HEIDENHAIN

iTNC 530

The Versatile Contouring Control for Milling, Drilling, Boring Machines and Machining Centers

March 2008

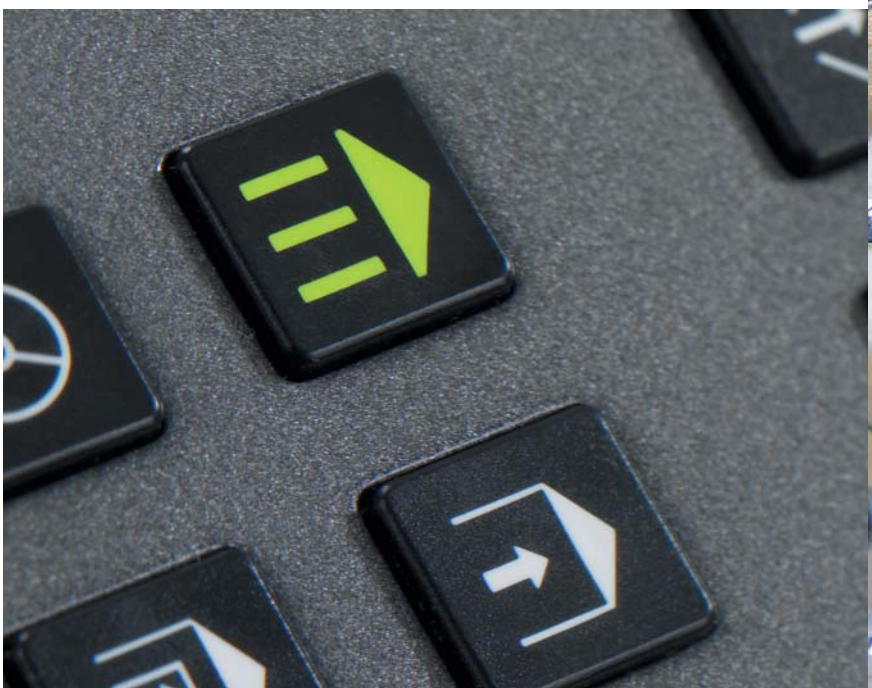
Start smart

For almost 30 years, TNC controls have been proving themselves in daily use on milling, drilling and boring machines, as well as machining centers. This success is due in part to their shop-oriented programmability, but also to their compatibility with programs of the predecessor models.

With the new **smarT.NC** operating mode, HEIDENHAIN has made yet another step forward towards greater ease of use. smarT.NC continues the success story of this shop-floor programmable user interface. 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.

And in spite of smarT.NC's entirely different user interface, it stands squarely on the proven HEIDENHAIN plain language dialog. Because in the background—out of view for the user—smarT.NC generates the program in conversational format.

HEIDENHAIN controls are powerful, user-friendly, and upwardly compatible so they are **prepared for the future** and allow you to look forward with confidence.



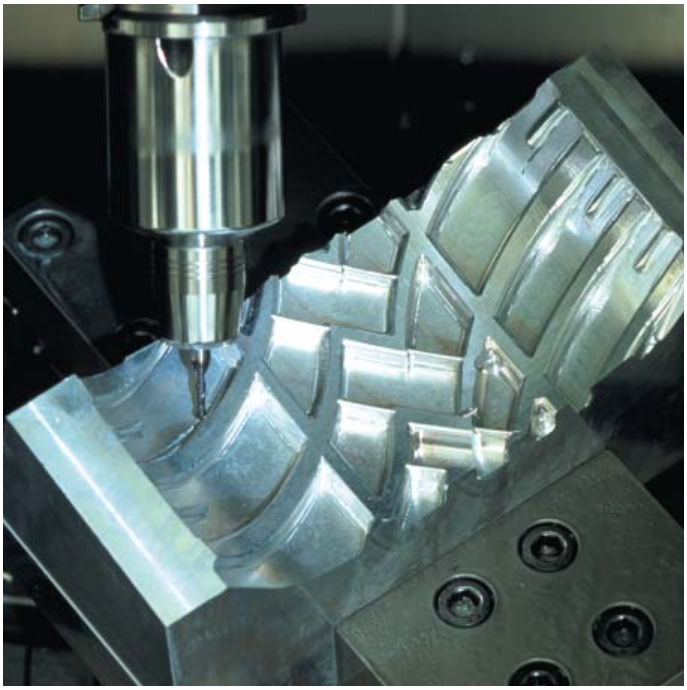
Contents

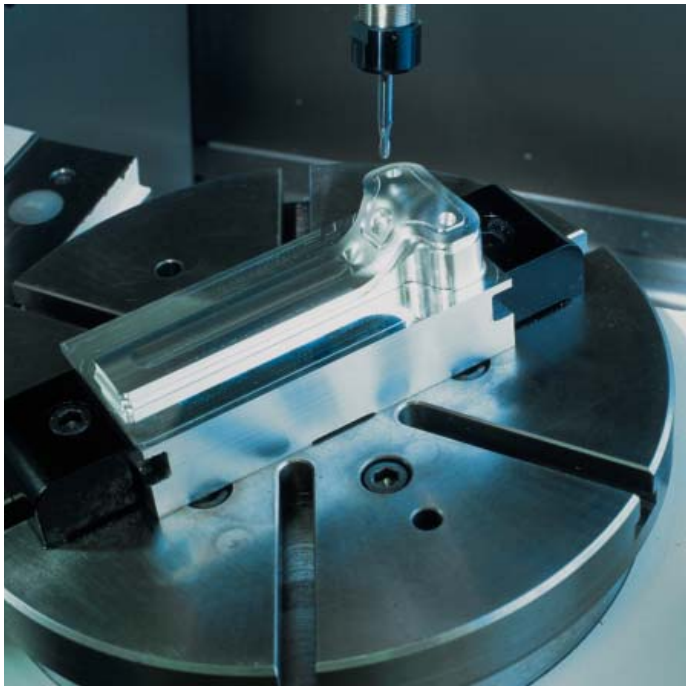
The iTNC 530...

Universally Applicable
– The Right Control for Scores of Applications

The iTNC 530 is versatile. It adapts optimally to the needs of your company—regardless of whether you are manufacturing single parts or batches, simple or complex parts, whether your shop works "on call" or is centrally organized.

The iTNC 530 is flexible. Do you prefer to work at the machine or at a programming station? With the iTNC 530 you can easily do both, because it is just as powerful in its **shop-floor programmability** as it is for **offline programming:**
You can program your own conventional milling, drilling, and boring operations at the machine in dialog with the control. The iTNC 530 gives you optimal support with smarT.NC or plain language—the conversational guidance from HEIDENHAIN—as well as with numerous graphic aids including practice-oriented fixed cycles. For simple work—such as face milling—you need not write a program, since it is easy to operate the machine manually with the iTNC 530. The iTNC 530 can be programmed remotely just as well—for example on a CAD/CAM system or at a HEIDENHAIN programming station. Your Ethernet interface guarantees very short transfer times, even of long programs.





Universal milling machine
- Workshop programming in HEIDENHAIN conversational format or with smarT.NC
- Upwardly compatible programs
- Fast presetting with HEIDENHAIN 3-D touch probe
- Electronic handwheel

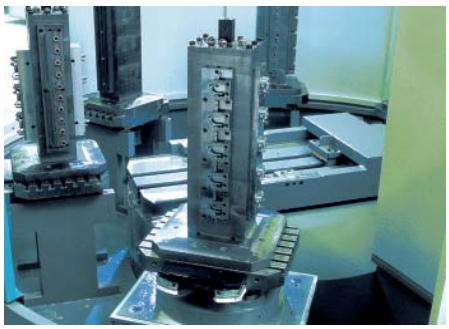
High speed milling
- Fast block processing
- Short control-loop cycle time
- Jerk-free path control
- High spindle speed
- Fast data transfer

Five-axis machining with swivel head and rotary table
- When you are programming away from the machine, the iTNC 530 automatically takes the machine geometry into account
- Tilting the working plane
- Cylindrical surface machining
- Tool Center Point Management (TCPM)
- 3-D tool compensation
- Fast execution through short block processing times

The iTNC 530 is universal. Its broad and complex range of applications proves it. Whether on simple 3-axis universal milling machines in tool and mold making, or on machining centers in interlinked production—in either case, the iTNC 530 is the right control. And it offers the applicable features both necessary and helpful.

Five-axis machining on very large machines
- Inspecting and optimizing machine accuracy with KinematicsOpt
- Global program settings for superimposition of various functions
- Procedure with handwheel superimposition in the virtual tool axis

Drilling and boring
- Cycles for drilling, boring and spindle alignment
- Drilling oblique holes
- Control of quills (parallel axes)



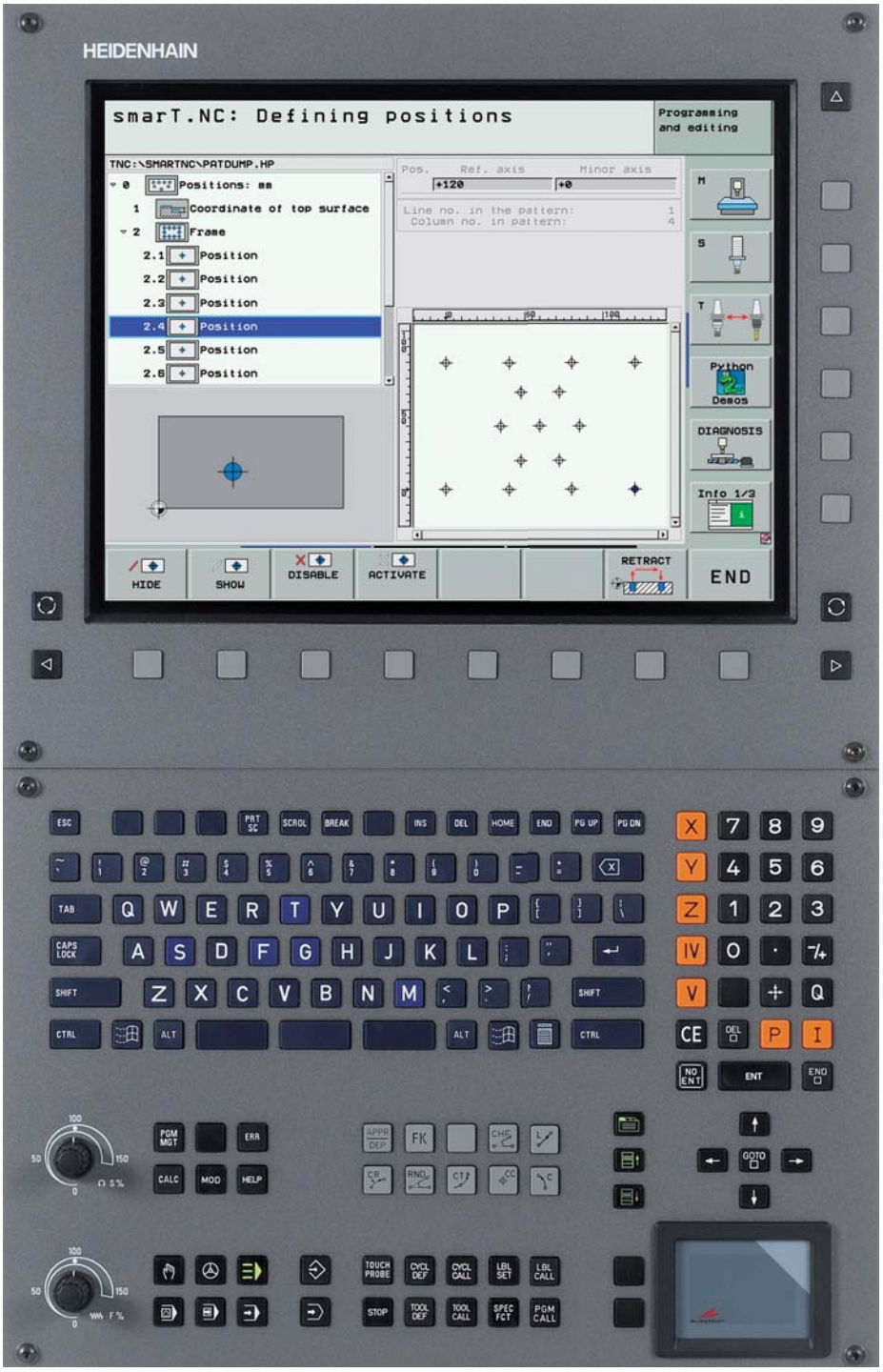
Machining centers and automated machining
- Tool management
- Pallet management
- Tool-oriented machining
- Controlled presetting
- Reference-point management with preset tables
- Automatic workpiece measurement with HEIDENHAIN 3-D touch probes
- Automatic tool measurement and breakage inspection
- Connection with host computer

Well Designed and User Friendly
– The iTNC 530 in Dialog with the User

The monitor
The TFT 15-inch color monitor shows a clear overview of all relevant information for programming, operating and monitoring the machine tool and control such as program blocks, comments and error messages. More information is provided through graphic support during program entry, test run and actual machining.

The selectable "split screen" display shows the part program blocks in one half of the screen and the graphics or the status display in the other half.

During the course of the program, status displays will always offer information on tool position, the current program, active cycles and coordinate transformations, and other data. The iTNC 530 even shows the current machining time.



The keyboard
As with all TNCs from HEIDENHAIN, the keyboard is tailored to the programming process. The well thought-out configuration of keys facilitates program input. Simple words and abbreviations or unambiguous symbols clearly indicate each key's function. Certain functions are entered through the iTNC 530's soft keys. The control is also equipped with a QWERTY alphabetic keyboard for DIN/ISO programming or for adding comments to a program. And it features a complete set of PC keys and a touchpad for using Windows functions.

Keys on the monitor

Select the screen layout

Display machine mode or programming mode

Soft keys for selecting functions in screen

Shift between soft-key rows

The smarT.NC mode is activated with a separate key and can be navigated with additional green keys.

Keys on the control panel

Program/file management, TNC functions

PGM MGT — Program management: Manipulate and delete programs

MOD — Supplementary operating modes

HELP — HELP function

ERR — Display error messages

CALC — Show pocket calculator

Machine operating modes

Manual Operation

Electronic Handwheel

Positioning with Manual Data Input

Program Run, Single Block

Program Run, Full Sequence

smarT.NC

Navigation

smarT.NC: Select next form

smarT.NC: Select previous/next frame

Programming modes

Programming and editing

Test run with graphic simulation

L, CHF — Straight line, chamfer

CC, C — Circular arc with center point

CR — Circular path with known radius

CT — Circular arc starting tangentially

RND — Corner rounding

APPR DEP — Contour approach and departure

FK — Free contour programming

P — Entry of polar coordinates

I — Entry of incremental dimensions

Q — Entry of a parameter instead of a fixed numerical value/Definition of parameters

Transfer of actual position to program

TOOL DEF, TOOL CALL — Definition and calling of tools

CYCL DEF, CYCL CALL — Definition and calling of cycles

LBL SET, LBL CALL — Labeling and calling of subroutines and program repeats

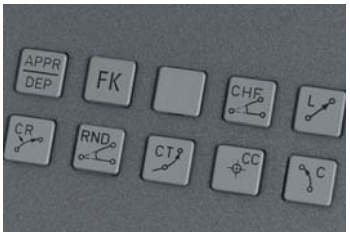
PGM CALL — Programmable program call

STOP — Programmed stop, interrupt/ discontinue

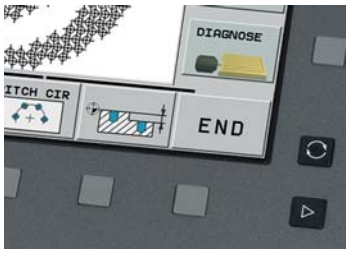
TOUCH PROBE — Touch probe functions

SPEC FCT — Special functions, e.g. TCPM or PLANE

With the gray path function keys and conversational guidance, you program line segments and circular arcs defined in various ways.



Many functions are entered by soft key.



Use the blue keys on the alphabetic keyboard to enter programs in accordance with the DIN/ISO standard.

Consistently Upward Compatible
– A Promising Future with HEIDENHAIN Contouring Controls

For over 20 years, HEIDENHAIN has been providing customers with contouring controls for milling, drilling and boring. Of course the controls have undergone development during this period: many new features have been added—also for more complex machines with more axes. The basic operational technique, however, has remained the same. The machinist who has been working with TNC does not have to relearn. On the iTNC 530 he immediately uses all of his previous experience with TNCs, programming and working as before.

2004: iTNC 530 with smarT.NC

2003: iTNC 530 with Windows XP

2001: iTNC 530

1997: TNC 426 M
TNC 430

1993: TNC 426 C/P

These contouring keys from the TNC 145 are also on the iTNC 530

1988: TNC 407
TNC 415

1987: TNC 355

1984: TNC 155

1983: TNC 150

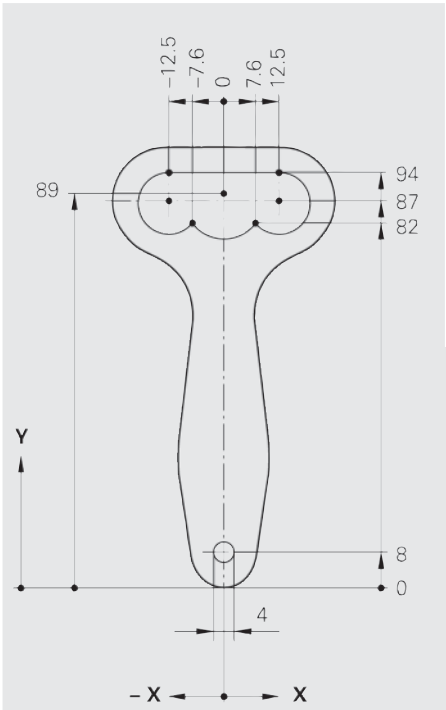
1981: TNC 145, the first contouring control from HEIDENHAIN

"Old" programs also run on newer TNC controls

Part programs from your NC program archive that were written on older TNC contouring controls can also be run on an iTNC 530 with very little effort. This ensures the greatest possible flexibility in machine utilization and provides enormous cost benefits if you find you have to produce "old" parts again. HEIDENHAIN contouring controls put you in the position to manufacture a replacement part quickly and economically even now—after more than 20 years—without having to reprogram it.

