HEIDENHAIN

iTNC 530

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

HSCI

June 2016

Uniformly digital

For over 35 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. Now HEIDENHAIN is introducing the iTNC 530 as a uniformly digital numerical control.

In the **uniformly digital control solution** of the iTNC 530, all components are connected over purely digital interfaces: the control components over HSCI (HEIDENHAIN Serial Controller Interface), the HEIDENHAIN real-time protocol for Fast Ethernet and the encoders over EnDat 2.2, the bidirectional interface from HEIDENHAIN.

This achieves a high degree of availability for the entire system. It can be diagnosed and is immune to noise—for everything from the main computer to the encoder.

The uniform digital solution from HEIDENHAIN guarantees very high accuracy and surface definition together with high traversing speeds.
So don't be afraid of innovation: HEIDENHAIN controls are powerful, user-friendly, and upwardly compatible so they are **prepared for the future.**

This brochure describes the functions and specifications of the iTNC 530 with NC software 60642x-04.

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? Both are simple with the iTNC 530 as you'll be impressed by their **workshop-oriented programmability** just as by their **external program creation:** 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 Klartext—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 CAM system or at a HEIDENHAIN programming station. Its Ethernet interface guarantees very short transfer times, even of long programs.

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

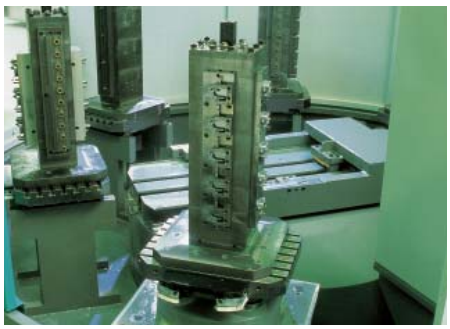
High speed milling
- Fast block processing
- Short control-loop cycle time
- Motion control with smoothed jerk
- High spindle speed
- Fast data transfer

Five-axis machining with swivel head and rotary table
- Offline programming away from the machine: the iTNC 530 automatically takes the machine geometry into account
- Tilting the working plane
- Cylinder surface machining
- Tool Center Point Management (TCPM)
- 3-D tool compensation
- Fast, accurate contour 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.

For 5-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

Boring mill
- 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
- Datum management
- 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

