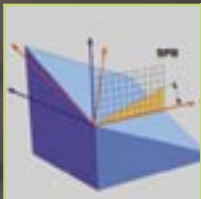


Klartext

Klartext – The TNC-Newsletter · Edition 41 · 10/2003

EMO-Edition



New iTNC 530 Functions



**New TS 640
3-D Touch Probe**



New HR 420 Handwheel



Editorial

Dear Klartext Reader!

HEIDENHAIN is strongly emphasizing the iTNC 530 contouring control at this year's EMO in Milan. A large number of new user functions will be presented. Of special note is the PLANE function, which makes it very easy to define a tilted working plane. New and powerful machining cycles, as well as a new datum management system using preset tables, improve the application possibilities and the user friendliness of the well-established TNC contouring controls.

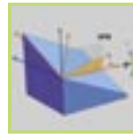
We would also like to welcome a new accessory: There is a new version of the HEIDENHAIN handwheel, the HR 420. The incremental motion of this handwheel ensures that manual, delicate positioning can now be carried out even more precisely. In addition, the handwheel features a display on which the handwheel status and several soft keys for certain control functions are shown. This makes direct alignment of the workpiece much more reliable and easy, and reduces your setup times.

Things have also changed with the 3-D touch probes. HEIDENHAIN will present the TS 640 with infrared transmission at the EMO. It features an integrated blasting unit, with which coolant or clean air is blasted onto the probing point. This clears any coarse contaminants from the workpiece before the actual probing operation, removing a source of significant error in the measurement result.

We hope you enjoy this issue!



Content



New iTNC 530 User Functions

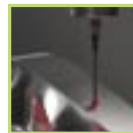
4

- Preset table 5
- PLANE function 6
- New milling cycles 8
- Programming support 9
- Other functions 10



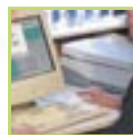
Tool-Oriented Machining

11



New Infrared TS 640 3-D Touch Probe

12



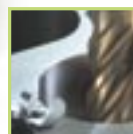
New Diagnostic Functions of the iTNC 530

14



HR 420: New Handwheel with Display

17



Programming any Shape of Pocket

18

Imprint

Publisher

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

Editor

Frank Muthmann
Fax: (0 86 69) 31-18 88
e-mail: info@heidenhain.de

Klartext on the Internet

www.heidenhain.de/english/print_0.htm

Layout

Expert Communication GmbH
Inselkammerstraße 5
82008 Unterhaching/München
Tel: (0 89) 66 63 75 0
e-mail: info@expert-communication.de
www.expert-communication.de

New iTNC 530 User Functions: Overview

During the course of 2003, numerous new, practice-oriented user functions were integrated in the iTNC 530 software. We also released functions that better support the machine operator during programming and test runs. Below is a summary of the most important improvements. More detailed information about each function is found in the individual articles.

New functions

- **Preset table** (page 5)
New powerful table for managing any number of datums, especially in combination with the PLANE function.
- **PLANE function** (page 6)
A function for easily defining tilted working planes. Animations simplify the process for selecting the correct plane definition. The function expands on Cycle 19 MACHINING PLANE, which was already included on the TNC 415.

New milling cycles (page 8)

- **Cycle 232 Plan Milling**
New powerful and flexible cycle for plan milling with allowances and in-feeds.
- **Cycles 251 to 254**
New cycles for milling rectangular and circular pockets, as well as slots and rounded slots.

Programming support (page 9)

- **Test graphics**
You can now set the speed of the test run.
- **Error messages**
All current error messages can be displayed with the ERR key.
- **Displaying messages on the screen**
Use the Q-parameter function FN16: FPRINT to display messages in any format on the iTNC display, in order to give the operator specific instructions, for example.
- **Convert FK to H**
The conversion of FK programs to conversational dialog programs was reintroduced.

Other functions (page 10)

- **Reentering a program**
The program reentry function (midprogram startup) has been available since



the TNC 415. With this function you can start at any block in a program.

- **Generate reverse contour**
Function for machining a programmed contour in the reverse direction.
- **Feed-rate programming**
Rather than entering a feed rate in a traversing block, you can also enter a time over which the programmed block is to be traversed.

New probing functions (page 13)

- **Probing a center axis**
New probing function in the Manual operating mode for capturing the center axis of two probe points.

Datum Management with the Preset Table

The new datum management for the iTNC via the preset table PRESET.PR greatly simplifies setup of your machine and reduces the time necessary. Now you can not just save any number of



Figure 1: Preset-Table PRESET.PR

datums in the preset table, but you can also save individual basic rotations for each datum. This possibility is especially helpful if you have clamped several workpieces in a fixture or in multiple vices.

Datum setting with an active tilted working plane has also become much more transparent. You can now save datums in any angular setting in the preset table, and then approach these datums from any other angular setting. Machines with an automatic head-changing system also profit from the preset table: The datum is maintained even after the head has been exchanged, and so does not need to be set again.

If you have a machine capable of multiple traverse ranges (such as for alternating table machining), the iTNC automatically creates a separate preset table for each traverse range. As soon as you switch between traverse ranges, the iTNC activates the most recently active datum from the appropriate preset table.

There are three different possibilities for saving datums very easily in the preset table:

- **Manual operating mode:**
Via soft key you can save the currently active datum to any line in the preset table. (Figure 1)

The manual probing functions were expanded to permit saving basic rotations and datums together in the preset table.

- **Automatic operation:**
The automatic probe cycles were expanded to permit saving measuring results for basic rotations (Cycles 400 to 403) and for datums (Cycles 410 to 419) in the preset table as well.

Switching between datums is also very easy:

- In the manual operating mode, via soft key directly from the preset table
- In automatic operation via Cycle 247 by entering the preset number

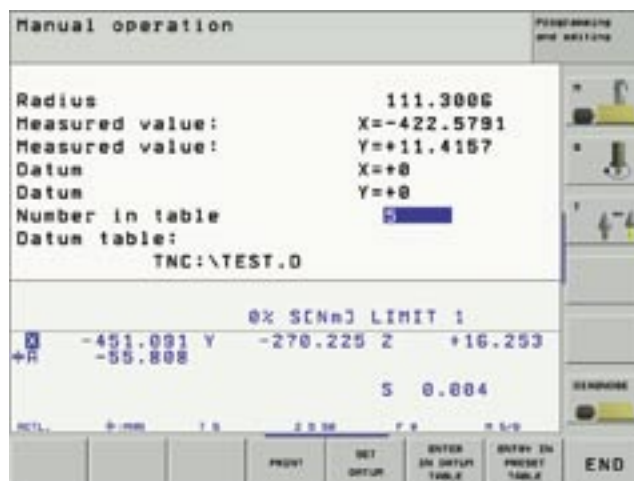


Figure 2: Expanded probing menu



Figure 3: Probe cycle with value transfer to the preset table



The PLANE Function

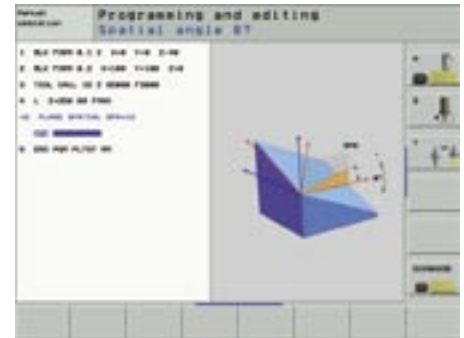


The introduction of the PLANE function in the spring of 2003 greatly expanded the possibilities for defining a tilted working plane, and at the same time simplified them. Using Cycle 19, only axis and space angles were permitted as plane definitions. The PLANE function gives you seven different methods for defining tilted working planes, depending on the information given in the workpiece drawings. In order to keep the procedures of these complex functions as simple as possible, the iTNC's support graphics have "learned how to walk." A separate animation is available for each possible plane definition. You can even view them before selecting the function. During the definition, comprehensive support graphics – long familiar from the cycle definitions – show you the entries that the iTNC expects.