Familiar function keys take on new tasks

Of course many innovations and improvements have been included in the iTNC 530—but the fundamental programming procedure has remained the same. When you switch to a new control you do not need to relearn the programming and operation. You only need to become familiar with the new functions. So you can apply your professional knowledge as a machinist immediately even on a new TNC control.



Inside contour—

31	CC	X+0,000	Y+8,000	
32	C	X+0,000	Y+0,000	
			DR+ RRF ... M 98	
33		Z+10,000		Werkzeug-Achse vom Werkstück wegfahren
			R0 F9999 M	
34	L	X+7,600	Y+82,000	Innenkontur anfahren
			RL F9999 M	
35		Z+1,000		In das Werkstück einstechen
			R0 F9999 M	
36		Z . . .		
			R0 F . . . M	
37	CC	X+12,500	Y+87,000	Innenkontur fräsen
38	C	X+12,500	Y+94,000	
			DR+ RL F . . . M	
39	L	X–12,500		
			RL F . . . M	
40	CC	X–12,500	Y+87,000	
41	C	X–7,600	Y+82,000	
			DR+ RL F . . . M	

programmed with the TNC 145...

...machined with the iTNC 530

Machining with Five Axes
– The iTNC 530 Permits Optimum Tool Movement

Modern machines often work with four or five positioning axes. This makes it possible to machine complex 3-D contours. The required programs are usually created on external CAD/CAM systems and comprise a large number of very short line segments that are transferred to the control. Whether the workpiece is actually machined according the program's instructions depends essentially on the geometric behavior of the control. With its optimized path control, its precalculation of the contour and its algorithms for jerk limitation, the iTNC 530 has the right functions for a perfect surface in the shortest possible machining time. See for yourself. In the end, it's the quality of the workpiece that proves the performance of the control.

3-D contour machining at its finest
The iTNC 530's **short block processing time** of only 0.5 ms for a 3-D line segment without tool compensation permits high traversing speeds even on complex contours. This enables you, for example, to mill molds or dies approximated with 0.2 mm line segments at feed rates as high as 24 meters per minute.

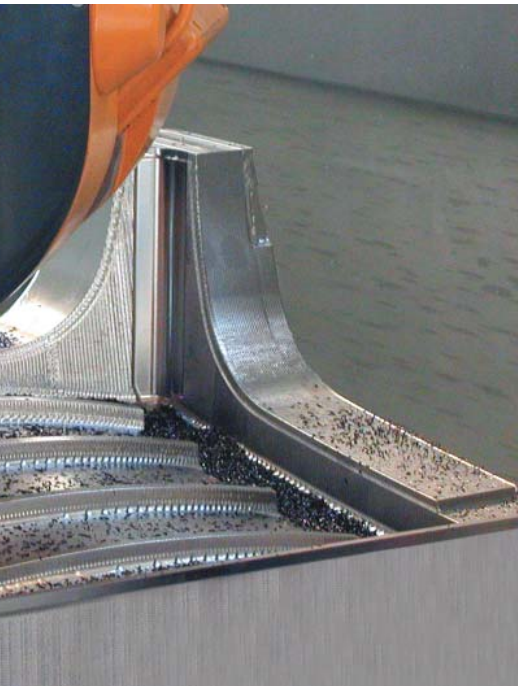
The particularly **jerk-free path control** when machining 3-D figures and the **defined rounding** of series of straight-line segments provide you with smoother surfaces as well as high dimensional accuracy.

The iTNC 530 thinks ahead. Its "look-ahead" function anticipates future changes in direction by adjusting the traversing speed to the programmed surface. If desired, it also enables the iTNC 530 to reduce the feed rate when plunging the tool into the workpiece. This lets you simply program the maximum machining speed as the feed rate. The iTNC 530 automatically adapts the actual speed of the workpiece contour to save you machining time.

For NC programs with normal vectors, such as those generated by CAD/CAM systems, the iTNC 530 automatically calculates a 3-D tool compensation for end mills, ball-nose cutters, or toroid cutters.

Guided tool tip
CAD/CAM systems use postprocessors to generate five-axis programs. In principle, such programs contain either all coordinates of the machine's existing NC axes, or NC blocks with surface normal vectors. When machining with five axes (three linear axes and two tilting axes)*, the tool can stay perpendicular, or if desired, inclined at a predetermined angle to the workpiece surface.

Regardless of what type of 5-axis programs you wish to run, the iTNC 530 makes all the compensating movements in the linear axes that result from movements in the tilting axes. The iTNC 530's **Tool Center Point Management** feature (TCPM)—an improvement upon the proven TNC function M128—provides optimal tool guidance and prevents contour gouging.



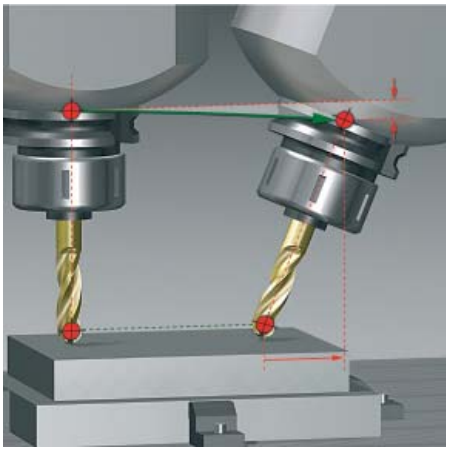
With TCPM you can define the behavior of the tilting and compensating movements automatically calculated by the iTNC 530.

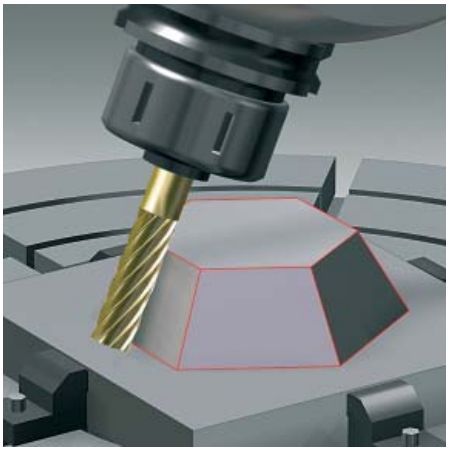
TCPM defines the **interpolation between the start and end positions:**
- During **face milling**—machining mainly with the front face of the tool—the tool point moves on a straight line. The path of the tool's cylindrical surface is not defined, but rather it depends on the machine geometry.
- During **peripheral milling** machining is mainly by the side of the tool. The tool tip also travels on a straight path, but additionally the tool's circumference machines an explicitly defined plane.

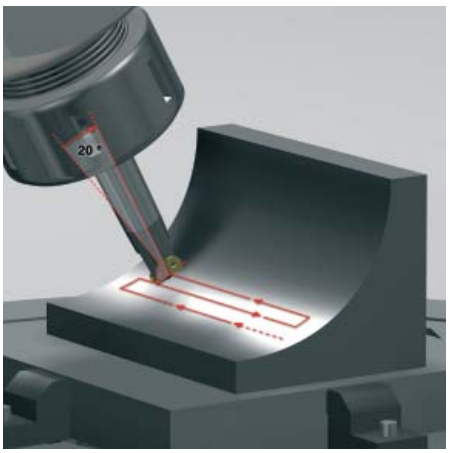
TCPM defines the effect of the **programmed feed rate** as desired either...
- as the actual velocity of the tool tip relative to the workpiece. Very high axis feed rates can result from large compensating motions during machining near the center of tilting.
- as contouring feed rate of the axes programmed in the NC block. The feed rate is usually lower, but you attain better surface quality during large compensating movements.

With TCPM you can also define the **effect of the inclination angle** for more uniform cutting passes when working with an inclined radius cutter:
- Angle of inclination defined as axis angle
- Angle of inclination defined as spatial angle

The iTNC takes the inclination angle into account in all 3-D machining—even with 45° swivel heads or tilting tables. You either specify the angle of inclination in the NC program via a miscellaneous function, or adjust it manually with an electronic handwheel. The iTNC 530 makes sure that the tool remains on the contour and does not damage the workpiece.







* These functions must be implemented by the machine tool builder.

Machining with Five Axes
– Swivel Head and Rotary Table Controlled by iTNC

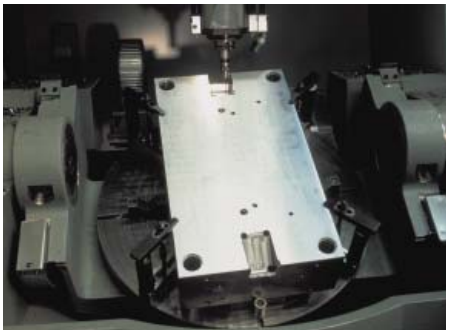
Many five-axis operations that at first glance may seem very complex can be reduced to conventional 2-D movements that are simply tilted about one or more rotary axes or wrapped onto a cylindrical surface. The iTNC supports you with application-oriented functions to help you write and edit such programs quickly and simply without a CAD/CAM system.

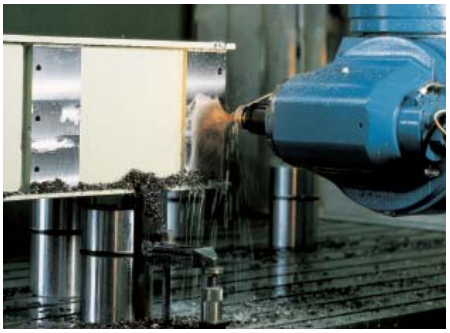
Tilting the working plane*
Programs for contours and holes on inclined surfaces are often very complex and require time-consuming computing and programming work. Here the iTNC 530 helps you to save a great deal of programming time.
You program the part as usual in the working plane (e.g. the X/Y plane), but it is machined in a plane that is rotated in one or more axes about the main plane.

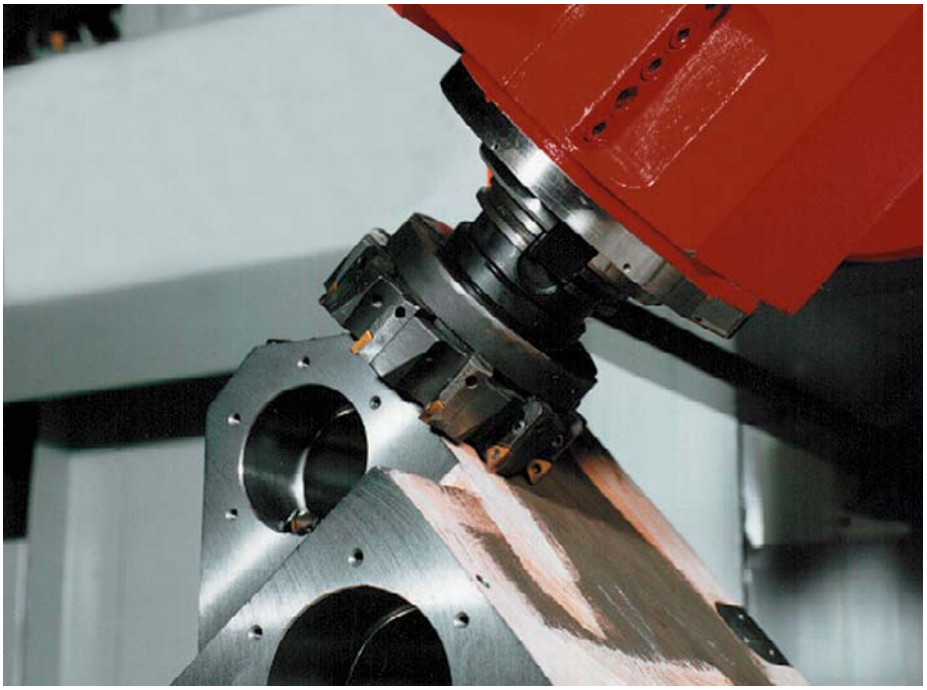
The PLANE feature makes it easy to define a tilted working plane: You can specify tilted working planes in seven different ways, depending on the information on the workpiece drawing. In order to keep the use of these complex functions as simple as possible, a separate animation is available for each possible plane definition, so that you can view them before selecting the function. Clearly arranged support graphics assist you during input.

You can define the rotary positioning behavior with the PLANE function so that there are no unpleasant surprises when the program is run. The settings for defining the positioning behavior are identical for all PLANE functions, making everything that much easier.

* These functions must be implemented by the machine tool builder.







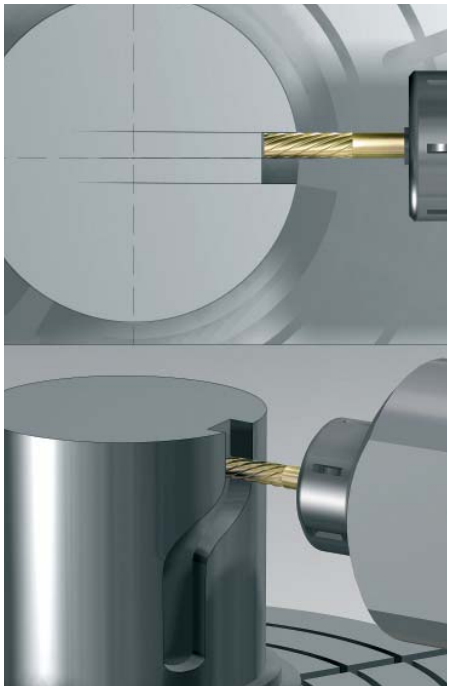
Machining cylindrical surfaces*
With the iTNC 530 it is quite easy to program contours (which consist of straight lines and arcs) on cylindrical surfaces using rotary and tilting tables: You simply program the contour in a plane as if the cylinder surface were unrolled. The iTNC 530 then executes the operation on the surface of the cylinder.

The iTNC 530 features four cycles for cylindrical surface machining:
- Slot milling (the slot width is the same as the tool diameter)
- Guide-groove milling (the slot width is greater than the tool diameter)
- Ridge milling
- Mill outside of contour

* These functions must be implemented by the machine tool builder.

Manual axis motion in the tool direction on 5-axis machines
The safe retraction of a tool is very important with five-axis machining. The "virtual tool axis" function is of assistance here. You can use it to traverse the tool in the current direction of the tool axis via the external direction keys or the handwheel. This function is especially useful if you want to
- retract the tool in the direction of the tool axis during interruption of a five-axis machining program, or
- use the handwheel or external direction keys to perform an operation in Manual mode and the tool is in contact.
- move the tool with the handwheel in the active tool axis direction during machining.

Linear feed rate for rotary tables in mm/min*
While the standard units of measure for programming feed rates for rotary tables are degrees per minute, the iTNC 530 can also interpret a feed rate programmed in millimeters or inches per minute. The feed rate at the contour is then independent of the distance of the tool center from the center of the rotary axis.



Intelligent Machining
– Dynamic Collision Monitoring Option (DCM)

The complex motions and high traversing speeds of five-axis machining make axis movements difficult to foresee. This makes collision monitoring a valuable function that relieves the machine operator and protects the machine from damage.

NC programs from CAM systems may avoid collisions of the tool or tool holder with the workpiece, but unless you invest in expensive offline machine simulation software they ignore the machine components located within the work envelope. And even then it cannot be guaranteed that machine conditions, such as the fixture position, will be identical to those of the simulation. In the worst case, a collision will remain undetected until the damage is done.

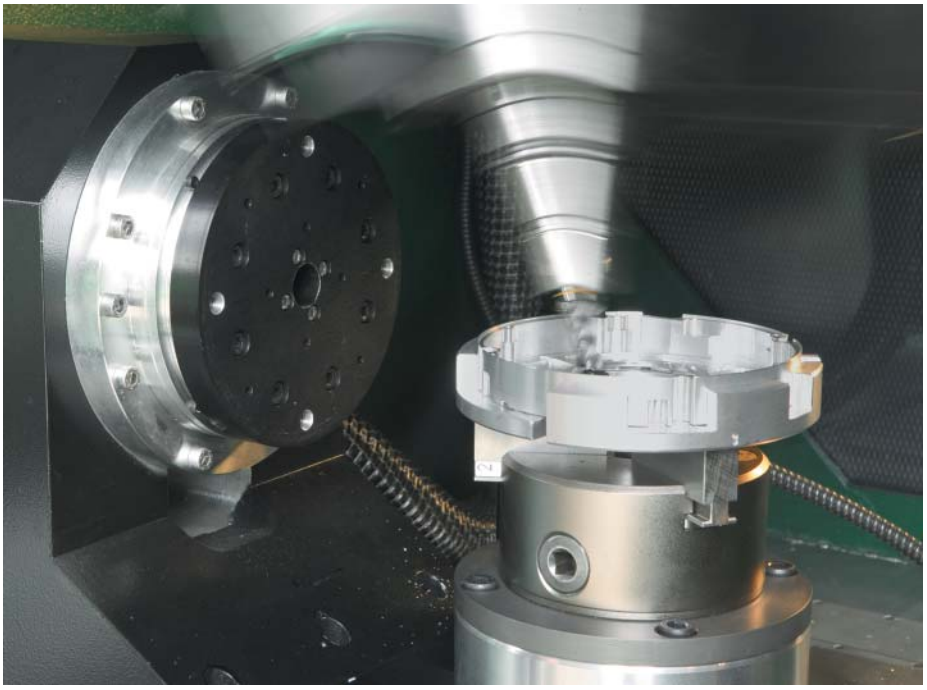
In cases such as these, the machine operator is supported by the **DCM dynamic collision monitoring*** feature of the iTNC 530 (only runs on the MC422 B/C). The control interrupts machining whenever a collision is imminent, thereby increasing the safety for the machine and its operator. This helps to prevent machine damage, which can result in costly downtimes. Unattended shifts become safer and more reliable.

