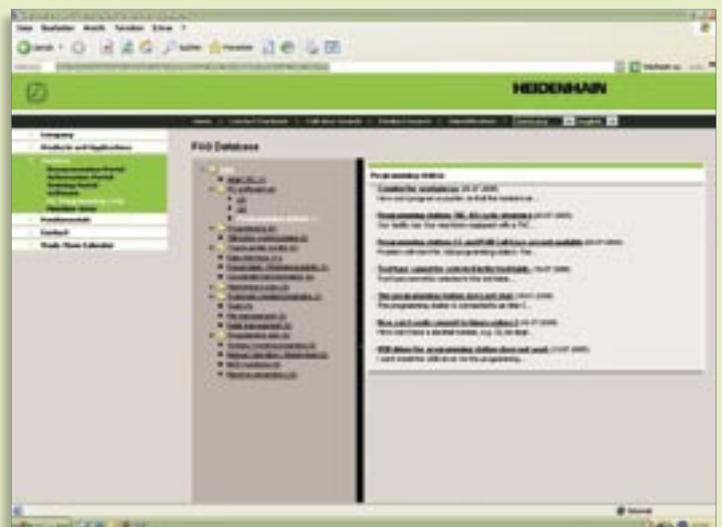


A New Look for the HEIDENHAIN Website

Come check us out again at www.heidenhain.de

Completely revised and more features than ever—this accurately describes HEIDENHAIN's new Internet presence. Due to the simple and clear structure, all information can be found with just a few clicks. Thanks to the uniform appearance worldwide, in the future you will find each bit of information at the same location in each language. The navigation is supported by a comprehensive search feature. Naturally the user documentation for TNC will still be available for download as free PDF files.

A new category will be of special interest to our TNC users: **The FAQ Database**. Here you will find the questions our hotline frequently answers, clearly summarized in one database. The FAQ is located in the "Service" chapter.



TNC 320— The New and Compact Contouring Control

Whether on simple 3-axis machines, boring machines, or as retrofit options, compact contouring controls from HEIDENHAIN such as the TNC 310 have proven themselves for over ten years.

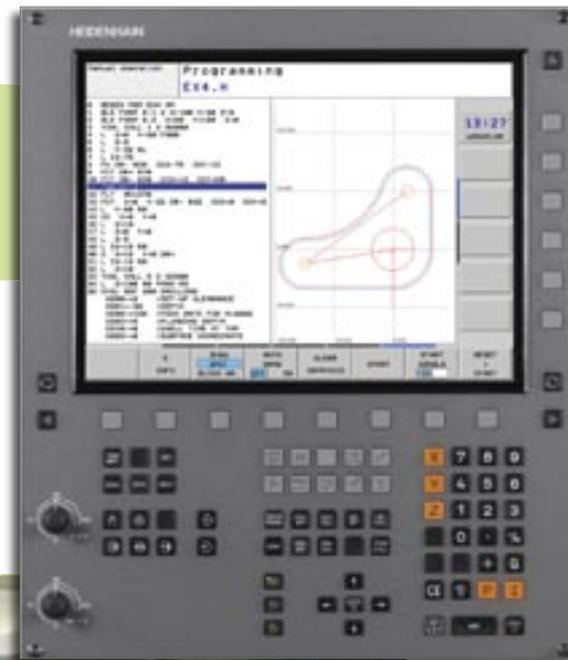
The TNC 320— compact, simple and versatile

A new hardware design and a new, innovative idea for the NC software are the most important characteristics of the TNC 320, the successor to the TNC 310 control.

If you compare the two controls, you immediately notice the large, 15-inch TFT flat panel display on the TNC 320. Having already proven itself on the iTNC 530 for years, the advantages of this screen over the monochrome flat screen of the TNC 310 become especially obvious during operation and programming.

The logical appearance as well as the clear structure of the soft keys help you to soon feel at home with the control. The keys for initiating typical TNC dialogs are included on the keyboard, so that compared to the TNC 310, you can access important TNC functions much quicker. In addition, the machine manufacturer can make machine-specific functions available in the vertical soft-key row.

When creating workshop-oriented programs using the tried-and-true conversational programming language from HEIDENHAIN, the programming graphics interactively show you step-by-step what you are currently programming. This is especially helpful when using the powerful free-contour programming feature to create parts not dimensioned for NC. The standard cycles available on the TNC 320 more than suffice for most applications.



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 320 assists you with meaningful support graphics and dialog texts while

you enter information for machining and touch-probe cycles.

After successful completion of a program, the test run graphics can give you a realistic impression of the finished part before you begin machining. The TNC also performs an internal test run to check your NC program for logical errors. This makes it easy to avoid downtimes. Tips about the cause of error, as well as possibilities for trouble-shooting, simplify the search for errors.



In order to keep set-up times as brief as possible, field-proven probing cycles are available in the Manual operating mode for you to quickly and accurately ascertain workpiece locations and orientations, as well as set reference points.

The TNC is not just exemplary in its support of the machine operator for program creation, program testing and setting up, but also during machining itself: The status display provides a clear and concise overview of various machine conditions. Of course you can tailor the display to your own needs, and specify via the keyboard which information the TNC is to show on the screen.

The new hardware does not just take advantage of its strengths during the graphic testing of programs or the editing of longer programs. The memory capacity for NC programs was increased by quite a large factor: Whereas the TNC 310 only had 128 KB of memory, the new hardware has 10 MB. This means that you have plenty of space for storing your data.

The TNC 320 is also much more capable than its predecessor 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.

Connect input and pointing devices, as well as external media (such as USB sticks or external hard drives), to the TNC 320 via the USB port (USB 1.1).

Compact and modern design

The compact design of the TNC 320 is impressive. The MC 320 main computer is located directly behind the integrated TNC keyboard and the large, clear, 15-inch TFT flat-panel display with XGA resolution (1024 x 768 pixels). All components are housed in one unit. Complicated wiring between the keyboard, screen and MC computing unit is now a thing of the past.

The TNC 320 is equipped with a powerful Intel processor with a clock frequency of 400 MHz. In addition, the 256 MB of RAM ensure that complex graphic simulations can be processed quickly.

HEIDENHAIN is following a new path regarding the memory medium for NC and PLC programs: The TNC 320 uses a CompactFlash memory card as data carrier. This memory card is immune to mechanical shock, thereby offering optimum safety for the storage of your programs.

The TNC 320 is offered with four controlled axes, and as an option, another controlled axis can be added. The nominal shaft speed values are transmitted to an external drive controller via the analog speed command interface.

Incremental and absolute encoder systems can be connected to position encoder inputs of the TNC 320. It doesn't matter whether you use 1 V_{pp} or EnDat 2.1, the TNC 320 offers you a maximum of flexibility.



HEIDENHAIN

With smarT.NC there are no detours

Only those who know where they're going get to their goal quickly and safely. HEIDENHAIN shows you the shortest path to conversational programming: the innovative smarT.NC user interface makes plain-language programming even simpler. Programming, testing and machining were never this easy: smarT.NC generates NC programs that can also be used in the plain-language interface. This means that all functions familiar to expert plain-language programmers are still available, and novices reach the goal much sooner. DR. JOHANNES HEIDENHAIN GmbH, 83292 Traunreut, Telefon: (08669) 31-0, Telefax: (08669) 5061, <http://www.heidenhain.de>, e-mail: info@heidenhain.de



start smart.

Angle Encoders + Linear Encoders + **Contouring Controls** + Digital Readouts + Length Gauges + Rotary Encoders