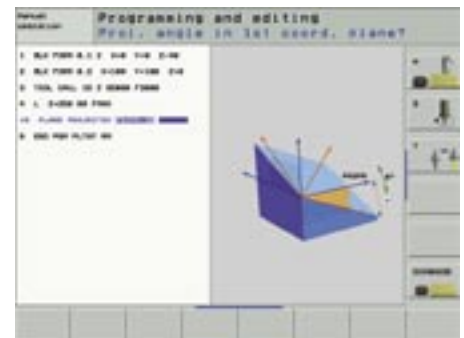
A special advantage of the PLANE function is that you can define the positioning behavior, so that there are no unpleasant surprises when the program is run. The settings available for defining the positioning behavior are identical for all PLANE functions, making everything that much easier.

Possible plane definitions

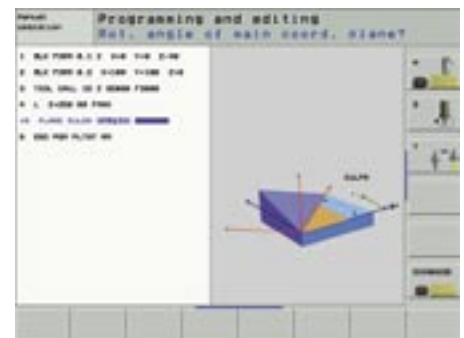
- **PLANE SPATIAL**
As with Cycle 19, the PLANE SPATIAL function is used to define up to three space angles: SPA, SPB and SPC.
- **PLANE PROJECTED**
In the PLANE PROJECTED function you define the working plane via two projection angles that are produced by the intersection of the tilted plane and the main coordinate planes (Z/X and Z/Y).
- **PLANE EULER**
(Euler was a Swiss mathematician)
The PLANE EULER function enables you to define the tilted working plane via three successive rotations about the momentarily active coordinate system.
- **PLANE VECTOR**
The PLANE VECTOR function enables you to define planes via two vectors. One vector defines the normal direction of the plane (vertical axis in the tilted plane), and the other vector defines the direction of the tilted X axis. PLANE VECTOR is suitable for when it is easy to determine the normal vectors using a CAD system.
- **PLANE POINTS**
In the PLANE POINTS function you simply define any three points in the tilted working plane, and the iTNC calculates the position of the rotary axes from which the plane can be reached. You can use the results from Cycle 431 (MEASURE PLANE) directly as input parameters for PLANE POINTS.
- **PLANE RELATIVE**
The PLANE RELATIVE function enables an incremental continuation of an already defined tilted working plane. This makes it very easy to machine chamfers in a tilted plane.



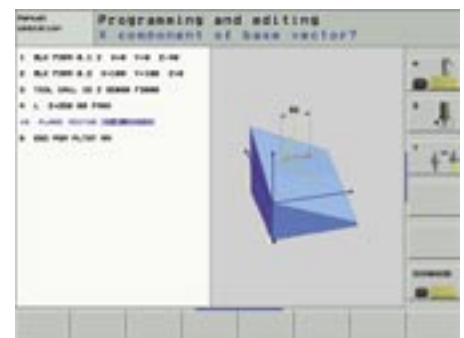
PLANE SPATIAL



PLANE PROJECTED



PLANE EULER



PLANE VECTOR

• **PLANE RESET**

The PLANE RESET function resets an active plane-tilting function, regardless of how you defined the tilted plane. The double resetting required by Cycle 19 is not necessary here. You can also define the positioning behavior for the resetting motions.

Specifying the positioning behavior

• **MOVE/TURN/STAY positioning (input mandatory)**

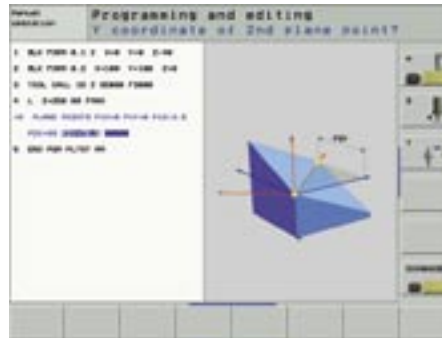
The MOVE function specifies that the iTNC is to automatically position the rotary axes while executing a PLANE function. The position of the tool relative to the workpiece is maintained, meaning that the iTNC carries out a compensating motion during positioning. You freely define the positioning feed rate and the distance from the center of rotation to the tool tip directly in the PLANE function.

The TURN function, developed especially for machines with rotary and/or tilting tables, also positions the rotary axes automatically, but does not change the position of the tool. This means that the iTNC does not carry out a compensating motion during positioning.

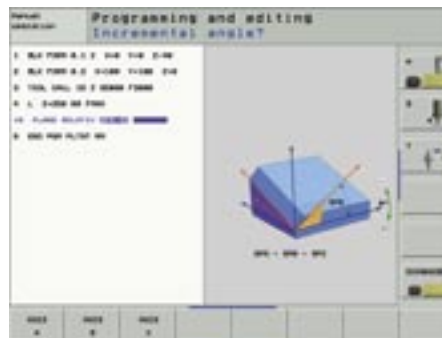
If you do not want any automatic positioning to occur, use the STAY function. In this case the iTNC only activates the tilted working plane mathematically, so you must use a separate positioning block to position the rotary axes to the angles calculated by the iTNC. The iTNC stores the calculated angle settings in parameters Q120 to Q122 when performing a PLANE function, just as with Cycle 19.

• **Selecting a solution SEQ (input optional)**

All plane definitions can be realized with two different settings of the machine axes. The SEQ function lets you specify yourself which of the two possibilities you would like to use.



PLANE POINTS



PLANE RELATIVE

• **Rotate table or coordinate system: TABLE ROT/COORD ROT (input optional)**

If a new machining plane, achievable by rotating just one rotary axis, becomes necessary, you can specify whether the iTNC should actually rotate the axis, or if it should only rotate the coordinate system internally. This function is useful when a very large workpiece is on the rotary table for machining, and the danger of collision makes it impossible to rotate the table.