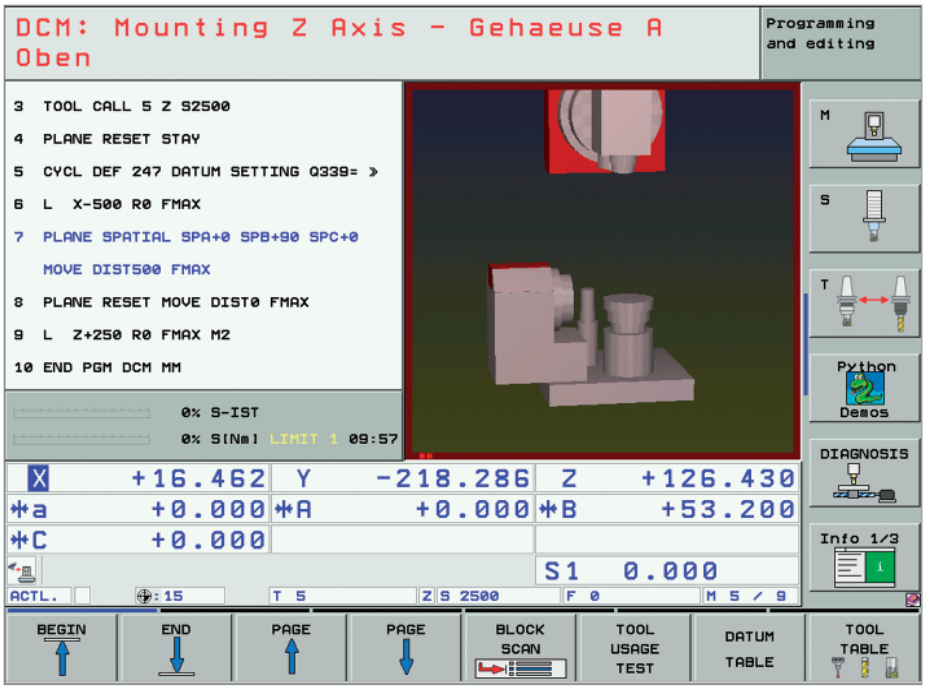
However, DCM works not only in **automatic mode.** It is also active in **manual operation.** If, for example, during setup the machine operator takes a collision course to a component in the working space, the iTNC 530 detects it, stops axis movement, and issues an error message.

If two machine components come too close to each other, there are **three warning stages:**

- Advance warning if below 14 mm clearance
- Warning if below 8 mm clearance
- Error if below 2 mm clearance

* These functions must be implemented by the machine tool builder.





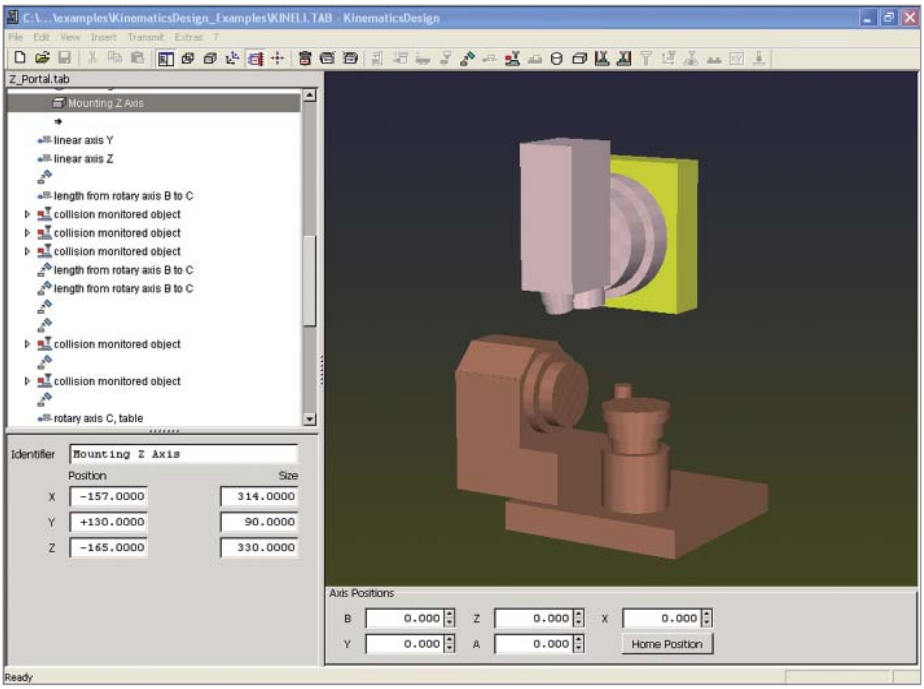
Of course the iTNC 530 also shows the machine operator—both with an error message and graphically—which machine components are endangered. If a collision warning is displayed, the TNC permits retracting the tool only in those directions which increase the clearance between the colliding objects.

The machine tool builder takes care of the required definition of **machine components.** The working space and the collision objects are described using geometric bodies such as planes, cubes and cylinders. Complex machine components can be modeled with multiple geometric bodies. The tool is automatically considered a cylinder of the tool radius (defined in the tool table). For tilting devices, the machine tool builder can use the tables for the machine kinematics also to define the collision objects.

The last step of the configuration process is defining which machine components can collide. Because the machine design in itself prevents collisions between certain machine components, they can be ruled out from the start. For example, a tool touch probe like the HEIDENHAIN TT clamped on the machine table can never come into contact with the machine cabin.

When using the dynamic collision monitoring, please note:

- While DCM can help reduce the danger of collision, it cannot eliminate it.
- Only the machine manufacturer can define collision objects.
- 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 (which means without feedforward).
- It is not possible to monitor for collision before machining.



Intelligent Machining
– Option for Adaptive Feed Rate Control (AFC)

Besides the feed rate for each block or cycle, HEIDENHAIN controls have always allowed the programmer to enter a manual compensation through the override potentiometer to adjust for the actual machining situation. But this always depends on the experience and, of course, the presence of the operator.

Adaptive feed rate control (AFC) automatically regulates the feed rate of the TNC, taking into consideration the respective spindle power and other process data.

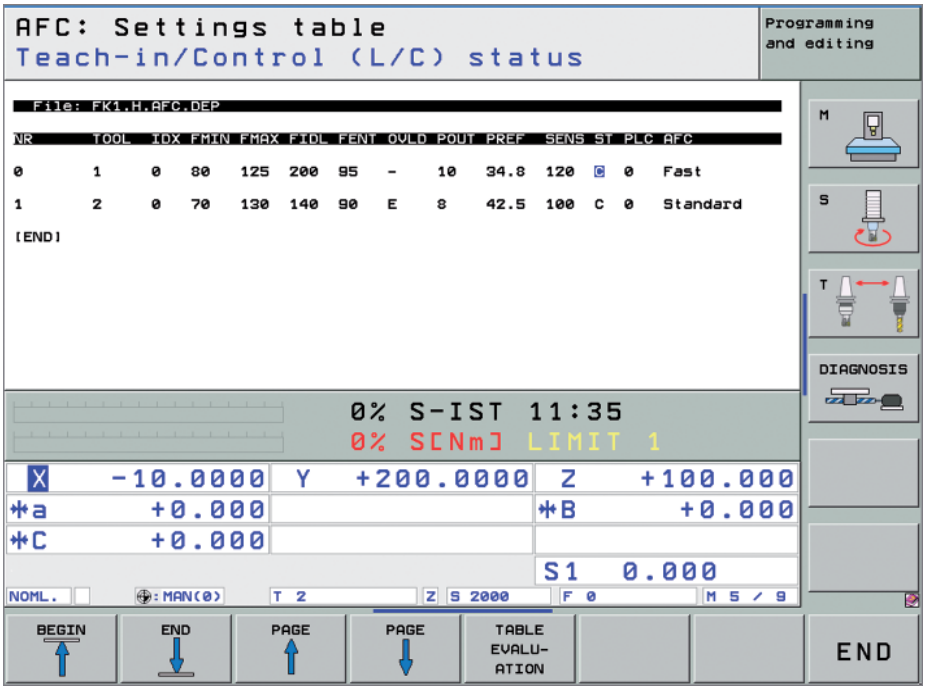
In a teach-in cut, the iTNC records the maximum spindle power. Then, before actual machining, you define in a table the respective limit values between which the iTNC can influence the feed rate in the adaptive control mode in the "control" mode. Of course, various overload reactions can be provided for, which can also be defined by your machine tool builder.

Adaptive feed rate control offers various advantages:

Optimizing the machining time
Fluctuations in dimensions or material (blowholes) often appear particularly on cast parts. With a corresponding adaptation of the feed rate, the control tries to keep the previously "learned" maximum spindle power during the entire machining time. The total machining time is shortened by an increased feed rate in the machining zones with less stock removal.

Tool monitoring
The iTNC's adaptive feed rate control permanently compares the spindle power with the feed rate. As a tool becomes blunt, the spindle power increases. As a result, the iTNC reduces the feed rate. As soon as the feed rate falls below a defined minimum, the iTNC reacts with an error message or by switching off. This prevents damage resulting from tool breakage or wear.

Protection of the machine mechanics
Reducing the feed rate down to the reference value whenever the learned maximum permissible spindle power is exceeded also reduces the strain and wear on the machine. It effectively protects the spindle from overload.



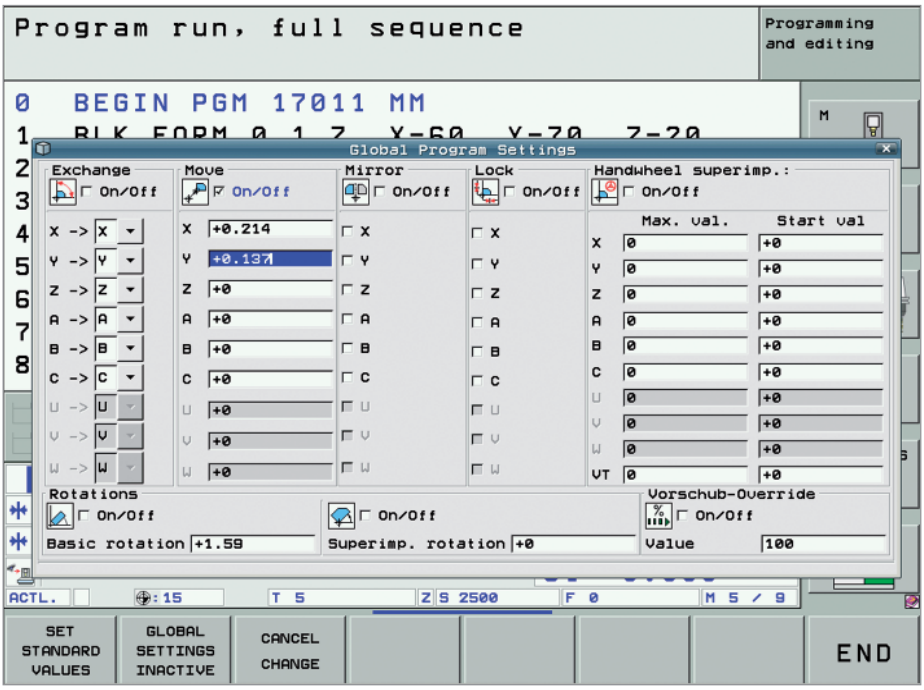
– Global Program Settings (Option)

The global program settings, which come into play particularly in large-scale mold making, are available in the Program Run and MDI modes. It allows you to define various coordinate transformations and settings with global and priority effect for the selected NC program, without having to edit it.

You can change the global program settings during a program stop even in mid-program. After program start the iTNC then moves, if necessary, to a new position with a positioning logic influenced by you.

The following functions are available:

- Exchanging axes
- Additional, additive datum shift
- Superimposed mirroring
- Axis locking
- Handwheel superimposition, with axis-specific memory of paths covered per handwheel, also in **virtual axis direction**
- Superimposed basic rotation
- Superimposed rotation
- Globally valid feed-rate factor

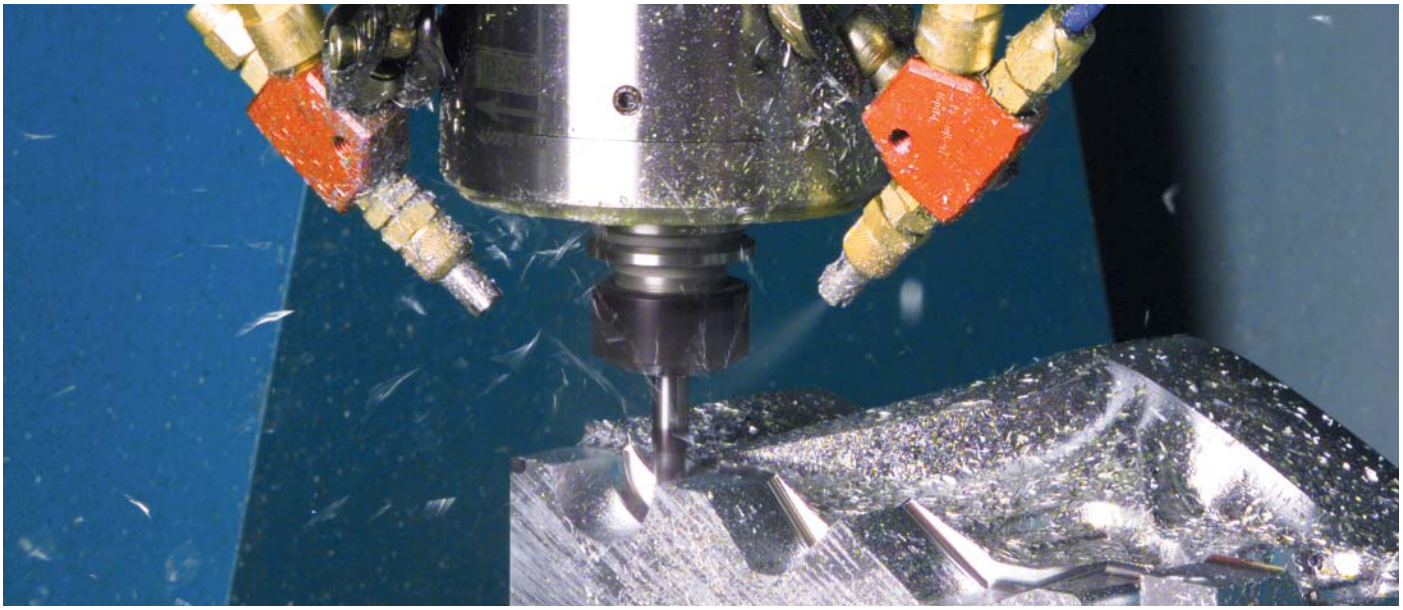


Higher Speed, More Accuracy, Truer Contours
– High Speed Milling with the iTNC 530

High speed cutting
High speed cutting stands for quick and efficient contour milling. The control must be able to transfer large amounts of data quickly, make long programs efficient to edit, and produce the desired ideal contour on the workpiece—all qualities that the iTNC 530 possesses.

Very short block processing times
Sophisticated feedforward servo methods are progressively pushing block processing as an influence ever further into the background. Nevertheless, short block processing times remain the best solution for certain machining tasks. One example is the machining of highly accurate contours with very high resolution. This is no problem for the iTNC 530. Here the control provides ideal performance with block processing times of less than one millisecond.

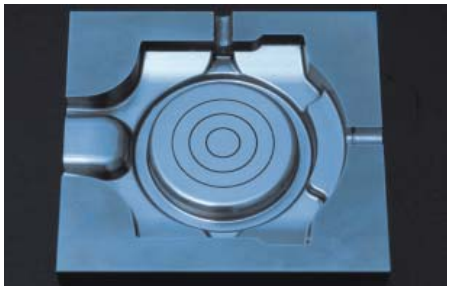
Very high contour accuracy
The iTNC 530 calculates the contour for up to 1024 blocks in advance. This enables it to adapt the axis velocities to the contour transitions. It controls the axes with special algorithms that ensure path control with the required limits to velocity and acceleration. The integrated filters can specifically suppress the natural frequencies of an individual machine while maintaining the desired surface accuracy.

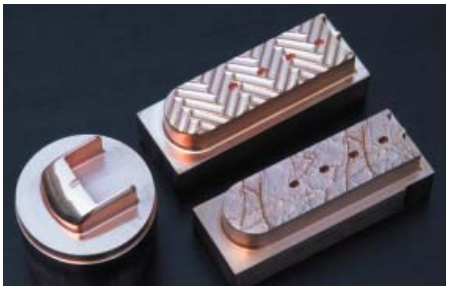


Fast machining at specified accuracy
You as user specify the accuracy of the machine contour—apart from the NC program. You simply enter in the control through a cycle the maximum permissible deviations from the ideal contour. The iTNC 530 automatically adapts the machining to the tolerance that you define. No contour damage occurs with this method.

Spline interpolation
If your CAD/CAM system describes contours as splines, you can transfer them directly to the control. The iTNC 530 features a spline interpolator and can process third-degree polynomials.

Digital drive technology
The position controller, speed controller and, if required, the current controller are integrated in the iTNC 530. The digital motor control makes it possible to attain very high feed rates. While interpolating simultaneously in up to five axes, the iTNC 530 reaches the required cutting speeds by digitally controlling spindle speeds up to **60000 rpm.**



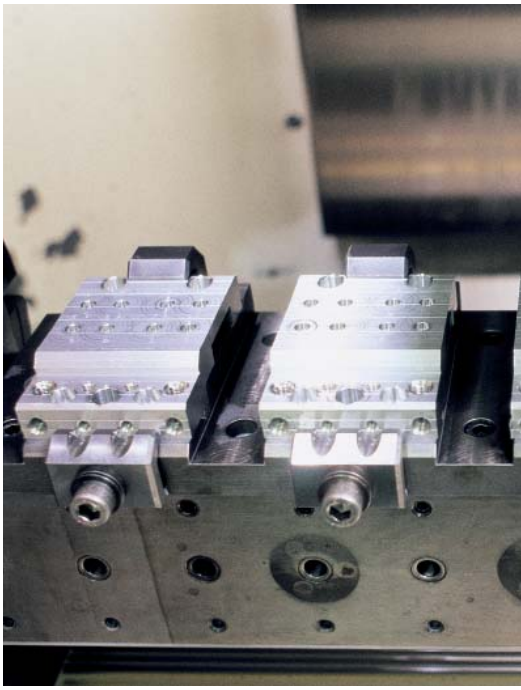


Automated Machining
– The iTNC 530 Manages, Measures and Communicates

The difference in requirements placed on the classical machine for tool and mold-making and machining centers are becoming ever less distinct. Today of course the iTNC 530 is particularly capable of controlling automated processes. It masters the range of functions needed to start the proper machining operations on individual workpieces in any setup and even in interlinked machining.

Tool management
For machining centers with automatic tool changers, the iTNC 530 offers a central tool file for up to 32 767 tools. The tool memory is a freely configurable file and can therefore be optimally fitted to your needs. You can even have the iTNC 530 manage your tool names. The control prepares the next tool change while the current tool is still cutting. This significantly reduces the non-cutting time required for changing tools.

Pallet management
The iTNC 530 can assign the appropriate program and datum shift to parts mounted on pallets and brought to the machine in a random sequence. If a pallet is exchanged, the iTNC 530 automatically calls the correct part program. This permits automatic machining of a variety of parts in any sequence.