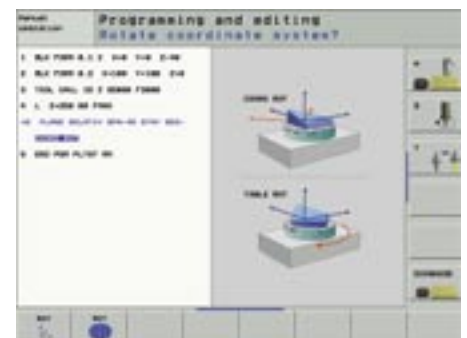


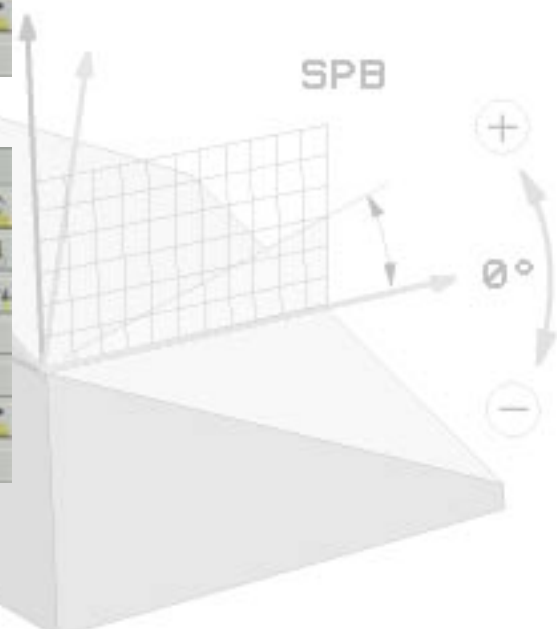
TABLE ROT/COORD ROT



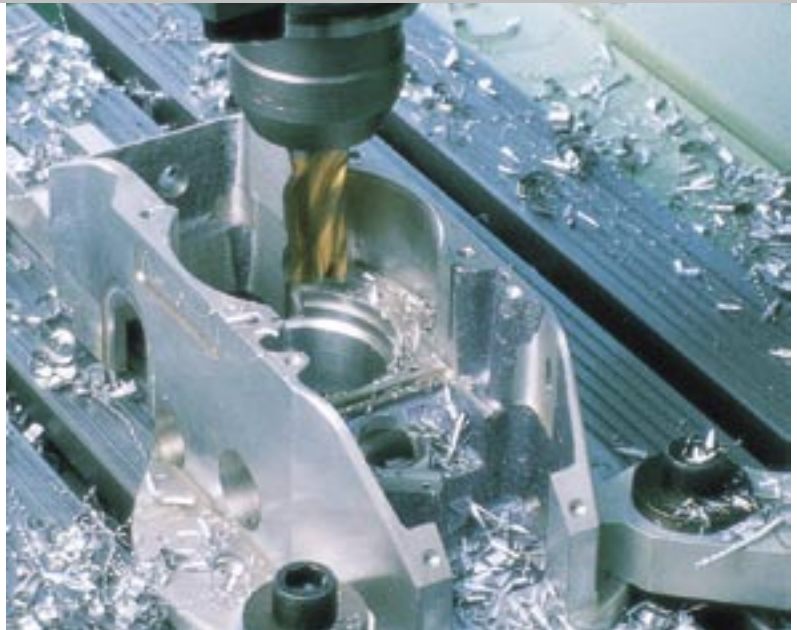
MOVE/TURN/STAY



Selecting a solution SEQ



New Machining Cycles



Cycles 251 to 254

Together with the PLANE function, the four milling cycles Rectangular Pocket (Cycle 251), Circular Pocket (Cycle 252), Slot (Cycle 253) and Circular Slot (Cycle 254) were introduced in the spring of 2003. An important goal was for the programming of these cycles to be as similar to each other as possible. After having used the cycles in machine shops for several months, machine tool operators gave us ideas for the rough-out strategies, which we have now incorporated into the cycles. Of special importance is that the cycles have been programmed to reduce non-productive motions as much as possible. Many unnecessary retraction and repositioning motions between the roughing and finishing operations were removed, which of course has a positive effect on the machining time.

Due to the rough-out strategy for slot cycles 253 and 254, until now the slot width was not allowed to be greater than three times the tool diameter. Now you can use tools with diameters as small as you like to create slots as large as you like. The cycles continually expand the slot from the inside out.

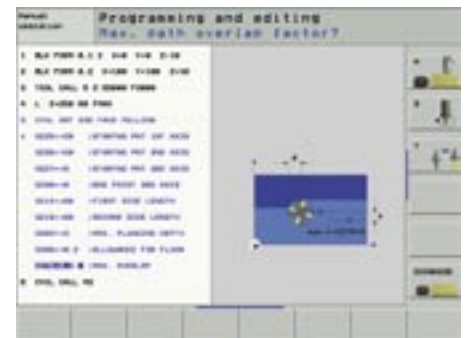
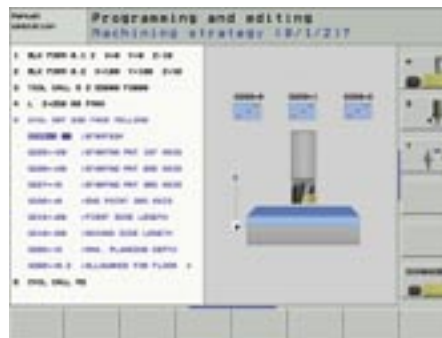
Cycle 232 Face Milling

Cycle 232 is used to face mill a level surface in multiple infeeds while taking the finishing allowance into account. Three machining strategies are available:

- **Strategy Q389 = 0**
Meander machining, stepover outside the surface being machined
- **Strategy Q389 = 1**
Meander machining, stepover within the surface being machined
- **Strategy Q389 = 2**
Line-by-line machining, retraction and stepover at the positioning feed rate

In order to machine the surface as evenly as possible, the TNC calculates both the plunging depth and the stepover so that each machining pass cuts in a uniform manner. The TNC even takes face-milling cutters whose tooth radius is entered in the tool table into account when calculating the stepover.

In order to optimize the machining time, returning to the entered starting point is omitted. At the end of the last milling pass, the TNC immediately plunges to the next depth and machines in reverse to the entered starting point.



Use the algebraic sign of the two side lengths to specify – beginning from the entered starting point in the working plane – in which direction the first milling pass takes place. This increases your flexibility, since the starting point is not at the bottom left of every workpiece.

New Functions for Programming Support

Displaying all current error messages

You can use the ERR key (only on the TE 530 keyboard unit) to open a pop-up window in which the iTNC lists all current error messages. The displayed errors can come from the NC or from the machine tool builder. The HELP key is especially useful, since parallel to the list of error messages you can display the error description for each message.



Error message

Displaying messages on the screen

Until now you have been able to use the Q-parameter function FN16: FPRINT to create formatted text files from within the NC program, such as for logging measurement results. Now, if you define the keyword SCREEN: instead of the file name, the TNC outputs the Q-parameter contents or the texts stored in a format file directly on the iTNC's screen, for giving the operator specific instructions, for example. The message remains on the screen until the operator acknowledges it. However, the program will continue to be machined by the iTNC, unless a STOP block is programmed directly after the FN16 block.