Tool-oriented machining
With tool-oriented machining, one machining step is performed on all workpieces on a pallet before the next machining step. This reduces the number of tool changes to a necessary minimum and the machining time is significantly shorter.

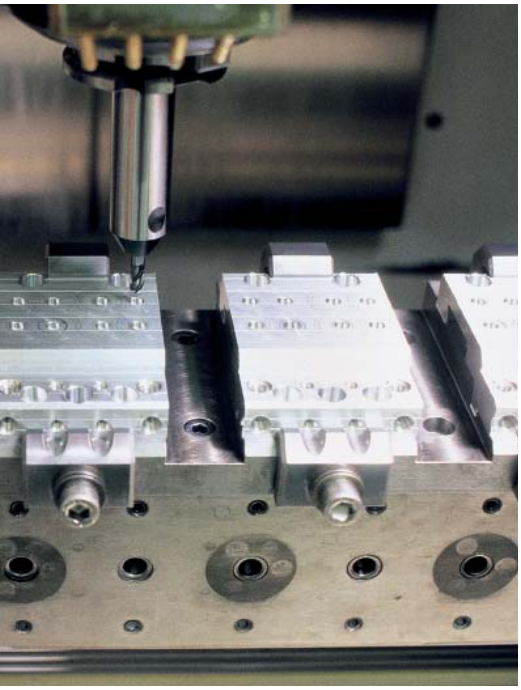
The iTNC 530 supports you with convenient input forms with which you can assign a tool-oriented machining operation to a pallet with several workpieces on several fixtures. You can write the program, however, in the familiar workpiece-oriented sequence.

You can also use this function even if your machine does not support pallet management. In the pallet file you then simply define the positions of the workpieces on your machining table.

Inspecting workpieces for proper machining and dimensional accuracy
The iTNC 530 features a number of measuring cycles for checking the geometry of the machined workpieces. To run the measuring cycles, you insert a 3-D touch probe from HEIDENHAIN (see page 40) into the spindle in place of a tool for the following tasks:

- Recognizing a workpiece and calling the appropriate part program
- Checking whether all machining operations were conducted correctly
- Determining infeeds for finishing
- Detecting and compensating tool wear
- Checking workpiece geometry and sorting parts
- Logging measured data
- Ascertaining machining error trends

Workpiece measurement and automatic compensation of tool data
Together with the TT 140, TL Nano and TL Micro systems for tool measurement (see page 41), the iTNC 530 can automatically measure tools while they are in the machine. The iTNC 530 saves the ascertained values of tool length and radius in the central tool file. By inspecting the tool during machining you can quickly and directly measure wear or breakage to prevent scrap or rework. If the measured deviations lie outside the tolerances, or if the monitored life of the tool is exceeded, the iTNC 530 locks the tool and automatically inserts a replacement tool.



Minimize Setup Times
– The iTNC 530 Makes Setup Easy

Before you can begin machining, you must first clamp the tool and set up the machine, find the position and orientation of the workpiece on the machine, and set the workpiece reference point. This is a time-consuming but indispensable procedure. After all, any error directly reduces the machining accuracy. Particularly in small and medium-sized production runs, as well as for very large workpieces, setup times become quite a significant factor.

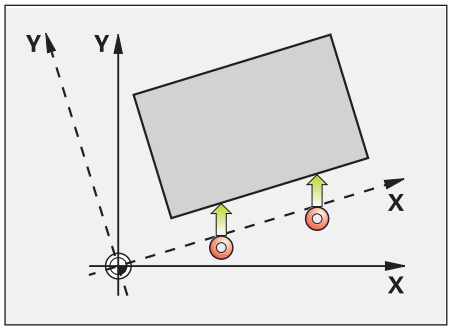
The iTNC 530 features application-oriented, real-world setup functions. They support the user, help to reduce non-productive time, and make overnight, unattended production possible. Together with the **3-D touch probes,** the iTNC 530 offers numerous probing cycles for automatic alignment of the workpieces, presetting, and measurement of the workpiece and the tool.

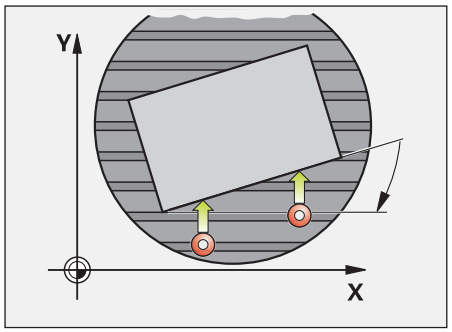
Delicate manual traverse
For setup, you can use the direction keys to move the machine axes manually or in incremental jog. A simpler and more reliable way, however, is to use the electronic handwheels from HEIDENHAIN (see page 42). Particularly with the portable handwheels you are always close to the action, enjoy a close-up view of the setup process, and can control the infeed responsively and precisely.

Workpiece alignment
With the 3-D touch probes from HEIDENHAIN (see page 40) and the probing functions of the iTNC 530, you can forgo any tedious manual alignment of the workpiece:

- Clamp the workpiece in any position.
- The touch probe ascertains the workpiece misalignment by probing a surface, two holes, or two studs.
- The iTNC 530 compensates the misalignment with a "basic rotation," which means that in the NC program the part is rotated by the measured misalignment.

Compensating workpiece misalignment
Compensate misalignment by rotating the coordinate system or turning the table





Workpiece presetting
You can use a reference point to assign a defined value in the iTNC display to any workpiece position. Finding this point quickly and reliably reduces nonproductive time and increases machining accuracy. The iTNC 530 features probing cycles for automatic presetting. Once found, you can save reference points

- in the workpiece preset table,
- in a workpiece datum table, or
- by directly setting the displayed value.

Preset table: The iTNC's central reference point management
The preset table makes flexible machining, shorter setup times and increased productivity possible. In other words, it makes it much easier to set up the machine.

In the preset table you can save **any number of reference points** and assign an individual basic rotation to each one.

When working in a **tilted plane** and presetting a reference point, the iTNC includes the respective positions of the rotary axes. In this way, the reference point also remains active in any other angular position.

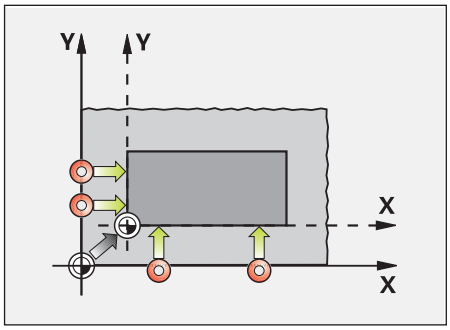
On machines with an **automatic spindle head changer,** the reference point remains unchanged after a head exchange, even if the heads differ in their kinematics (i.e. in their dimensions).

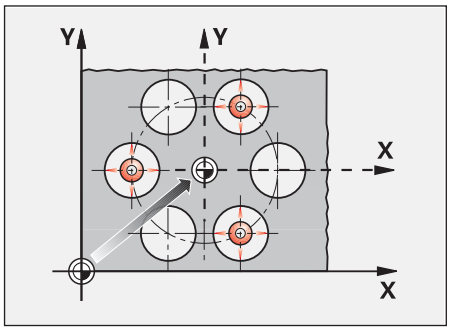
The iTNC automatically creates separate preset tables for individual **traverse ranges** (such as for alternating table machining) When changing traverse ranges, the iTNC activates the correct preset table with the most recently active reference point.

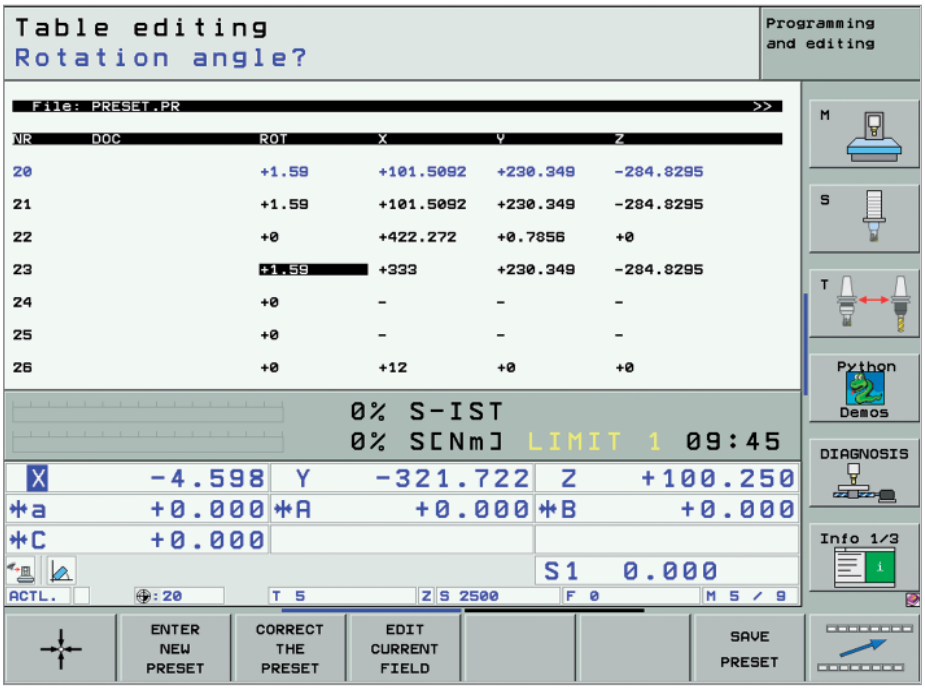
There are three ways to save reference points in the preset table:

- In the Manual mode by soft key
- By using the probing functions
- With the automatic probing cycles

Workpiece presetting
At a corner, for example, or in the center of a bolt hole circle







Programming, Editing, Testing
– The iTNC 530 Opens Endless Possibilities

The iTNC 530 is just as universal in application as it is flexible in machining and programming.

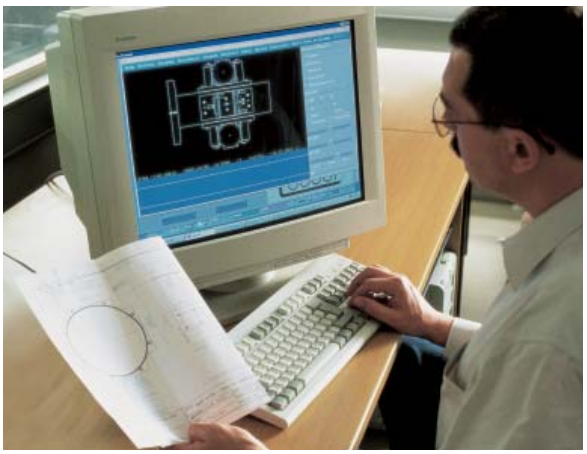
Programming at the machine
HEIDENHAIN controls are workshop oriented, which means that they were conceived for programming right at the machine. The iTNC 530 supports you with two user interfaces:

For almost 30 years, HEIDENHAIN **conversational programming** has been the standard programming language for all TNC controls and for shop-floor programming in general. The new **smarT.NC** intuitive and self-explanatory operating mode uses straightforward input forms to guide you through the complete NC programming process all the way to the actual machining. There's no need to learn G functions or any special programming languages. The control "speaks" with you using easily understandable questions and prompts. Whether plain-language prompts, dialog guidance, programming steps or soft keys, all texts are available in numerous languages.

If you are used to **DIN/ISO programming,** however, the iTNC is still the right control—you can enter DIN/ISO programs over the alphanumeric keyboard. The most frequently needed letters are already highlighted in color.

Positioning with manual data input
You can start working with the iTNC 530 even before writing a complete part program. Simply machine a part step by step—switching as you want between manual operation and automatic positioning.

Creating programs offline
The iTNC 530 is also well equipped for offline programming. The iTNC 530 can be integrated through its interfaces into networks and connected with programming stations, CAD/CAM systems or other data storage devices.



– Fast Availability of All Information

Do you have questions on a programming step but your User's Manual is not at hand? No problem: The iTNC 530 numerical control and iTNC 530 programming station now feature TNCguide, a convenient help system that can show the user documentation in a separate window.

You can activate the TNCguide by simply pressing the help key on the iTNC keyboard or by clicking any soft key with a cursor in the shape of a question mark. You switch the cursor by simply clicking the help symbol ( ) that appears on all TNC screens.

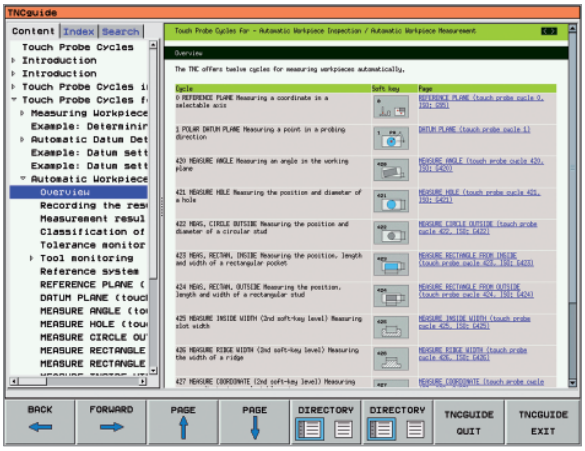
The TNCguide usually displays the information in the immediate context of the element in question (context-sensitive help). This means that you immediately receive the relevant information. This function is particularly helpful with the soft keys. The method and effect of operation is explained in detail.

The iTNC 530 is shipped with integrated documentation for the respective NC software in English and German. Other languages are available for download free of charge as soon as the translations become available. After download, you can save the national language files in the corresponding language directory on the TNC's hard disk.

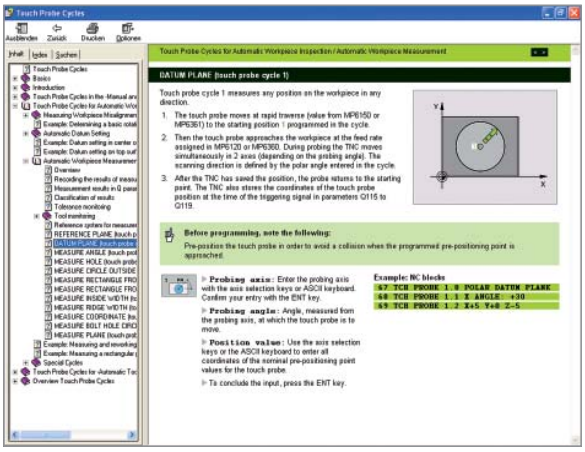
The following manuals are available in the help system:

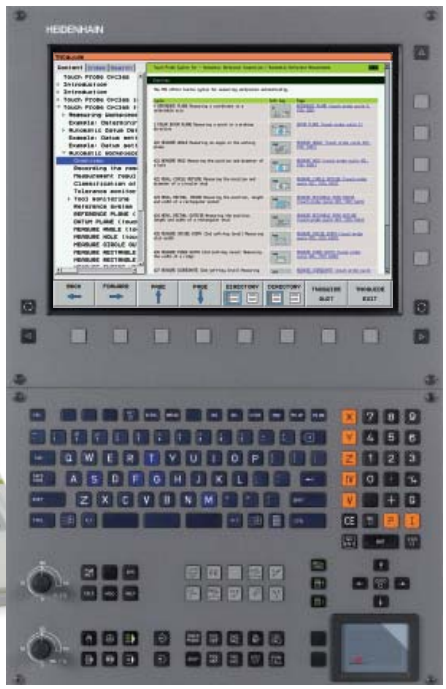
- User's Manual for Conversational Programming
- User's Manual for smarT.NC (same format as a "Pilot")
- User's Manual for Touch Probe Cycles
- User's Manual for DIN/ISO Programming
- User's Manual for the iTNC 530 Programming Station (only included in the programming station)

TNCguide integrated in the control, e.g. on the iTNC 530 ...



... or at the programming station.





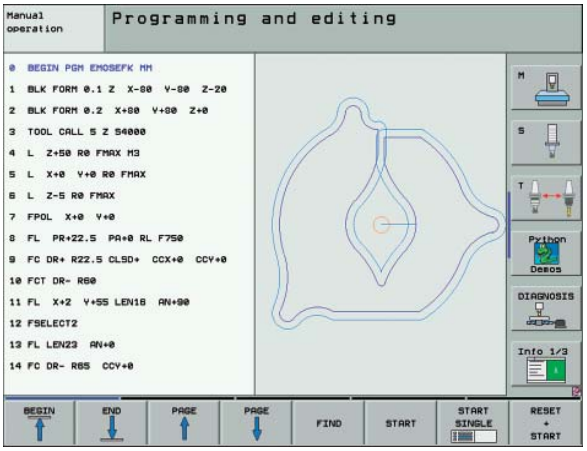
* Only with at least 256 MB RAM

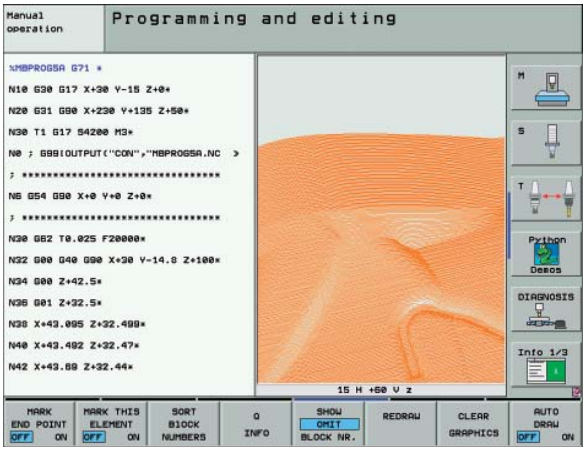
Programming, Editing, Testing
– Graphic Support in Any Situation

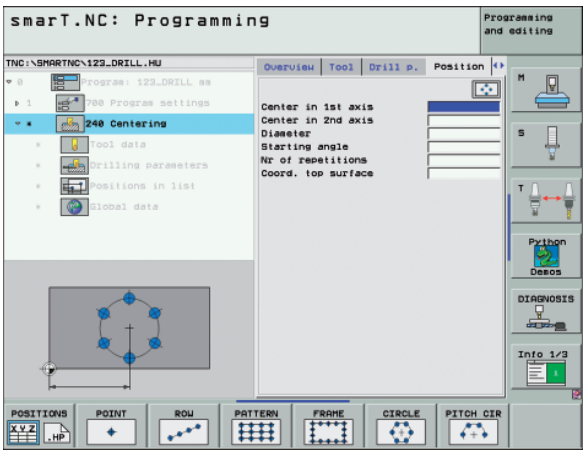
Programming graphics
The two-dimensional programming graphics give you additional security: while you are programming, the iTNC 530 draws every entered traverse command on the screen.

3-D line graphics
The 3-D line graphics display the programmed tool center point path in three dimensions. With the powerful zoom function you can also see the finest details. You should especially use the 3-D line graphics to inspect programs created offline for irregularities before machining, in order to avoid undesirable traces of the machining process on the workpiece, e. g. 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 highlighted in the left window. In addition, the respective programmed end points can be displayed to show any concentrations of points.

Help graphics
During cycle programming in the plain-language dialog, the iTNC shows a separate illustration for each parameter. This makes it easier to understand the function and accelerates programming. In smarT.NC you will find help graphics for all required input.







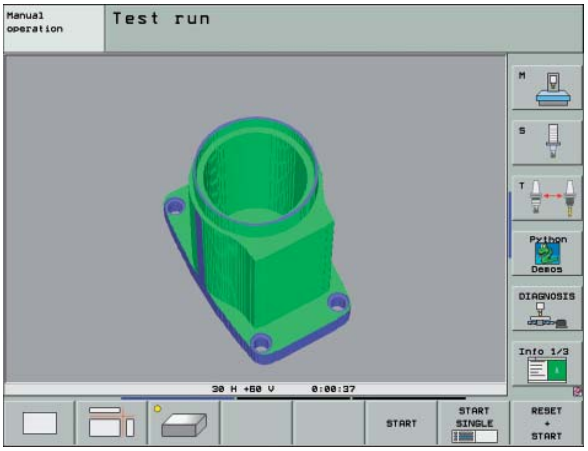
Program verification graphics
To play it safe before running a program, the iTNC 530 can graphically simulate the machining of the workpiece. It can display the simulation in the following ways:

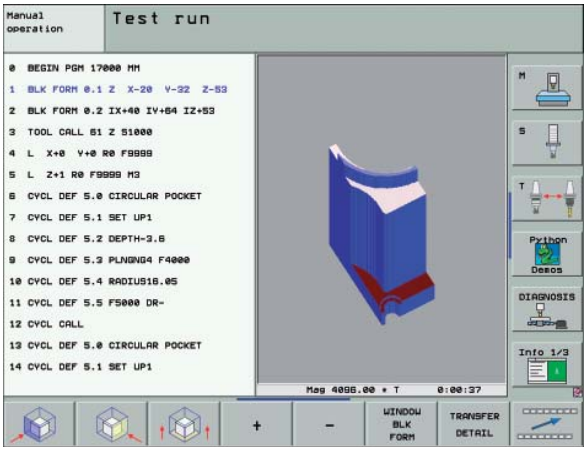
- In a plan view with different shades of depth
- In three planes (as in the workpiece drawing)
- In a solid model, 3-D view

Details can be displayed in magnification. The high resolution of the 3-D view visualizes even very fine contours true to detail and enables you to see even hidden details clearly and reliably. A simulated light source provides realistic light-and-shadow conditions.

When testing complex five-axis programs, even operations with tilted planes or multi-side machining can be displayed. In addition, the iTNC 530 indicates the calculated machining time in hours, minutes and seconds.

Program-run graphics
On the iTNC 530, you can run the programming graphics or verification graphics even while the workpiece is being machined. Also, it shows a real-time graphic of the machining progress during program run. Coolant spray and protective enclosures usually obstruct any direct view of the actual workpiece. You can get around this with a simple keystroke to see the simulated progress of workpiece machining.





Programming in the Workshop
– Straightforward Function Keys for Complex Contours

Programming 2-D-Contours

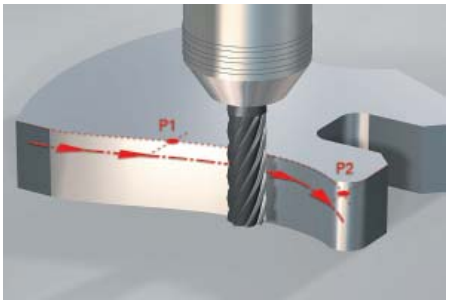
2-D contours are the daily bread of the modern machine shop. The iTNC 530 offers a variety of possibilities here.

Programming with path function keys
If contours are dimensioned for NC, which means that the end points are specified in Cartesian or polar coordinates, then you can program them directly with the path function keys.

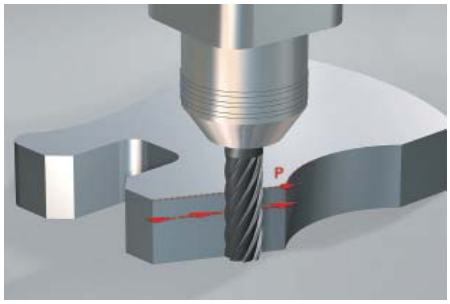
Straight and circular contour elements
To program a line segment, for example, simply press the key for linear traverse. The iTNC 530 asks for all information required for a complete programming block, such as target coordinates, feed rate, cutter radius compensation and machine functions. Appropriate path function keys for circular movement, chamfers, and corner rounding simplify your programming. To avoid surface blemishes during approach or departure from the contour, it must be approached smoothly—that is, tangentially.

You simply specify the starting or end point of the contour and the approaching or departing radius of the cutter edge—the control does the rest for you.

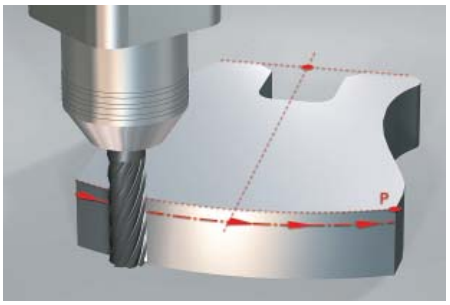
The iTNC 530 can look ahead over a radius-compensated contour for up to 99 blocks to watch for back cutting and avoid contour damage such as can occur when roughing a contour with a large tool.



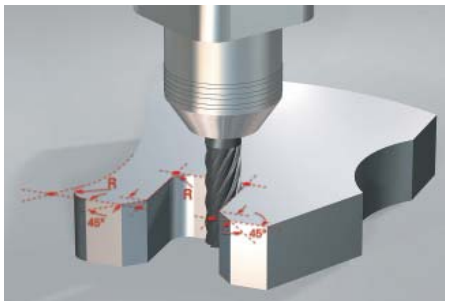
CT
Circular path defined by its end point, with a smooth (tangential) departure from the previous contour element



L
Straight line defined by its end point

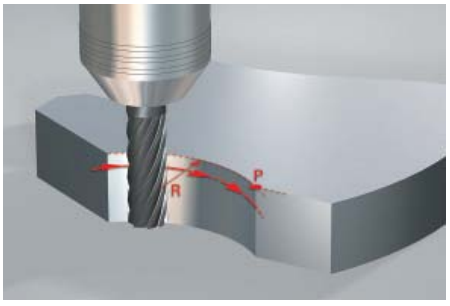


CC C
Circular path defined by its center point, end point, and rotational direction



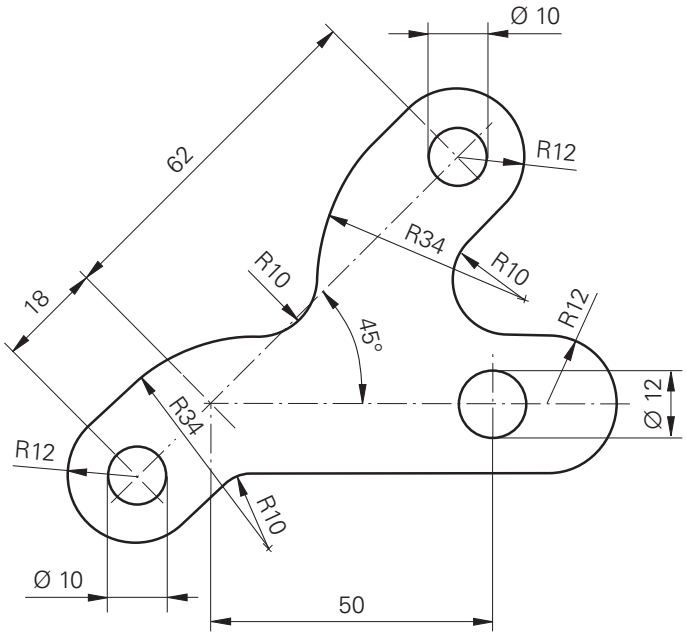
RND Rounding: circular path defined by radius and corner point, with a smooth (tangential) transition to its adjoining contour elements

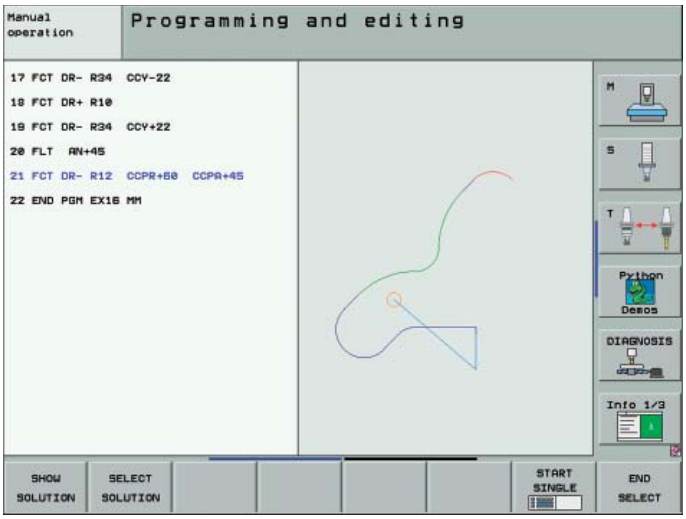
CHF Chamfer: defined by the corner point and the chamfer length



CR
Circular path defined by its radius, end point and rotational direction

FK free contour programming
Not all workpieces are dimensioned for conventional NC programming. Thanks to FK, the iTNC's free contour programming feature, in such cases you simply type in the known data—without first having to convert or calculate your data! It does not matter if individual contour elements are not completely defined as long as the complete contour has been. If the given data result in more than one mathematical solution, the helpful iTNC 530 programming graphics present the possible variants for your selection.





Programming in the Workshop
– Field-Proven Cycles for Recurring Operations

Comprehensive Fixed Cycles for Milling, Drilling and Boring

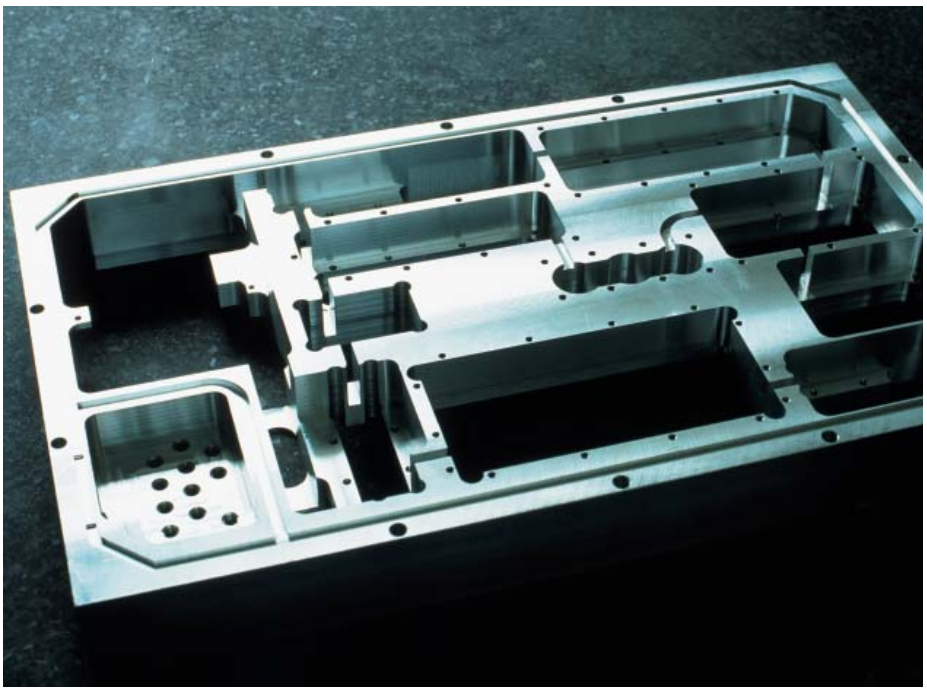
Frequently recurring operations that comprise several working steps are stored in the iTNC 530 as cycles. You program them under conversational guidance and supported by graphics that clearly illustrate the required input parameters.

Standard cycles
Besides the fixed cycles for drilling, tapping (with or without floating tap holder), thread milling, reaming and boring, there are cycles for hole patterns and milling cycles for clearing plane surfaces, and for roughing and finishing pockets, slots and studs.

Cycles for complex contours
SL cycles (SL = Subcontour List) are particularly helpful for clearing pockets with combined contours. This term is used to identify machining cycles for pilot drilling, roughing and finishing when the contour or subcontours are specified in subroutines. In this way, one contour description can be used for more than one operation using different tools.

Up to twelve **subcontours** can be superimposed for machining. The control automatically calculates the resulting contour and the tool paths for roughing or clearing the surfaces. Subcontours can be pockets or islands. Different components are combined to form a single pocket in which the tool circumnavigates the islands.

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.



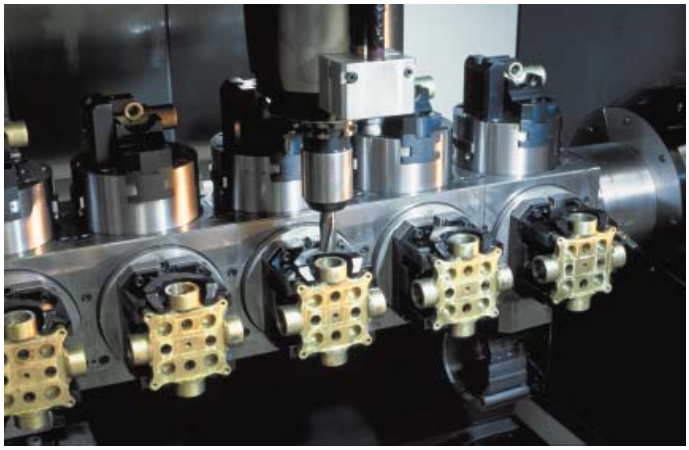
The iTNC 530 maintains a **finishing allowance** on the wall and floor surfaces during roughing. When **roughing** with different tools, the control identifies material remaining in inside corners so that it can be cleared later with smaller tools. A separate cycle is used for milling to the finished dimension.

You can also program **"open" contours** with the SL cycles. This enables the iTNC 530 to observe allowances for 2-D contours, to move the tool in alternating directions after each infeed, to avoid contour damage at undercuts and to maintain the defined milling direction (climb or conventional) after coordinate transformations such as mirroring.

OEM cycles
As original equipment manufacturers (OEMs), machine tool builders can contribute their special manufacturing know-how by designing additional fixed cycles and saving them in the iTNC 530. However, the end user can write his own cycles as well. HEIDENHAIN makes this possible with its PC program CycleDesign. CycleDesign enables you to organize the input parameters and soft-key structure of the iTNC 530 to suit your own needs.

3-D machining with parametric programming
With parameter functions you can program simple 3-D geometric figures that can easily be described mathematically. Here you can use the basic arithmetical operations, trigonometric functions, roots, powers, logarithmic functions, parentheses, and logical comparisons with conditional jump instructions. Parametric programming also offers you a simple method of realizing 3-D operations for which there are no standard cycles. Of course, parametric programming is also suited for **2-D contours** that cannot be described with line segments or circular arcs, but rather through mathematical functions.

Coordinate transformation
If you should need a contour that has already been programmed at another position or in a different size, the iTNC 530 offers you a simple solution:
Coordinate transformation
With coordinate transformation you can, for example, **rotate or mirror** the coordinate system or **shift the datum.** With a **scaling factor** you can enlarge or reduce contours to respect shrinkage allowance or oversizes.



Well Thought Out, Simple and Flexible
– smarT.NC — The Alternative Operating Mode

TNC controls from HEIDENHAIN have always been user friendly: Thanks to their simple programming in HEIDENHAIN conversational language, field-proven cycles, unambiguous function keys, and clear and vivid graphic functions, they have grown over the last 30 years to become one of the most popular shop-floor programmable controls.

The alternative smarT.NC operating mode makes programming even easier. With the well thought-out input forms, you can create your NC program in a few quick steps. Of course you'll be aided by help graphics. As always, HEIDENHAIN has also placed great value on compatibility. You can switch at any time between smarT.NC and conversational programming. But not only can you write programs with smarT.NC—you can test and run them as well.

Programming made simple
With smarT.NC, you program with the aid of easy-to-use, unambiguous forms. For simple operations, you need only enter a few machining data. With smarT.NC, you define such a machining step simply and quickly in a single overview window.

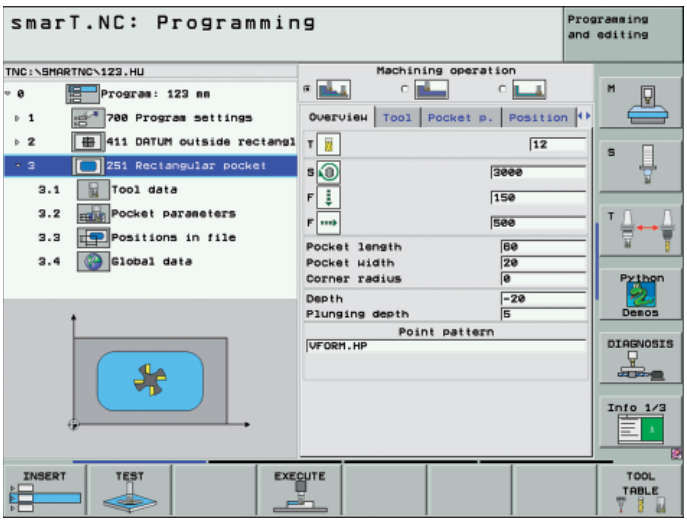
Of course, if required, you can define additional machining options. These options are available in subforms in which, with a few additional keystrokes, you can enter the parameters for machining options. You can define other functions, such as measuring cycles, in separate forms.