Conversion of FK (Free Contour programming) to conversational dialog

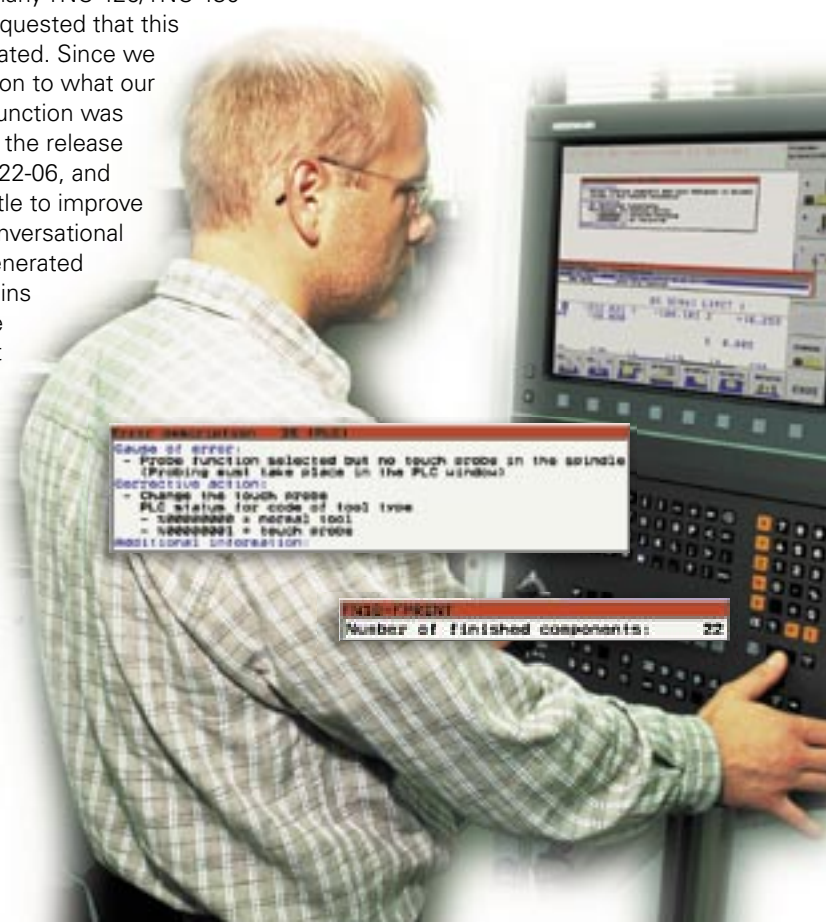


Displaying messages on the screen

When the iTNC 530 was introduced, the "Convert FK program to conversational dialog" function familiar from older controls was removed from the software. As it turns out, however, this function is intensely used by many TNC 426/TNC 430 operators, who requested that this function be reinstated. Since we always pay attention to what our users want, this function was reintroduced with the release of software 340 422-06, and was modified a little to improve the clarity: The conversational dialog program generated by the iTNC contains as a comment the block number that the FK block had in the FK program, making it easier to align the sequences of the source and target files.

Setting the speed of the test-run graphics

The increased processing power of the iTNC 530 naturally led to the graphic simulation running at a higher speed. However, for TNC beginners this – supposedly pleasing situation – actually led to uncertainties, since due to the high speed of the simulation the tool path was no longer clearly visible. Now you can use soft keys to set the speed of the graphic simulation to any rate that you want. In effect, the soft keys work like the feed-rate potentiometer when machining a program.



Other NC Functions

Reentering the program

The program reentry function (mid-program startup) has been available since the TNC 415. With this function you can start at any block in a program. A new feature is that with the iTNC, the point of interruption is remembered if an emergency stop or sudden power interruption occurs. When you restart the machine, a message stating that the NC program was interrupted is displayed immediately. If you activate the reentry function in the Program Run, Full Sequence mode, the iTNC displays the point of interruption, which you can select via soft key as the reentry point.



Reentering the program

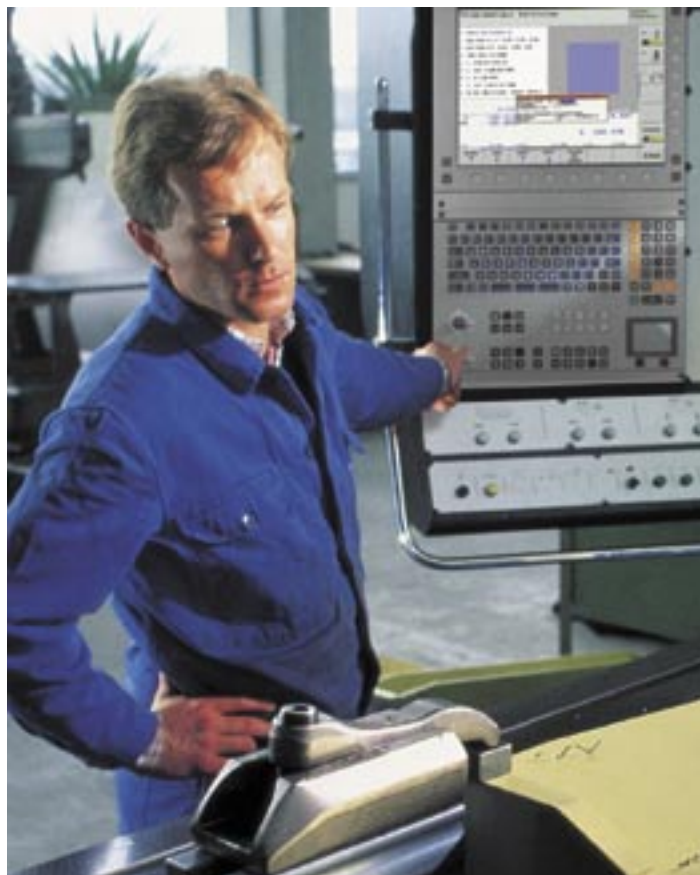
Generate reverse contour

This new iTNC function is used to reverse the machining direction of an existing contour, which preferably was created with a CAD/CAM system. This function, well-known from electrical discharge machines, lets you machine a contour forwards and backwards in multiple in-feeds, in order to avoid any unnecessary non-productive times caused by retracting motions.

The contour itself may contain all common iTNC path-function elements, including FK blocks. The iTNC shifts RND and CHF blocks so that they are at the correct position in the reverse program.

Feed-rate programming

Rather than entering a feed rate in a traversing block, you can also enter a time in seconds over which the programmed block is to be traversed. The function is activated with the FT soft key in the feed rate dialog box, and is effective block-wise.



Tool-Oriented Machining

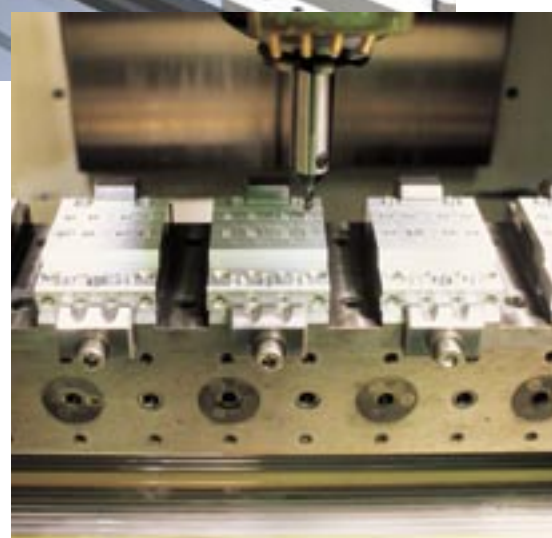
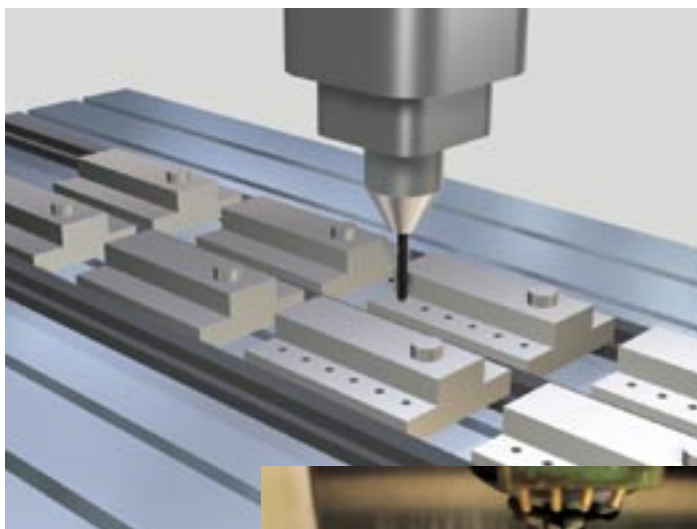
With tool-oriented machining you can machine multiple identical parts with just one machining program, using the tool-oriented method. The advantage comes from machining each workpiece entered in the program before proceeding with the next machining step. This means that all workpieces are machined with the required tool, before the next tool change. This decreases the number of tool changes to a necessary minimum, greatly reducing the machining time.

Tool-oriented machining was originally intended for pallet management in order to machine multiple workpieces on one pallet in as brief a time as possible. However, it is obvious that this machining method can also be used for other applications. For example, this means that you can save a great deal of time when machining identical workpieces on multiple fixtures or vices on the machine table, without needing a special structure in the NC program.

The advantages of tool-oriented machining are not limited to just the savings in time and therefore production costs. Another significant advantage is the support from clear and simple entry forms. Here you enter in a pallet file at which location each machining step is to take place. Use the soft keys at any time within the pallet file to toggle between a simplified over-

view (with all the workpieces listed) and the detail view for each workpiece. This way you can easily and efficiently program your workpieces using the tool-oriented method.

The machine tool builder must prepare the iTNC for tool-oriented machining. Once this has been done, you simply need a conventional machining program and a pallet file.



Entry form: simplified view for a workpiece

The reality of the workshop demands great flexibility when working with NC machines. Often jobs must be interrupted to carry out more urgent orders. Tool-oriented machining handles this very easily. While executing the pallet file, the iTNC stores a code (valid for two weeks) used to reenter the program at the point of interruption.

A field-tested function is the assignment of blank spaces in the pallet file. This way you can take into account tool breakage or the machining of the remaining workpieces of a job, easily and without much effort. Of course other special features, such as automatic program start at a specified time, are available in connection with the pallet table.



Entry form: detailed view for a workpiece



Automatic program start

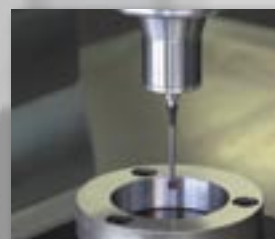
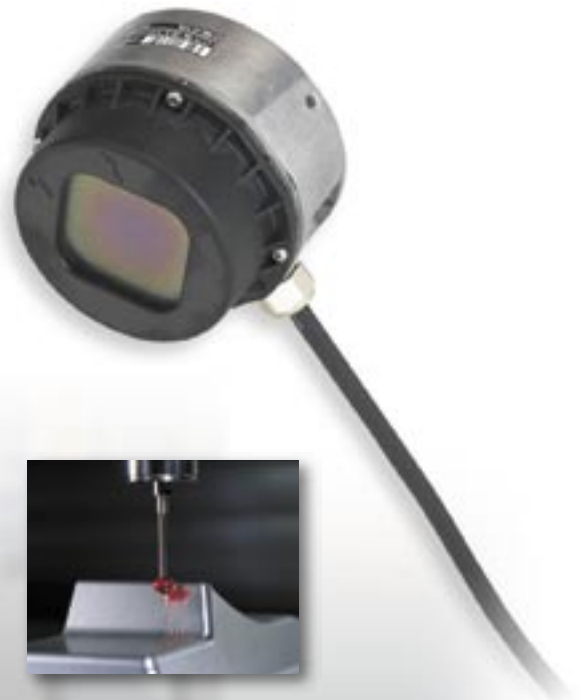
New Infrared TS 640 3-D Touch Probe – For Aligning and Measuring Workpieces

The 3-D touch trigger probes from HEIDENHAIN are used to carry out repeated setup procedures easily, exactly and quickly. We are proud to present a new addition to this product group – the **TS 640**. In combination with the introduction of the new version of the iTNC 530, the **TS 632** is being discontinued. The old **EA 632** receiver unit will be replaced by the new **SE 640** transceiver unit. The most interesting new feature on the **TS 640** is the integrated blasting unit consisting of three nozzles at the bottom of the touch probe. The nozzles use a blast of air or coolant to remove coarse contaminants from the area being probed. This saves time and enables unattended, automated measuring cycles.

We were also able to improve the transmission of the infrared signal. Now the infrared LEDs

and receiver modules for signal transmission are evenly arranged around the circumference of the **TS 640**. This ensures transmission in all directions of the working plane as well as a reliable signal reception without requiring an oriented spindle stop. In addition, the signal can also be transmitted by reflection, which lets you take advantage of the transceiver's generous mounting tolerances, and most importantly, permits the use of swivel heads.

The **TS 640**'s area of transmission was also greatly enlarged. It was increased from 3 m to 7 m, making the touch probe suitable for use on larger machines without requiring additional receiver units within the machine envelope. The **TS 640** is switched on and off by the new **SE 640**, meaning that **no mechanical switch is necessary**. The IP 67 protection class (dust-proof and temporary submersion in liquids) also increases the system reliability.



New Touch Probe Functions

The SE 640 transceiver unit has an impact-resistant synthetic window (immune to high-pressure emulsion), which also significantly increases the system reliability. The status display immediately shows the user whether the transmission path is in order and whether the system is ready. The improved data transmission makes the system less susceptible to light contamination (such as machine-floor lighting) or magnetic influences (improved EMC protection).