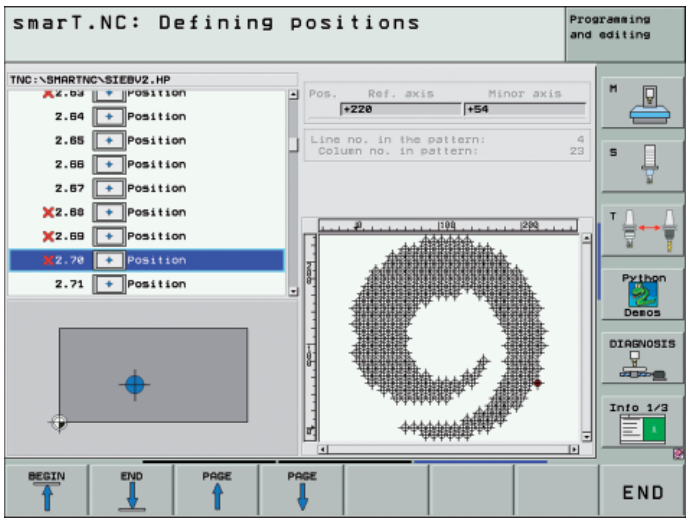
Stay simple and flexible when programming machining patterns
Machining positions are often arranged in patterns on the workpiece. With the pattern generator in smarT.NC, you can program very diverse machining patterns simply and extremely flexibly—of course with graphic support.

You can define as many point patterns as desired with various numbers of points in one file. smarT.NC displays the point patterns in a tree structure.

smarT.NC can even handle irregular patterns in the tree structure by allowing you to simply hide or delete any machining positions of a regular pattern.

If necessary, you can even edit the coordinates of the workpiece surface in individual machining patterns.





Programming contours
You define contours in the same way as you do part programs—using forms with graphic guidance. The individual contour elements are likewise displayed in the outline; the associated data are shown in a form. The TNC saves the contour itself in a separate file as a plain-language program so that you can reuse the contour later individually for various operations.

If a workpiece is not dimensioned for conventional NC programming, smarT.NC also features the powerful HEIDENHAIN FK free contour programming.

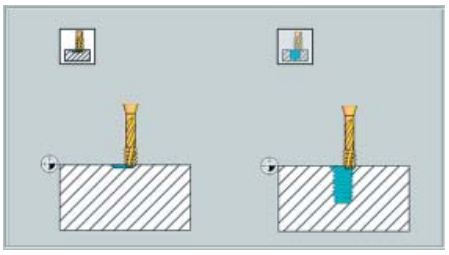
Well-designed for fast operation
With the split-screen, smarT.NC provides an **easily understandable program structure.** On the left screen, you can navigate quickly in a variable tree structure. On the right, clearly arranged input forms immediately show you the defined machining parameters. The soft-key row shows the input options.

smarT.NC means **reduced input:** You can enter global program parameters such as setup clearances, position feed rates, etc. once at the beginning of the program to avoid multiple definitions.

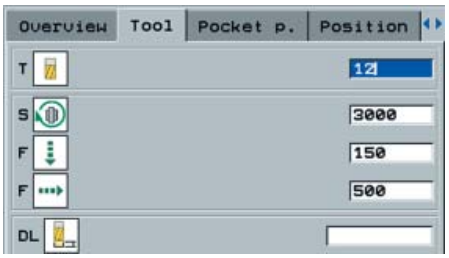
smarT.NC allows **fast editing:** With the new navigation keys, you can quickly reach any machining parameter in an input form. With a separate key you can switch directly between the form views.

Superior graphic support
Even as a CNC beginner, you can program with smarT.NC quickly and without extensive schooling. smarT.NC gives you optimal support.

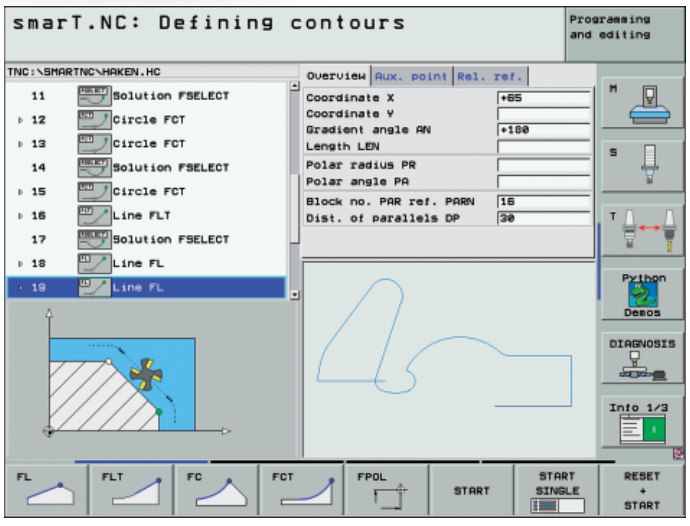
Clearly arranged **help graphics** illustrate all required input.



Graphic symbols increase concept recognition when the same type of input is required for different operations.



Tooltips appear from under the mouse pointer to help you along.



Open for Communication
– The iTNC 530 Understands DXF Files (Option)

Why program complex contours when your drawing is already in DXF format anyway? As of NC software 340 49x-02 you can open DXF files directly on the iTNC 530 to extract contours. Not only does this save time otherwise spent on programming and testing, but you can also be sure that the finished contour is exactly according to the designer's specifications.

The DXF format—particularly the DXF format supported by the iTNC 530—is very widespread, and is supported by all common CAD and graphics programs.

After the DXF file has been loaded onto the iTNC from the network or a USB stick, you can open the file just like an NC program in the iTNC's file manager. Meanwhile the iTNC considers the operating mode in which you started the DXF converter and generates either a contour program for smarT.NC or a program in conversational format.

As a rule, DXF files contain multiple layers, with which the designer organizes the drawing. 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. The iTNC can select a contour train even it has been saved in **different layers.**

The iTNC also supports you when **defining the workpiece preset.** 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.

You can define the following locations as reference point:

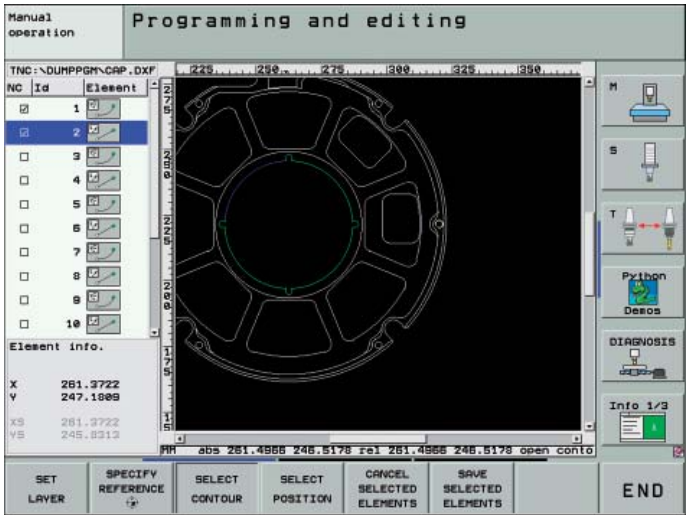
- The beginning, end or mid-point of a line
- At the beginning, end or center point of a circular arc
- Quadrant transitions or center point of a circle
- Intersection of two lines, regardless of whether it is located inside or outside the programmed segments
- Intersection of a line and a circular arc
- Intersection of a line and a circle

If multiple intersections can result between two elements (e.g., between a straight line and a circle), you can select the correct intersection with a mouse click.

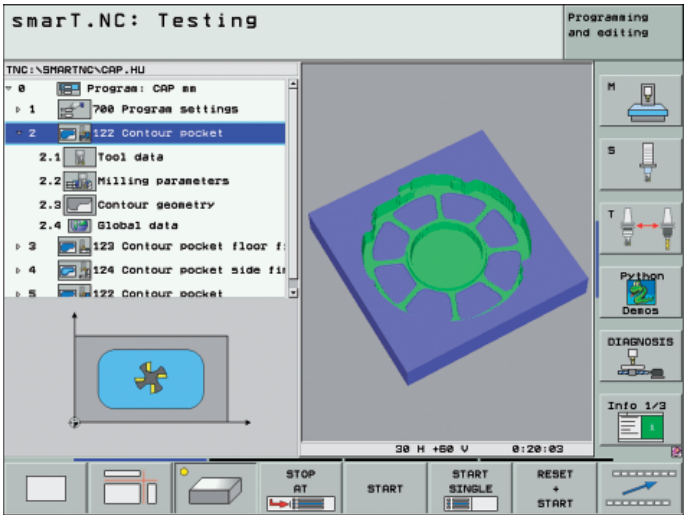
Contour selection is exceptionally user friendly. You select any element by clicking it with the mouse. As soon as you select a second element, the iTNC detects your desired direction of machining, and starts the **automatic contour detection.** The iTNC automatically selects all clearly identifiable contour elements until the contour closes or branches out. There you click the immediately following contour element. In this way you can define even extensive contours with just a few mouse clicks. If desired you can also shorten, lengthen or interrupt the contour elements.

But you can also select **machining positions** and save them as point files, particularly to use drilling positions or starting points for pocket machining. You enjoy the same choices as when defining the reference-point.

A 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 uploaded in case you want to use it on older TNC controls, or a transition tolerance if occasionally the elements do not quite adjoin.



Zoom in to details of an imported DXF file



Part program on the basis of the imported DXF file

Open for Communication
– Program Offline and Enjoy the Advantages of iTNC

Frequently, 3-D programs are written on offline CAD/CAM systems and then transferred to the control over a data line. Here, too, iTNC 530 performance comes to the fore. The **fast data transfer** over the Ethernet interface functions safely and reliably, even with large 3-D programs. You can make full use of the iTNC 530's **ease of use**—even for offline programming.

Programs created offline
NC programs for five-axis operations are normally written on CAD/CAM systems. The workpiece geometry is described in the CAD system, while the required technology data is added in the CAM system. The technology data specify the methods (such as milling, drilling, or boring), the strategy (area pocket clearance, plunge milling, etc.), and the parameters (spindle speed, feed rate, etc.) for machining the workpiece. A postprocessor uses the geometry and technology data to create an executable program, which is then usually transferred over the company network to the iTNC 530.

In principle, postprocessors generate two types of NC programs, both of which can be run by the iTNC 530:

- Machine-specific NC programs take the respective machine configuration into account and include all coordinates of the NC axes on that machine.
- Machine-neutral NC programs define the contour and use vectors to define the respective tool position on the contour. The iTNC 530 then uses the information to calculate the axis positions of the actual machine axes. The key benefit here is that you can run such NC programs on various machines with differing axis configurations.

The postprocessor is the link between the CAM system and the CNC control. Standard features on all prevalent CAM systems include post-processors for DIN/ISO format as well as for the proven and user-friendly HEIDENHAIN conversational format. This enables you to use **special TNC functions** that are available only in conversational format. These are for example:

- TCPM function
- Program structuring
- Cutting data calculation through tables
- Special Q-parameter functions

Program optimization is also easy. As usual, the conversational programming is graphically supported. And of course you can use all proven **setup functions** of the iTNC 530 to quickly and economically locate the workpiece.

CAD/CAM systems don't always generate programs optimized for the machining process. Therefore, the iTNC 530 offers a point filter for smoothing externally created NC programs. The filter function creates a copy of the original program, and then adds any points required by the parameters that you set. This smoothes the contour to allow the program to run more quickly and with less jerk.

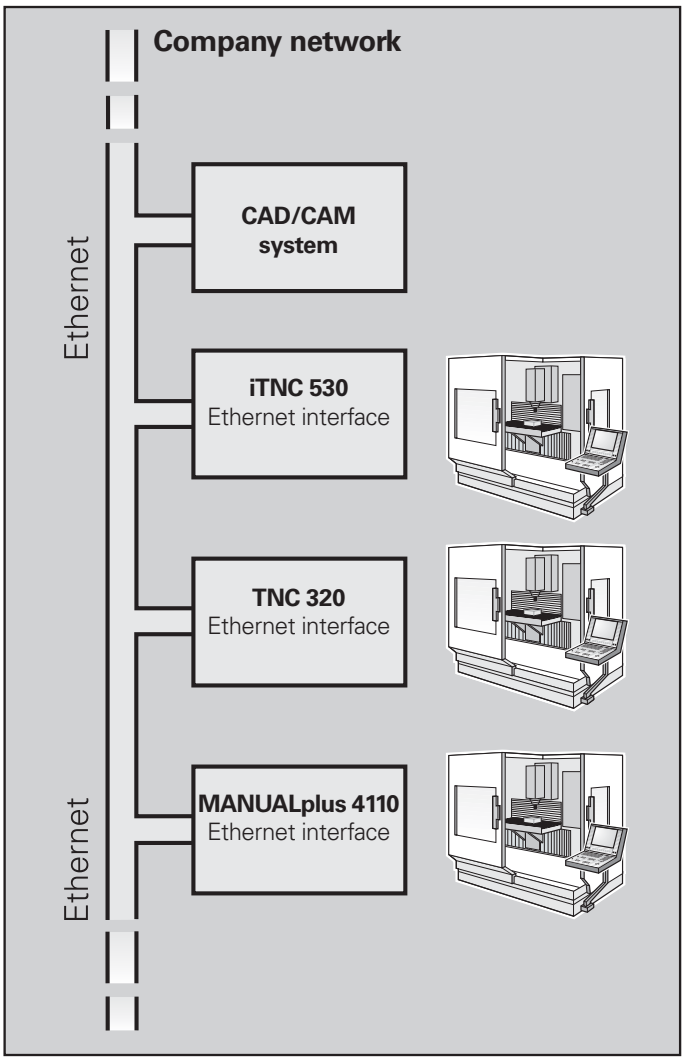
– Fast Data Transfer with the iTNC

The networked iTNC 530
The iTNC 530 can be integrated into networks and connected with PCs, programming stations and other data storage devices. Even in its standard version, iTNC 530 features a latest-generation Fast Ethernet interface in addition to its RS-232-C/V.24 and RS-422/V.11 data interfaces. The iTNC 530 communicates with NFS servers and Windows networks in TCP/IP protocol without needing additional software. The fast data transfer at rates of up to 100 Mbps guarantees very short transfer times even of comprehensive 3-D programs with ten thousands of blocks.

The transmitted programs are saved on the iTNC's hard disk and are run from it at high speed. In this way you can already begin machining while the data is still being transferred.

With the aid of the PC program TNCremoNT from HEIDENHAIN and an Ethernet or other data interface you can
- transfer remotely stored part programs and tool or pallet tables bidirectionally,
- start the machine,
- make backups of the hard disk,
- and query the operating status of the machine.

TNCremoNT uses the LSV2 protocol to operate the iTNC 530 remotely.



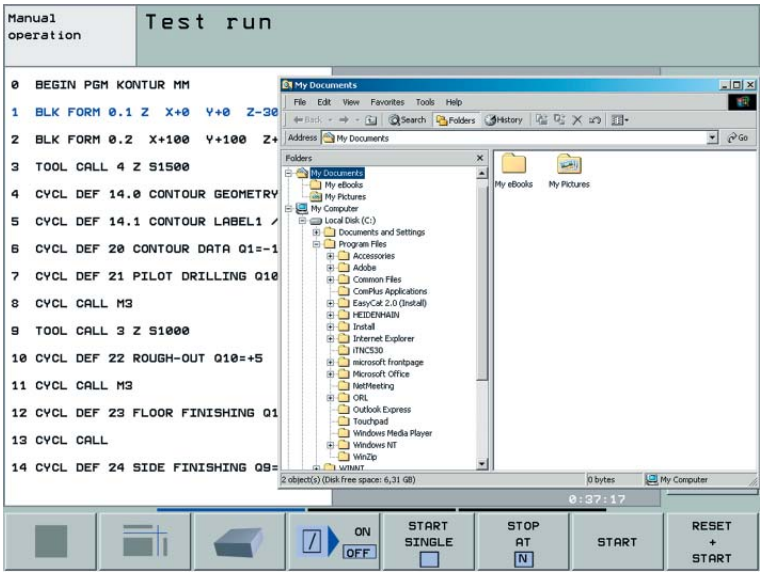
Open for Communication
– The iTNC 530 with Windows XP

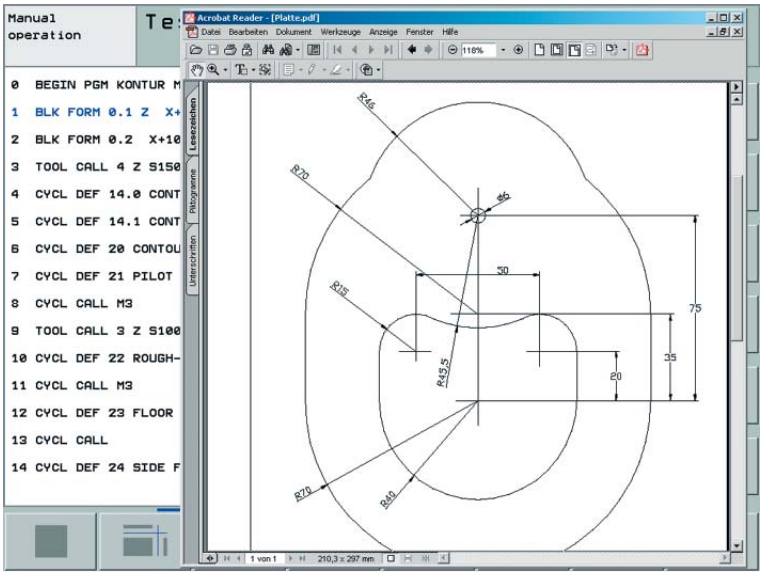
Windows applications on the iTNC 530
The iTNC 530 hardware option with two processors additionally features the Windows XP operating system as a user interface, enabling the use of standard Windows applications. One processor attends to the real-time tasks and the HEIDENHAIN operating system, while the second processor stays available exclusively for the standard Windows operating system, giving the user access to the world of information technology.

What are the benefits of this technology?
Firmly integrated in the company network, the iTNC 530 provides the technician with all relevant information: CAD drawings, tooling sketches, tool lists and other data. It is also possible to access Windows-based tool databases where the machine operator can very quickly find tool data such as cutting speeds or permissible plunge angles. You spare yourself the time otherwise lost on printing and distributing production documents.

Also, machine data and production data are easily acquired with the iTNC 530 and your Windows applications. In this way you always have control over your productivity.