Probing a center axis

With this new touch probe function in the Manual operating mode, you can have the iTNC determine the center point between any two probing points (of a slot or ridge, for example) in the active working plane, and then

- directly set it as a datum,
- save it in a datum table, or
- save it in the preset table.

In addition, the iTNC displays the measured distance.



Manual operation		Preparation and setting
Measured width:	52.7389	
Measured value:	-272.5287	
Datum	0.0	
Number in table	5	
Datum table:	TNC\TEST.D	
0% SCHEJ LIMIT 1		
X	-101.091	Y -200.777 Z +210.524
+R	+0.000+0	-0.001
	S	0.524
PRESET SET ENTER TO ENTER IN END OFF ON TABLE PRESET TABLE		



New Diagnostic Functions of the iTNC 530

For several years now, HEIDENHAIN has produced controls with complete drive packages. These consist of inverters and the appropriate motors. The inverters provide the electrical energy, which (regulated by the controls' nominal values) is passed on to the motors. Each inverter consists of a power supply unit and a power module. The motors (usually synchronous motors for axis drives and asynchronous motors for spindle drives) transform the electrical energy into mechanical energy, i.e. for moving the axes and spindle. It goes without saying that the motors are equipped with rotary encoders from HEIDENHAIN for capturing the rotational speeds.

In combination with the newest HEIDENHAIN software, the complete packages from HEIDENHAIN make it possible to run multifaceted diagnostic functions "when the time comes" in order to localize the problem as quickly as possible. After the error has been found, all the service technician needs to do is correct the error. A significant advantage is gained by using the TNCdiag PC software from HEIDENHAIN, which displays the possible error causes as well as giving suggestions for further procedures.

The diagnostic functions permit a simple, fast and convenient search for the errors:

- Support while commissioning a control
- Automatic identification of motors and power modules
- Automatic function test of motors and inverters
- Diagnosis of hardware errors (in the wiring, for example); display of possible errors, their causes and further procedures for correcting the problems

- Display and evaluation of internal control statuses (including the controllers)
- Analysis of the position and speed controller signals
- Display of the momentary values of the dc-link voltage and dc-link current, supply voltages, and the temperatures of inverters, motors, and controller PCBs
- Use of the diagnostic functions via remote diagnosis as well

Electronic ID labels

A significant step towards improving the ease of diagnosing HEIDENHAIN components was equipping the inverter components and synchronous motors with absolute rotary encoders with an "electronic ID label." This electronic ID label is a memory module in the devices, containing the model name, the ID number and the serial number of the device. The first advantage comes into play when commissioning a HEIDENHAIN system: The power modules and motors are automatically detected by the control; this simplifies the parameterization and avoids input errors. Other advantages become apparent when a device fails. Axis motors, for example, are often mounted in shafts, making it anywhere between difficult and impossible to read the ID label. The TNCdiag software automatically reads the required information. You can also use the TeleService software for customers who are hundreds of miles away.

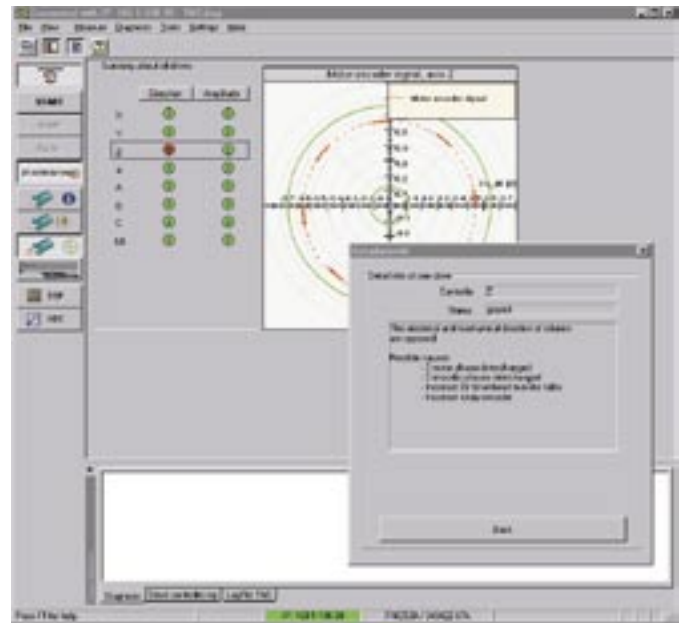
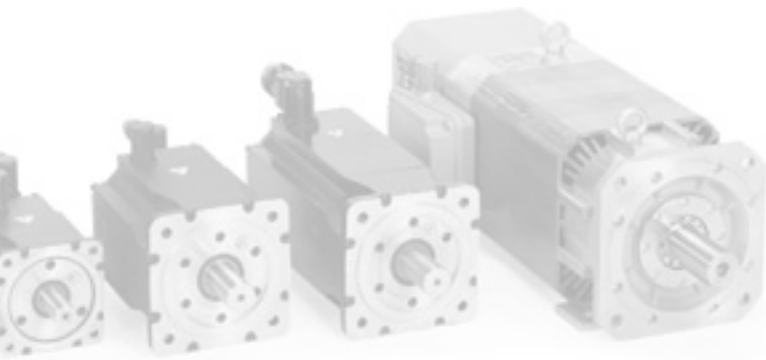


During startup, the iTNC automatically detects the connected UM 121D power module of the Y axis via the electronic ID label

This ensures that the service technician will bring the correct motor with him. After the new motor (or power module) has been mounted, the control compares the new device with the previous device, and displays a message if the two do not match. Then the service technician can decide if this is OK or not. In effect, the control regulates the exchange of the components. Of course controls from HEIDENHAIN are extremely reliable, which is important for avoiding machine downtimes. But sometimes a machine tool can develop a problem that cannot be solved without support.

Automatic function test of the drive components

The highlight of the new diagnostic functions is the automatic function test of the complete drive, meaning that the motor and inverter are tested. First the correct communication between the control and inverter is checked, to ensure that the components are attached to each other correctly. Then a test algorithm which automatically checks the drive components is begun. No action by a technician is necessary. Now even errors can be



TNCdiag automatically detects faulty parameterization of the direction of rotation for the motor in the Z axis

found that previously only a technician with additional inspection devices could find. The following errors are detected automatically:

- Faulty connection between controller and inverter (e.g. connecting cable attached incorrectly or not at all)
- Significant wiring error (e.g. defective main contactor)
- Error in the power supply unit (e.g. due to incorrect dc-link voltage)
- Ground fault or short circuit of a power transistor (e.g. defective power module)
- Ground fault of individual motor phases (e.g. defective socket connection results in a ground fault of a motor phase)
- Interruptions of individual motor phases (e.g. break in the power supply cable)
- Short circuit of two motor phases (e.g. defective motor coils)

Automatic function test of the encoders

The new HEIDENHAIN diagnostic functions are not limited to just the drive components, but naturally also support the encoders – for capturing both the position and the rotational speed.

An automated test procedure checks the peak-to-peak values of the encoder signals, and displays the result via status LEDs. The signal is simultaneously displayed on an XY graph for the service technician.

Integrated in the test procedure for speed encoders is a test of the direction of rotation. The motor's direction of rotation stored in the control is compared with the actual direction of rotation, and the result is shown by the status LEDs. It is a quick and simple test, detecting misleading subsequent errors that can occur during parameterizing while commissioning the control.

Diagnostic functions for supporting the service technician

With a subject matter as complex as control and drive technology of machine tools, especially with the many ambient conditions to be considered, it is often not possible to find an error automatically. When this is the case, the service technician's experience and ability to "put two and two together" are required in order to diagnose the correct cause of error from the measurements. But even here the HEIDENHAIN diagnostic func-

tions are very helpful. Since the hardware and software of the controller work together, all control-loop relevant information of the hardware and the software is displayed together in the window. Of course the status LEDs are used to clearly display this information. All analog signals of the drive system, which are also available to the controller software, are shown in another window – from the momentary dc-link voltage, dc-link current, and all motor temperatures to the power supply and auxiliary voltages of the controller hardware. As a supplement to these windows, TNCdiag offers suggestions to the possible causes of error as well as how to fix them. Here you can also generate an overview of the complete drive system including the parameterization and the data from the electronic ID label. There is no need to search through various system files for the desired information – it is all here at a glance.

Remote diagnosis – searching for errors online

The diagnostic functions described up until now are carried out either directly on the control via soft keys, or from an external PC connected to the control



TNCopt detects, for example, interrupted motor phases in the X axis



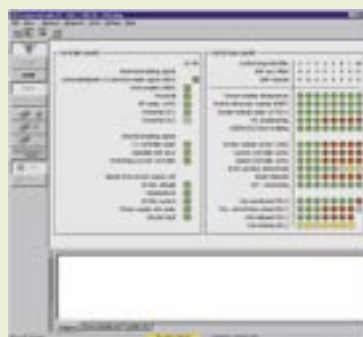


via the Ethernet interface, for example. However, a requirement for this is always an appropriately-trained service technician at the machine. The technician's travel time to the machine is already an expense, even though the cause of error might not be completely clear. This is where the advantages of remote diagnosis become obvious. The technician, still sitting in his office, establishes a connection with the machine via the telephone lines. This results in the cost of a simple phone call. Using the TeleService software from HEIDENHAIN, the technician can then access all relevant information

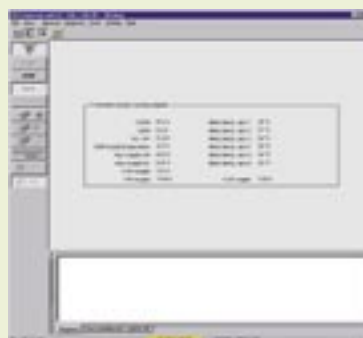
about the control. Once the cause of the error has been detected, it can be solved by the end user (in simple cases, such as a loose connector), or in more complicated cases the service technician can at least narrow down the possible error causes. TNCdiag is an integral part of the newest version of TeleService, meaning that all the possibilities described above are also available online. These diverse tests can now be carried out remotely, and electronic ID labels can also be read. These are advantages that use today's information technology to help shorten repair times and therefore reduce costs.



Remote diagnosis of the iTNC 530 with the TeleService PC software



Viewing control-loop relevant information with the status LEDs on the iTNC 530 and with TNCdiag



Viewing the analog signals of the drive system on the iTNC 530 and with TNCdiag

Conclusion

HEIDENHAIN uses proprietary test algorithms together with state-of-the-art software components and drive technology adapted to this combination for these new diagnostic functions. The experiences gained, as well as the application thereof with the greatest possible flexibility regarding these functions – directly at the control, externally on a PC, or even via remote diagnosis – raise HEIDENHAIN above other control manufacturers when it comes down to ease of diagnosis. So if, “when the time comes,” a HEIDENHAIN component should ever fail, comprehensive and automated tools are available for ensuring a quick and reliable search for errors. This helps to reduce machine downtimes, from which machine manufacturers and end users all profit.



Since 1981, electronic handwheels have been as much a part of TNC controls as conversational programming has. They are an indispensable aid for expert machinists, especially when setting up workpieces.

They are used to move the axis slides via the feed motors according to the rotation of the handwheel – just as delicately as on a manually-operated machine tool.

HEIDENHAIN will present the new **HR 420** incremental handwheel with status display at the EMO in Milan.

The display provides the machine operator with much useful information. It directly shows the actual position of each axis. The display also presents the traverse path of the axes for each handwheel increment, the programmed feed rate, the programmed spindle speed, the selected operating mode and any error messages.

Use the keys to select the axis to be moved, as well as the traverse path per

handwheel increment. You can also move the axes continuously. And if you want to capture the current position for the program, all you have to do is press one key. Of course you can also start or stop the spindle or the NC program with the handwheel*. The handwheel also features dials for the spindle and feed-rate override. The machine tool builder can implement additional functions via the soft keys and integrated PLC*.

The handwheel is a complete accessory, and belongs on every machine tool using a HEIDENHAIN contouring control.

* Beginning of 2004

iTNC 530 Programming Station

As many TNC operators likely know, software has been available for an iTNC 530 programming station attached to a Windows PC since the beginning of 2003. The programming station software closely resembles that of an iTNC attached to a machine tool. The keyboard remains unchanged except that it also includes the soft keys, which are otherwise integrated in the iTNC's visual display unit. You connect the iTNC keyboard to your PC's USB port. The PC screen displays the familiar iTNC user interface.

The cycle structure can be switched via MOD function to the following controls for downwardly compatible programming:

- iTNC 530 with preset table (software no. 340 422-xx)
- iTNC 530 without preset table (software no. 340 420-xx)
- TNC 426/TNC 430 (software no. 280 476-xx)
- TNC 410 (software no. 286 060-xx)

This makes it very easy to also create programs for the controls listed above using the iTNC 530 programming station. You simply have to ensure that you do not program any functions specific to the iTNC 530 (such as the PLANE function).



You can download the current programming station software from the HEIDENHAIN homepage in the Service Area under "Download Software" (approx. 45 MB), or simply request a demo CD-ROM in the "Documentation" area!

Programming any Shape of Pocket



The SL Cycles of the iTNC enable you to form complex contours by combining up to 12 subcontours (pockets or islands). Until now the contour descriptions had to be defined as contour labels directly in each parts program. The radius compensation and programmed machining direction of the tool were used to determine if the contour was a pocket or an island. The starting positions of the subcontours were used to determine the final shape of the contour. These functions were already available on the TNC 415.

The new **Contour Formula** function significantly increases the flexibility of these powerful machining cycles.

- You define the subcontours in a separate program, but:
 - Without radius compensation,
 - Without needing to take the machining direction of the tool into account,
 - Without needing to take the starting point into account, and
 - Without entry of technology data (feed rates, M functions).

This means that once a subcontour has been defined, it can be used in any program, whether as part of a pocket or an island. Of course you can also create the subcontour with the very powerful **Free Contour Programming FK** function of the iTNC.