As a rule, any additional windows applications are installed by the machine tool builder, who then tests the function of the entire system. If you intend to install software yourself, please consult first with your machine tool builder. Faulty installation or unsuitable software can impair the proper function of the machine.





– The iTNC Programming Station

Why a programming station?
It's well known that it is easy to create part programs on iTNCs at the machine, even while another part is being machined. Nevertheless, it can often happen that short reloading times and other machining tasks hinder any prolonged or concentrated programming work. With the iTNC programming station you have the capability to program just as you do at the machine, but away from the noise and distractions of the shop floor.

Creating programs
Programming, testing and optimizing your smarT.NC, HEIDENHAIN conversational or DIN/ISO programs for the iTNC 530 with the programming station substantially reduces machine idle times. You need not adjust your way of thinking—every keystroke fits. On the programming station you program on the same keyboard as at the machine.

Testing programs created offline
Of course you can also test programs that were written on a CAD/CAM system. The high-resolution program verification graphics help you even with complex 3-D programs to easily spot contour damage and hidden details.

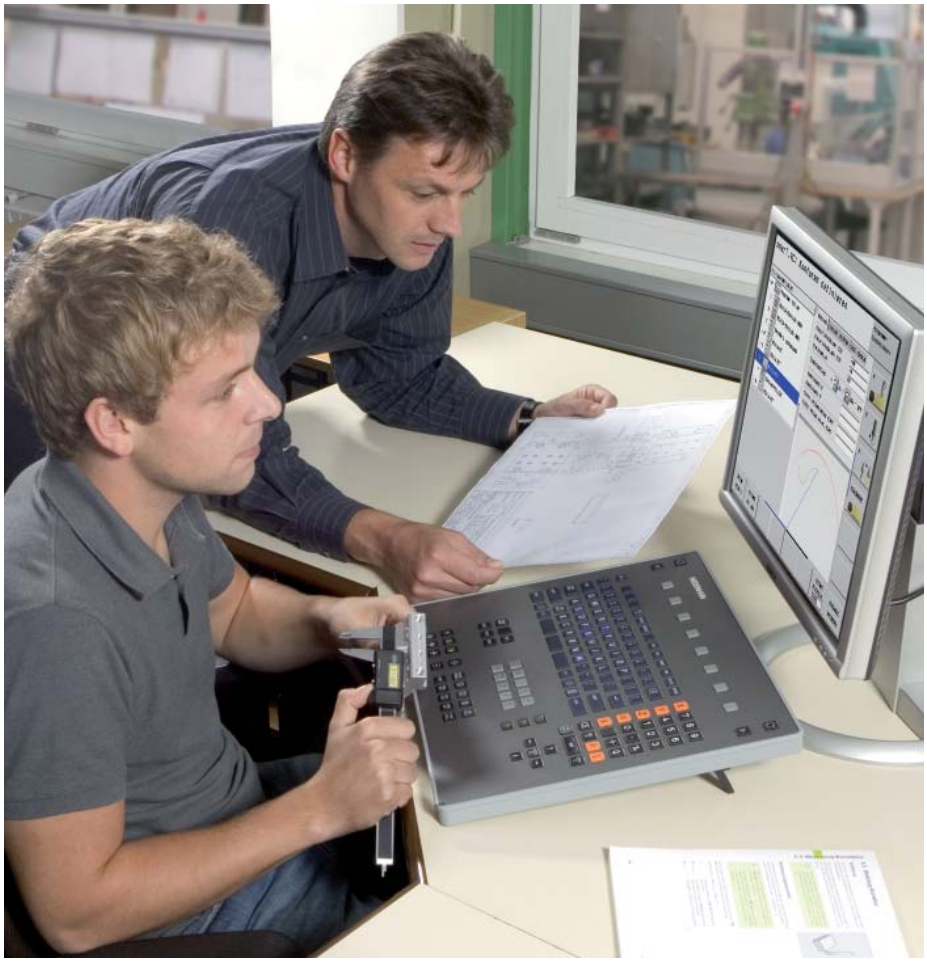
Training with the iTNC programming station
Because the iTNC programming station is based on the same software as the iTNC 530, it is ideally suited for apprentice and advanced training. The program is entered on the original iTNC keyboard unit. Even the program verification functions exactly as it does on the machine. This gives the trainee the experience needed to enable him to safely operate the machine later.

Because it can be programmed with smarT.NC, in plain language, and in DIN/ISO, the iTNC programming station can also be used in schools for TNC programming training.

More information about the programming station and a free demo version are available on the Internet at *www.heidenhain.de.* Or simply ask for the *iTNC Programming Station* CD or brochure.

The workstation
The iTNC programming station software runs on a PC. The programming station is only slightly different from an iTNC built onto a machine tool. The familiar TNC keyboard remains unchanged except that it now includes the soft keys, which are otherwise integrated in the visual display unit. You connect the iTNC keyboard to your PC's USB port. The PC screen shows the familiar TNC user interface.

Or as an alternative, you can even operate the programming station without an iTNC keyboard. You can use a virtual keyboard instead—it is displayed together with the iTNC Control Panel and features the most important dialog initiation keys of the iTNC.



Workpiece Measurement
– Setup, Presetting and Measuring with Touch Trigger Probes

Workpiece touch probes from HEIDENHAIN help you to reduce costs in the workshop and in series production: Together with the iTNC 530, touch probes can automatically perform setup, measuring and inspection functions.

The stylus of a TS three-dimensional touch trigger probe is deflected upon contact with a workpiece surface. At that moment the TS generates a trigger signal that, depending on the model, is transmitted either by cable or over an infrared beam to the control.

The 3-D touch probes* are inserted directly into the machine tool spindle. They can be equipped with various shanks. The ruby ball tips are available in several diameters, and the styli in different lengths.

* The touch probes must be interfaced to the iTNC 530 by the machine tool builder.

Touch probes with **cable connection for signal transmission** for machines with manual tool change:
TS 220 – TTL version
TS 230 – HTL version

Touch probes with **infrared signal transmission** for machines with automatic tool change:
TS 440 – Compact dimensions
TS 444 – Compact dimensions, battery-free power supply through integrated air turbine generator over central compressed air supply
TS 640 – Standard touch probe with wide-range infrared transmission
TS 740 – High probing accuracy and repeatability, low probing force

SE 640

TS 640

TS 440

TS 220

More information about workpiece touch probes is available on the Internet at www.heidenhain.de or in the brochure or CD *Touch Probes*.

Tool Measurement
– Measuring Length, Radius and Wear Directly in the Machine

The tool is of course a definitive factor in ensuring a consistently high level of production quality. This means that an exact measurement of the tool dimensions and periodic control of the tool for wear and breakage, as well as the shape of each tooth, are necessary. HEIDENHAIN offers the TT 140 trigger tool touch probe as well as the non-contacting TL Nano and TL Micro laser systems for tool measurement.

The systems are installed directly in the machine's workspace, where they permit tool measurement either before machining or during interruptions.

The **TT 140 tool touch probe** captures the tool length and radius. When probing the tool, either while rotating or at standstill (such as for measuring individual teeth), the contact plate is deflected and a trigger signal is transmitted to the iTNC 530.

The **TL Nano** and **TL Micro laser systems** are available for various maximum tool diameters. Using a laser beam, they probe the tool without contact, and can detect form deviations of individual teeth along with the tool length and radius.

TT 140

TL Micro

More information about tool touch probes is available on the Internet at www.heidenhain.de or in the brochure or CD *Touch Probes*.

Inspecting and Optimizing Machine Accuracy
– Calibrating Rotary Axes with KinematicsOpt (Option)

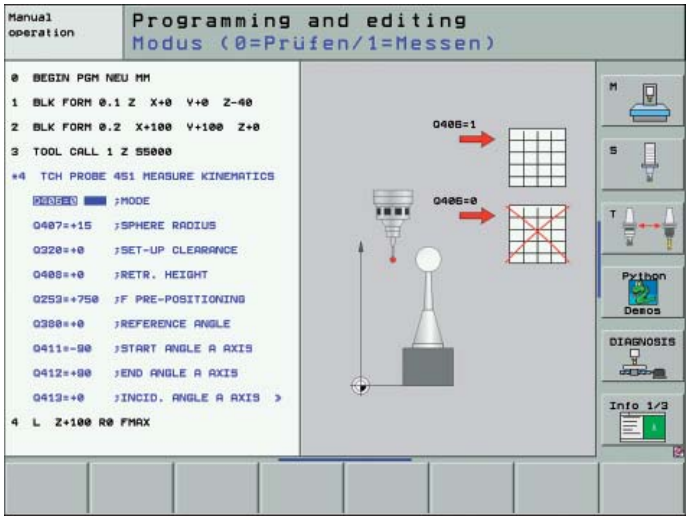
Accuracy requirements are becoming increasingly stringent, particularly in the area of 5-axis machining. Complex parts are required to be manufactured with precision and reproducible accuracy even over long periods.

The new TNC function **KinematicsOpt** is an important component to help you meet these high requirements: With a HEIDENHAIN touch probe inserted, a 3-D-touch probe cycle measures your machine's rotary axes fully automatically. The results of measurement are the same regardless of whether the axis is a rotary table, a tilting table or a swivel head.

To measure the rotary axes, a calibration ball is fixed at any position on the machine table and probed with the HEIDENHAIN touch probe. But first you define the resolution of the measurement and define for each rotary axis the area that you want to measure.

From the measured values, the TNC calculates the statistical tilting accuracy. The software minimizes the spatial error arising from the tilting movements and, at the end of the measurement process, automatically saves the machine geometry in the respective machine constants of the kinematic table.

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



Positioning with the Electronic Handwheel
– Delicate Axis Traverse

You can move machine axes controlled by the iTNC 530 by simply pressing the axis keys. A simpler and more sensitive way, however, is to use the electronic handwheels from HEIDENHAIN.

You can move the axis slide through the feed motors in direct relation to the rotation of the handwheel. For delicate operations you can set the transmission ratio to certain preset distances per handwheel revolution.

HR 130 and HR 150 panel-mounted handwheels
The panel-mounted handwheels from HEIDENHAIN can be integrated in the machine operating panel or mounted at another location on the machine. An adapter permits connection of up to three HR 150 electronic handwheels.

HR 410 and HR 420 portable handwheels
If you need to get a closer look at the workpiece in the machine's working envelope, then ideally you should use the HR 410 or HR 420 portable handwheels. The axis keys and certain functional keys are integrated in the housing. This way you can switch axes and set up the machine at any time—regardless of where you happen to be standing.

The following functions are available:

HR 410
- Traverse direction keys
- Three keys with preset feed rates for latched traverse
- Actual-position-capture key
- Three keys for machine functions to be defined by the machine tool builder
- Permissive buttons
- Emergency stop button

HR 420 with display
- Traverse distance per revolution can be set
- Display for operating mode, actual position value, programmed feed rate and spindle speed, error messages
- Override potentiometer for feed rate and spindle speed
- Axis selection via keys and soft keys
- Keys for continuous traverse of the axes
- Emergency stop button
- Actual-to-nominal value transfer
- NC start/stop
- Spindle on/off
- Soft keys for machine functions defined by the machine tool builder

If you no longer need the handwheel, just attach it to the machine somewhere by its built-in magnets.

HR 420

HR 410

... and If There's a Problem?
– Diagnostics for HEIDENHAIN Controls

In recent years, the operating reliability of machine tools and controls has improved continually. However, interruptions or problems can still occur. Often it is simply a programming or parameter problem. Here is where the true advantage of remote diagnosis comes into play: The service technician communicates online with the control over modem, ISDN or DSL, analyzes the control, and repairs it immediately.

HEIDENHAIN offers the **TeleService** and **TNCdiag** PC software for remote diagnosis. They enable an extensive search for problems on the control as well as in the inverter system, and even in the motors themselves.
TeleService also features comprehensive remote operation and remote monitoring of the control.*

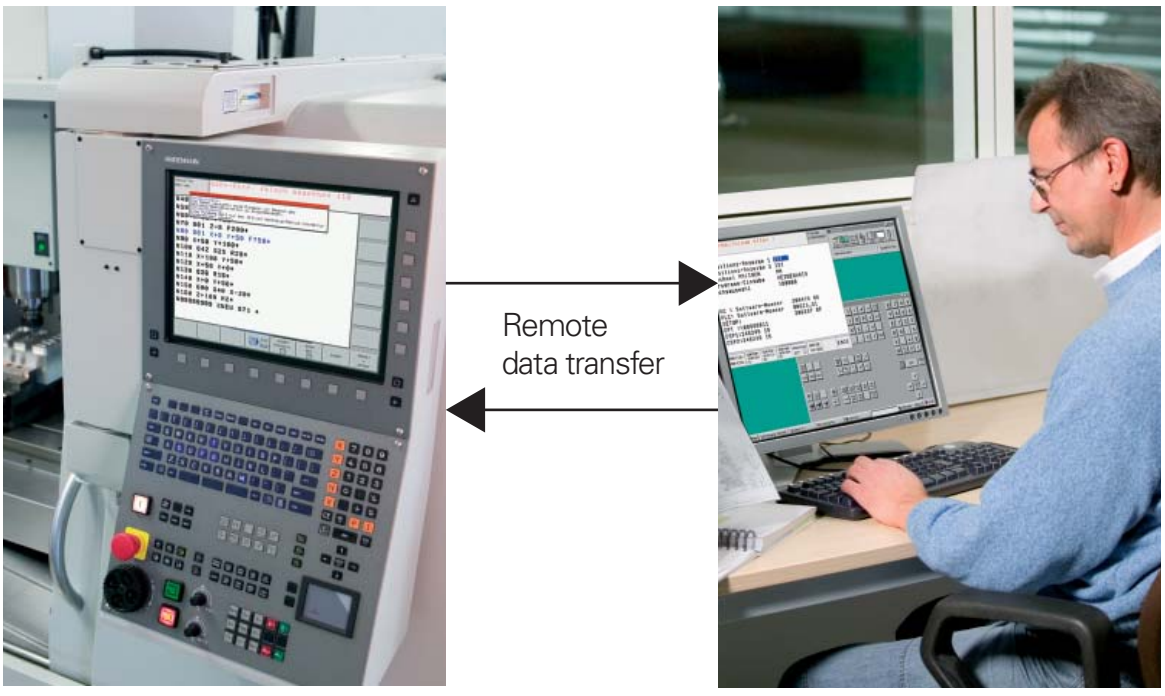
* The iTNC must be prepared by the machine tool builder for this function.

Remote diagnosis with TeleService
The TeleService PC software from HEIDENHAIN enables the machine tool builder to provide quick and simple remote diagnosis and programming support for the iTNC 530 controls.

The TeleService software is also of interest to you, the NC user: Installing it on a network PC enables remote operation and remote monitoring of the iTNC 530 connected to the network.

Drive diagnosis with TNCdiag
TNCdiag enables the service technician to quickly and easily troubleshoot the drives. With its dynamic display of status signals, you can even examine ambient conditions that lead to errors.

Please contact your machine manufacturer to enable TNCdiag on your control.



Overview
– User Functions

User functions	Standard	Option	FCL	
Brief description	• •	 ○ 0-7 		Basic version: 3 axes plus spindle 4th NC axis plus auxiliary axis or 8 additional NC axes or 7 additional axes plus 2nd spindle Digital current and spindle speed control
Program entry	•	 42		With smarT.NC, in HEIDENHAIN conversational format and according to DIN/ISO Direct loading of contours or machining positions from DXF files and saving as smarT.NC or conversational contouring programs, or as point tables
Program optimization			02	Point filter for smoothing externally created NC programs
Position data coordinates	• • • •			Nominal positions for lines and arcs in Cartesian coordinates or polar coordinates Incremental or absolute dimensions Display and input in mm or inches Display of the handwheel path during machining with handwheel superimposition
Tool compensation	• • •			Tool radius in the working plane and tool length Radius-compensated contour look-ahead for up to 99 blocks (M120) Three-dimensional tool-radius compensation for changing tool data without having to recalculate an existing program
Tool tables	•			Multiple tool tables with any number of tools
Cutting data	• • •			Cutting data tables for automatic calculation of spindle speeds and feed rates from tool-specific data (cutting speed, feed per tooth) Cutting speed as alternate entry to the spindle shaft speed Feed rate can also be entered as F_Z (feed per tooth) or F_U (feed per revolution)
Constant contour speed	• •			Relative to the path of the tool center Relative to the tool's cutting edge
Parallel operation	•			Creating programs with graphic support while another program is being run
3-D machining		9 9 9 9 9 9 9	 02	Particularly jerk-free path control 3-D compensation through surface normal vectors Tool Center Point Management (TCPM): Using the electronic handwheel to change the angle of the swivel head during program run without affecting the position of the tool point Tool perpendicular to contour Tool radius compensation normal to the tool direction Spline interpolation Manual traverse in the active tool-axis system
Machining with rotary tables		8 8		Programming of cylindrical contours as if in two axes Feed rate in length per minute
Adaptive Feed Control		45		AFC: Adaptive feed control adjusts the contouring feed rate to the current spindle power
Collision monitoring		40	 04	DCM: Dynamic Collision Monitoring (only with MC 422 B/C) Graphic depiction of the active collision objects

Overview
– User Functions (Continued)

User functions	Standard	Option	FCL	
Contour elements	• • • • • • •			Straight line Chamfer Circular path Circle center Circle radius Tangentially connecting circular arc Corner rounding
Approaching and departing the contour	• •			Via straight line: tangential or perpendicular over a circle
FK free contour programming	•			FK free contour programming in HEIDENHAIN conversational format with graphic support for workpiece drawings not dimensioned for NC
Program jumps	• • •			Subroutines Program section repeats Calling any program as subroutine
Fixed cycles	• • • • • • • • •			Cycles for boring, and conventional and rigid tapping Drilling cycles for pecking, reaming, boring, counterboring, centering Cycles for milling internal and external threads Multioperation machining of rectangular and circular pockets Cycles for clearing level and inclined surfaces Multioperation machining of straight and circular slots Linear and circular point patterns Contour train, contour pocket—also with contour-parallel machining OEM cycles (special cycles developed by the machine tool builder) can be integrated
Coordinate transformation	•	8		Datum shift, rotation, mirror image, scaling factor (axis-specific) Tilting the working plane, PLANE function
Q parameters Programming with variables	• • • • • •			Mathematical functions =, +, –, *, /, sin α, cos α, tan α, arc sin, arc cos, arc tan, a^n, e^n, ln, log, $\sqrt{a}$, $\sqrt{a^2 + b^2}$ Logical operations (=, = /, <, >) Parentheses Absolute value of a number, constant π, negation, truncation of digits before or after the decimal point Functions for calculation of circles Functions for text processing
Programming aids	• • • • • •		03	Calculator Complete list of all current error messages Context-sensitive help function for error messages TNCguide: The integrated help system. User information available directly on the iTNC 530 (only with at least 256 MB RAM) Graphical support for programming cycles Comment and structure blocks in the NC program
Actual position capture	•			Actual positions can be transferred directly into the NC program
Program verification graphics Display modes	• • •			Graphic simulation before a program run, even while another program is running Plan view / projection in 3 planes / 3-D view, also in tilted working plane Detail enlargement