- Use the new **Contour Formula** function to specify how the iTNC should combine the subcontours with each other:

- Form the intersection of the subcontours
- Form the area of inclusion of the subcontours
- Form the area of inclusion but without the intersection of the subcontours
- Mill subcontour 1 without the intersection of subcontour 2
- Mill subcontour 1 (e.g. a pocket) without subcontour 2 (e.g. an island)



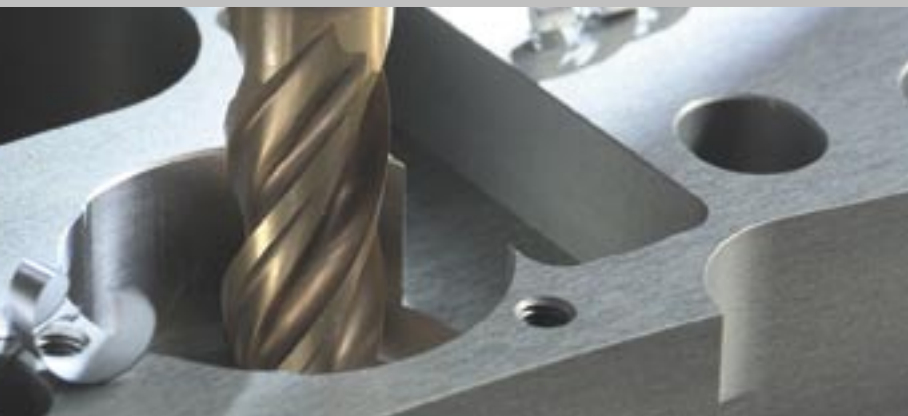
- The programming graphics of the iTNC display the programmed contours in differing colors (which can be selected via machine parameters):
 - Each **subcontour** in blue
 - The **total contour** in green
 - The **traverse paths** of the tool in black

You define the machining parameters in the Cycles 20 CONTOUR DATA and 22 ROUGH-OUT. The machining cycle carries out the approach and departure movements itself, and naturally finishing allowances for side and depth can be taken into account. Cycle 22 ROUGH-OUT can also be used to define the fine roughing of subsections with a smaller tool: simply define Cycle 22 again, and define the number of the coarse-roughing tool in the appropriate cycle parameter. When the cycle is called, the TNC then only machines the areas that the larger tool could not machine.

The following programming example is intended to show how easily the contour formula can be used, and how simple it is to define the fine roughing.



Traverse paths of the tool



Subcontour 1: Ridge, used as an island

```
0 BEGIN PGM ISLAND MM
1 L X+55 Y+40
2 L X+25
3 CR X+25 Y+60 R+10 DR-
4 L X+55
5 CR X+55 Y+40 R+10 DR-
6 END PGM ISLAND MM
```

Subcontour 2: Rectangular contour with corner radii, used as a pocket

```
0 BEGIN PGM RECPOC1 MM
1 L X+25 Y+20
2 L X+60
3 LY+75
4 L X+2
5 RND R2
6 LY+20
7 RND R2
8 L X+25
9 END PGM RECPOC1MM
```

Subcontour 3: Circular contour, used as a pocket

```
0 BEGIN PGM CIRCPOC1 MM
1 CC X+60 Y+55
2 LP PR+35 PA+0 R0
3 CP PA+360 DR-
4 END PGM CIRCPOC1 MM
```

Contour-calculation program

```
0 BEGIN PGM CONTCALC MM
1 DECLARE CONTOUR QC1 = „RECPOC1“
2 DECLARE CONTOUR QC2 = „CIRCPOC1“
3 DECLARE CONTOUR QC3 = „ISLAND“
4 QC10 = ( QC1 | QC2 ) \ QC3
5 END PGM CONTCALC MM
```

Block/Meaning

Blocks 1 to 3: Definition of the subcontour
 Block 4: Contour formula: Resulting contour 10 (QC10) should be calculated as intersection of contours QC1 and QC3 without contour QC3

Machining program

```
0 BEGIN PGM MILL MM
1 BLK FORM 0.1 Z X+0 Y+0 Z-40
2 BLK FORM 0.2 X+10 Y+100 Z+0
3 TOOL CALL 5 Z S3000
4 L Z+100 R0 FMAX M3
5 SEL CONTOUR „ CONTCALC „
6 CYCL DEF 20 CONTOUR DATA
    Q1=-20 ;MILLING DEPTH
    Q2=+1 ;TOOL PATH OVERLAP
    Q3=+0 ;ALLOWANCE FOR SIDE
    Q4=+0 ;ALLOWANCE FOR FLOOR
    Q5=+0 ;WORKPIECE SURFACE COORD.
    Q6=+2 ;SETUP CLEARANCE
    Q7=+50 ;CLEARANCE HEIGHT
    Q8=+0 ;ROUNDING RADIUS
    Q9=+1 ;DIRECTION OF ROTATION
7 CYCL DEF 22 ROUGH-OUT
    Q10=-5 ;PLUNGING DEPTH
    Q11=+150 ;FEED RATE FOR PLUNGING
    Q12=+500 ;FEED RATE FOR MILLING
    Q18=+0 ;COARSE ROUGHING TOOL
    Q19=+100 ;RECIPROCATION FEED RATE
8 CYCL CALL
9 L Z+100 R0 FMAX M3
10 TOOL CALL 2 Z S3000
11 CYCL DEF 22 FINE-ROUGHING
    Q10=-5 ;PLUNGING DEPTH
    Q11=+150 ;FEED RATE FOR PLUNGING
    Q12=+750 ;FEED RATE FOR MILLING
    Q18=+5 ;COARSE ROUGHING TOOL
    Q19=+100 ;RECIPROCATION FEED RATE
12 CYCL CALL
13 L Z+100 R0 FMAX M02
14 END PGM MILL MM
```

Block/Meaning

Block 3: Tool call, cutter diameter 10
 Block 4: Retract tool
 Block 5: Select contour to be machined
 Blocks 6 and 7: Define coarse roughing machining data
 Block 8: Call machining: Contour-parallel coarse roughing (see figure 1)
 Block 9: Retract tool
 Block 10: Tool call, cutter diameter 4
 Block 11: Define fine roughing machining data
 Block 12: Call machining: Contour-parallel fine roughing (see figure 2)
 Block 13: Retract in the tool axis, end program



Figure 1: Traverse paths during roughing



Figure 2: Traverse paths during fine roughing

- angle encoders
- linear encoders
- contouring controls
- digital readouts
- length gauges
- rotary encoders

*How well-conditioned does
a control have to be?*



Perfect performance depends on proper conditioning and professional techniques. That's why the TNC is the right control for your daily milling operations: Its forward-looking precontrol techniques, integrated digital motor control and jerk-limited path control result in workpieces with very high surface definition. The dimensional accuracy of the finished workpieces can be attributed largely to HEIDENHAIN linear and angular encoders, which capture the axis motions directly at the machine table. That's how you can achieve quick gains and stay in excellent shape. HEIDENHAIN (G.B.) Limited, 200 London Road, Burgess Hill, West Sussex RH 15 9RD, Great Britain, phone (01444) 247711, fax (01444) 870024, e-mail: info@heidenhain.co.uk

HEIDENHAIN