User functions	Standard	Option	FCL	
3-D line graphics			02	For verification of programs created offline
Programming graphics	•			In the Programming and Editing mode, the contour of the NC blocks is drawn on screen while the blocks are being entered (2-D pencil-trace graphics), even while another program is running
Program-run graphics Display modes	• •			Graphic simulation during real-time machining Plan view / projection in 3 planes / 3-D view
Machining time	• •			Calculation of machining time in the Test Run operating mode Display of the current machining time in the Program Run operating mode
Returning to the contour	• •			Mid-program startup in any block in the program, returning the tool to the calculated nominal position to continue machining. The graphic support in smarT.NC also lets you return to a point pattern. Program interruption, contour departure and reapproach
Preset tables	•			One preset table per traverse range for storing reference points
Datum tables	•			Several datum tables for storing workpiece-related datums
Pallet tables	•			Pallet tables (with as many entries as desired for the selection of pallets, NC programs and datums) can be machined workpiece by workpiece or tool by tool
Touch probe cycles	• • • •	 48	 02 03	Touch probe calibration Compensation of workpiece misalignment, manual or automatic Datum setting, manual or automatic Automatic tool and workpiece measurement Global setting of touch-probe parameters Probing cycle for three-dimensional measurements. Results of measurement shown as desired in the coordinate system of the tool or the machine Automatic measurement and optimization of the machine kinematics
Conversational languages	•	 41		English, German, Czech, French, Italian, Spanish, Portuguese, Swedish, Danish, Finnish, Dutch, Polish, Hungarian, Russian (Cyrillic), Chinese (traditional, simplified) For more conversational languages, see *Options*

Overview
– Accessories

Accessories	
Electronic handwheels	• One **HR 410** portable handwheel or • One **HR 420** portable handwheel with display or • One **HR 130** panel-mounted handwheel or • Up to three **HR 150** panel-mounted handwheels via the HRA 110 handwheel adapter
Workpiece measurement	• **TS 220** 3-D touch trigger probe with cable connection or • **TS 440** 3-D touch trigger probe with infrared connection or • **TS 444** 3-D touch trigger probe with infrared connection or • **TS 640** 3-D touch trigger probe with infrared connection or • **TS 740** 3-D touch trigger probe with infrared connection
Tool measurement	• **TT 140** triggering 3-D touch probe • **TL Nano** laser system for contact-free workpiece measurement or • **TL Micro** laser system for contact-free workpiece measurement
Programming station	Control software for PCs for programming, archiving, and training • Single-station license with original control keyboard • Single-station license with virtual keyboard • Network license with virtual keyboard • Demo version (operated via PC keyboard—free of charge)
Software for PCs	• **TeleService:** Software for remote diagnosis, monitoring, and operation • **TNCdiag:** Software for fast and easy fault diagnosis • **CycleDesign:** Software for creating your own cycle structure • **TNCremoNT:** Software for data transfer—free of charge

– Options

Option number	Option	As of NC software 34049x-	ID	Comment
0 **1** **2** **3** **4** **5** **6** **7**	Additional axis	01	354540-01 353904-01 353905-01 367867-01 367868-01 370291-01 370292-01 370293-01	Additional control loops 1 to 8
8	Software option 1 (for MC 420)	01	367591-01	**Machining with a rotary table** • Programming of cylindrical contours as if in two axes • Feed rate in length per minute **Coordinate transformation** • Tilting the working plane, PLANE function **Interpolation** • Circle in 3 axes with tilted working plane
9	Software option 2 (for MC 420)	01	367590-01	**3-D machining** • Particularly jerk-free path control • 3-D compensation through surface normal vectors • Tool Center Point Management (TCPM): Using the electronic handwheel to change the angle of the swivel head during program run without affecting the position of the tool point • Tool perpendicular to contour • Tool radius compensation normal to the tool direction • Manual traverse in the active tool-axis system **Interpolation** • Line in 5 axes (subject to export permit) • Spline: execution of splines (3rd degree polynomials) **Block processing time** 0.5 ms
18	HEIDENHAIN DNC	01	526451-01	Communication with external PC applications over COM component
40	DCM Collision	02	526452-01	Dynamic Collision Monitoring DCM (only with MC 422 B, MC 422 C)
41	Additional language	 02 03 03 03 03 03 04 04	 530184-01 530184-02 530184-03 530184-04 530184-06 530184-07 530184-08 530184-09	Additional conversational language: Slovenian Slovak Latvian Norwegian Korean[1)] Estonian Turkish Romanian
42	DXF Converter	02	526450-01	Load and convert DXF contours
44	Global PGM Settings	03	576057-01	Global program settings
45	AFC Adaptive Feed Control	03	579648-01	Adaptive Feed Control
46	Python OEM Process	04	579650-01	Python application on the iTNC[2)]
48	KinematicsOpt	04	630916-01	Touch probe cycles for automatic measurement of rotary axes
53	Feature content level	02	529969-01	Level of features

[1)] Only with at least 256 MB RAM
[2)] Only with at least 512 MB RAM

Overview
– Function Upgrades

As of NC software 340 49x-02, error fixes and software improvements were separated from each other. An update of NC software will predominantly 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 in function** will be offered as "feature upgrades," and are enabled via the Feature Content Level option.

If, for example, a control is updated from 340 49x-01 to 340 49x-02, the functions identified with "FCL 02" in the following tables are only available if the **Feature Content Level** is set from 01 to 02.

Of course, the current feature content level also includes the upgrade functions of the previous NC software levels.

All of the **options** included in the respective NC software can be purchased, no matter which feature content level you have.

Mode of operation	FCL	Description
Miscellaneous	02	USB support for peripheral memory devices (memory sticks, hard disks, CD-ROMs)
	02	DHCP (Dynamic Host Configuration Protocol) and DNS (Domain Name Server) possible for network settings
	03	TNCguide: The integrated help system. User information available directly on the iTNC 530 (only with at least 256 MB RAM)
	04	Graphic display of machine kinematics in the Program Run modes of operation[1)]
	04	3-D basic rotation: aligning workpieces in three dimensions[1)]
smarT.NC	02	Cycles for coordinate transformation introduced
	02	PLANE function introduced
	02	Contour pocket: Separate depth can be assigned for each subcontour
	02	Block-scan with graphic support
	03	smarT.NC editor in the Programming and Editing operating mode
	03	Machining a contour pocket on a point pattern
	03	Individually definable positioning heights in point patterns
	03	Touch probe cycles 408 and 409 for setting datums in the centerline of a slot or ridge
	03	Setting of probing parameters in a separate UNIT 441
	03	Automatic feed rate reduction in contour pockets during full tool engagement
Conversational programming	02	Cycle 441 for global setting of touch-probe parameters
	02	Point filter for smoothing externally created NC programs
	02	3-D line graphics for verification of programs created offline
	02	Manual traverse in the active tool-axis system
	03	Touch probe cycles 408 and 409 for setting datums in the centerline of a slot or ridge
	03	Probing cycle for three-dimensional measurements. Results of measurement shown as desired in the coordinate system of the tool or the machine
	03	Automatic feed rate reduction in contour pockets during full tool engagement

[1)] The function must be adapted by the machine tool builder

– Specifications

Specifications	Standard	Option	Windows XP Option	
Components	• • • •		○	MC 422 C and MC 420 main computers MC 422 C main computer with 2 processors CC 422 and CC 424 B controller units TE 530 B, TE 520 B and TE 535 operating panels BF 150 15.1-inch color TFT flat-panel display with soft keys
Operating system	•		○	HEROS real-time operating system for machine control Windows XP PC operating system as user interface
Memory	• •		○	RAM memory: 512 MB for control applications 512 MB for Windows applications Hard disk with at least 13 GB program memory
Input resolution and display step	• •			Linear axes: up to 0.1 µm Up to 0.0001°
Input range	•			Maximum 99999.999 mm (3937 inches) or 99999.999°
Interpolation	• • •	9 8 9		Linear in 4 axes Linear in 5 axes (subject to export permit) Circular in 2 axes Circular in 3 axes with tilted working plane Helical: superimposition of circular and straight paths Spline: execution of splines (3rd degree polynomials)
Block processing time	•	9		3.6 ms (3-D straight line without radius compensation) Option: 0.5 ms
Axis feedback control	• • • •			Position loop resolution: Signal period of the position encoder/1 024 Cycle time of position controller: 1.8 ms Cycle time of speed controller: 600 µs Cycle time of current controller: minimum 100 µs
Range of traverse	•			Maximum 100 m (3937 inch)
Spindle speed	•			Maximum 60000 rpm (with 2 pole pairs)
Error compensation	• •			Linear and nonlinear axis error, backlash, reversal spikes during circular movements, thermal expansion Stick-slip friction
Data interfaces	• • • •	 18		One RS-232-C/V.24 and one RS-422/V.11, each with max. 115 Kbps Extended data interface with LSV2 protocol for remote operation of the iTNC 530 over the data interface with the HEIDENHAIN software TNCremoNT 100BaseT fast Ethernet interface 2 x USB HEIDENHAIN-DNC for communication between a Windows application and iTNC (DCOM interface)
Diagnostics	•			Fast and simple troubleshooting through integrated diagnostic aids
Ambient temperature	• •			Operation: 0 °C to +45 °C Storage: –30 °C to +70 °C

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☎ +49 (8669) 31-0
FAX +49 (8669) 5061
E-Mail: info@heidenhain.de

www.heidenhain.de

DE **HEIDENHAIN Technisches Büro Nord**
12681 Berlin, Deutschland
☎ (030) 54705-240
E-Mail: tbn@heidenhain.de

HEIDENHAIN Technisches Büro Mitte
08468 Heinsdorfergrund, Deutschland
☎ (03765) 69544
E-Mail: tbm@heidenhain.de

HEIDENHAIN Technisches Büro West
44379 Dortmund, Deutschland
☎ (0231) 618083-0
E-Mail: tbw@heidenhain.de

HEIDENHAIN Technisches Büro Südwest
70771 Leinfelden-Echterdingen, Deutschland
☎ (0711) 993395-0
E-Mail: tbsw@heidenhain.de

HEIDENHAIN Technisches Büro Südost
83301 Traunreut, Deutschland
☎ (08669) 31-1345
E-Mail: tbso@heidenhain.de

AR **NAKASE SRL.**
B1653AOX Villa Ballester, Argentina
☎ +54 (11) 47684242
E-Mail: nakase@nakase.com

AT **HEIDENHAIN Techn. Büro Österreich**
83301 Traunreut, Germany
☎ +49 (8669) 31-1337
E-Mail: tba@heidenhain.de

AU **FCR Motion Technology Pty. Ltd**
Laverton North 3026, Australia
☎ +61 (3) 93626800
E-Mail: vicsales@fcrmotion.com

BE **HEIDENHAIN NV/SA**
1760 Roosdaal, Belgium
☎ +32 (54) 343158
E-Mail: sales@heidenhain.be

BG **ESD Bulgaria Ltd.**
Sofia 1172, Bulgaria
☎ +359 (2) 9632949
E-Mail: info@esd.bg

BR **DIADUR Indústria e Comércio Ltda.**
04763-070 – São Paulo – SP, Brazil
☎ +55 (11) 5696-6777
E-Mail: diadur@diadur.com.br

BY Belarus → **RU**

CA **HEIDENHAIN CORPORATION**
Mississauga, Ontario L5T 2N2, Canada
☎ +1 (905) 670-8900
E-Mail: info@heidenhain.com

CH **HEIDENHAIN (SCHWEIZ) AG**
8603 Schwerzenbach, Switzerland
☎ +41 (44) 8062727
E-Mail: verkauf@heidenhain.ch

CN **DR. JOHANNES HEIDENHAIN (CHINA) Co., Ltd.**
Beijing 101312, China
☎ +86 10-80420000
E-Mail: sales@heidenhain.com.cn

CS Serbia and Montenegro → **BG**

CZ **HEIDENHAIN s.r.o.**
106 00 Praha 10, Czech Republic
☎ +420 272658131
E-Mail: heidenhain@heidenhain.cz

DK **TP TEKNIK A/S**
2670 Greve, Denmark
☎ +45 (70) 100966
E-Mail: tp-gruppen@tp-gruppen.dk

ES **FARRESA ELECTRONICA S.A.**
08028 Barcelona, Spain
☎ +34 934092491
E-Mail: farresa@farresa.es

FI **HEIDENHAIN Scandinavia AB**
02770 Espoo, Finland
☎ +358 (9) 8676476
E-Mail: info@heidenhain.fi

FR **HEIDENHAIN FRANCE sarl**
92310 Sèvres, France
☎ +33 01 41 143000
E-Mail: info@heidenhain.fr

GB **HEIDENHAIN (G.B.) Limited**
Burgess Hill RH15 9RD, United Kingdom
☎ +44 (1444) 247711
E-Mail: sales@heidenhain.co.uk

GR **MB Milionis Vassilis**
17341 Athens, Greece
☎ +30 (210) 9336607
E-Mail: bmilioni@otenet.gr

HK **HEIDENHAIN LTD**
Kowloon, Hong Kong
☎ +852 27591920
E-Mail: service@heidenhain.com.hk

HR Croatia → **SL**

HU **HEIDENHAIN Kereskedelmi Képviselet**
1239 Budapest, Hungary
☎ +36 (1) 4210952
E-Mail: info@heidenhain.hu

ID **PT Servitama Era Toolsindo**
Jakarta 13930, Indonesia
☎ +62 (21) 46834111
E-Mail: ptset@group.gts.co.id

IL **NEUMO VARGUS MARKETING LTD.**
Tel Aviv 61570, Israel
☎ +972 (3) 5373275
E-Mail: neumo@neumo-vargus.co.il

IN **ASHOK & LAL**
Chennai – 600 030, India
☎ +91 (44) 26151289
E-Mail: ashoklal@satyam.net.in

IT **HEIDENHAIN ITALIANA S.r.l.**
20128 Milano, Italy
☎ +39 0227075 1
E-Mail: info@heidenhain.it

JP **HEIDENHAIN K.K.**
Tokyo 102-0073, Japan
☎ +81 (3) 3234-7781
E-Mail: sales@heidenhain.co.jp

KR **HEIDENHAIN LTD.**
Suwon, South Korea, 443-810
☎ +82 (31) 2011511
E-Mail: info@heidenhain.co.kr

MK Macedonia → **BG**

MX **HEIDENHAIN CORPORATION MEXICO**
20235 Aguascalientes, Ags., Mexico
☎ +52 (449) 9130870
E-Mail: info@heidenhain.com

MY **ISOSERVE Sdn. Bhd**
56100 Kuala Lumpur, Malaysia
☎ +60 (3) 91320685
E-Mail: isoserve@po.jaring.my

NL **HEIDENHAIN NEDERLAND B.V.**
6716 BM Ede, Netherlands
☎ +31 (318) 581800
E-Mail: verkoop@heidenhain.nl

NO **HEIDENHAIN Scandinavia AB**
7300 Orkanger, Norway
☎ +47 72480048
E-Mail: info@heidenhain.no

PH **Machinebanks` Corporation**
Quezon City, Philippines 1113
☎ +63 (2) 7113751
E-Mail: info@machinebanks.com

PL **APS**
02-489 Warszawa, Poland
☎ +48 228639737
E-Mail: aps@apserwis.com.pl

PT **FARRESA ELECTRÓNICA, LDA.**
4470 - 177 Maia, Portugal
☎ +351 229478140
E-Mail: fep@farresa.pt

RO Romania → **HU**

RU **OOO HEIDENHAIN**
125315 Moscow, Russia
☎ +7 (495) 931-9646
E-Mail: info@heidenhain.ru

SE **HEIDENHAIN Scandinavia AB**
12739 Skärholmen, Sweden
☎ +46 (8) 53193350
E-Mail: sales@heidenhain.se

SG **HEIDENHAIN PACIFIC PTE LTD.**
Singapore 408593,
☎ +65 6749-3238
E-Mail: info@heidenhain.com.sg

SK Slovakia → **CZ**

SL **Posredništvo HEIDENHAIN SAŠO HÜBL s.p.**
2000 Maribor, Slovenia
☎ +386 (2) 4297216
E-Mail: hubl@siol.net

TH **HEIDENHAIN (THAILAND) LTD**
Bangkok 10250, Thailand
☎ +66 (2) 398-4147-8
E-Mail: info@heidenhain.co.th

TR **T&M Mühendislik San. ve Tic. LTD. ŞTİ.**
34738 Erenköy-Istanbul, Turkey
☎ +90 (216) 3022345
E-Mail: info@tmmuhendislik.com.tr

TW **HEIDENHAIN Co., Ltd.**
Taichung 407, Taiwan
☎ +886 (4) 23588977
E-Mail: info@heidenhain.com.tw

UA Ukraine → **RU**

US **HEIDENHAIN CORPORATION**
Schaumburg, IL 60173-5337, USA
☎ +1 (847) 490-1191
E-Mail: info@heidenhain.com

VE **Maquinaria Diekmann S.A.**
Caracas, 1040-A, Venezuela
☎ +58 (212) 6325410
E-Mail: purchase@diekmann.com.ve

VN **AMS Advanced Manufacturing Solutions Pte Ltd**
HCM City, Viêt Nam
☎ +84 (8) 9123658 - 8352490
E-Mail: davidgoh@amsvn.com

ZA **MAFEMA SALES SERVICES C.C.**
Midrand 1685, South Africa
☎ +27 (11) 3144416
E-Mail: mailbox@mafema.co.za

Zum Abheften hier falzen! / Fold here for filing!

Vollständige Adressen siehe www.heidenhain.de
For complete addresses see www.heidenhain.de

363 807-27 · 50 · 3/2008 · F&W · Printed in Germany · Subject to change without notice