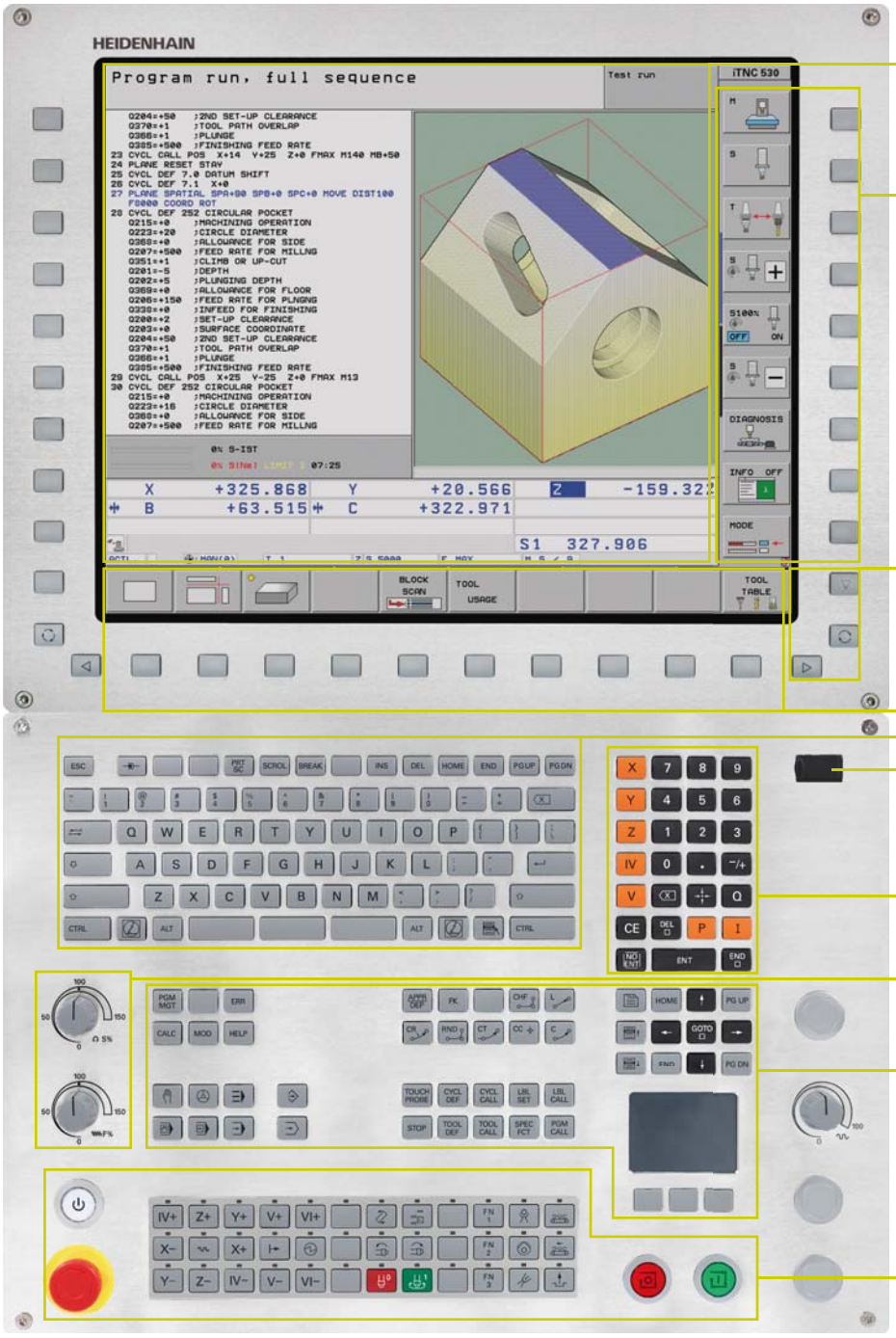
The screen
The large 19-inch TFT color flat-panel display shows a clear overview of all relevant information for programming, operating and inspecting the machine tool and control, such as program blocks, comments and error messages. More information is provided through graphic support during program 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 to keep you up to date 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 operating panel is oriented to the programming process. The well-thought-out arrangement of keys in a clear division into **function groups**, i.e. programming modes, machining modes, management/TNC functions and navigation, supports you during program input. Simple key assignment, easily understandable symbols or abbreviations clearly indicate each key's function.

The **alphabetic keypad** enables you to easily enter comments and G codes. The integrated **machine operating panel** features easily exchangeable snap-on keys that allow simple adaptation to the respective machine configuration. You use the **override potentiometers** to make delicate adjustments of feed-rate, rapid traverse and spindle speed. And the operating panel features a complete **set of PC keys** and a **touchpad** that can be used, for example, for operating the DXF converter.



The screen content includes two operating modes, the program, graphics and the machine status

PLC function keys (soft keys) for machine functions

Keys for **screen management** (screen layout), operating mode and switching between soft-key rows

Self-explanatory **function keys** (soft keys) for NC programming

Alphanumeric keyboard for comments or DIN/ISO programs and a **set of PC keys** for controlling the operating system functions.

USB port for additional data storage or pointing devices

Axis-selection keys and **numeric keypad**

Override potentiometers for feed rate, rapid traverse and spindle speed

Function keys for programming modes, machine modes, TNC functions, management and navigation

Machine operating panel with snap-on keys and LEDs

Ergonomic and elegant, state-of-the-art and field-proven—HEIDENHAIN controls in a new design. Judge for yourself:

Durable
The high-quality stainless steel design of the iTNC 530 features a special protection coating and is therefore highly resistant to soiling and wear.

Smooth
The rectangular, slightly rounded keys are pleasant to the touch and reliable in operation. Their inscriptions do not wear off, even under extreme workshop conditions.

Flexible
The integrated machine operating panel features easily exchangeable snap-on keys.

Reliable
The elevated key bed of the machine operating panel prevents accidental actuation. LEDs serve for status display of each key by clearly indicating the active machine functions.

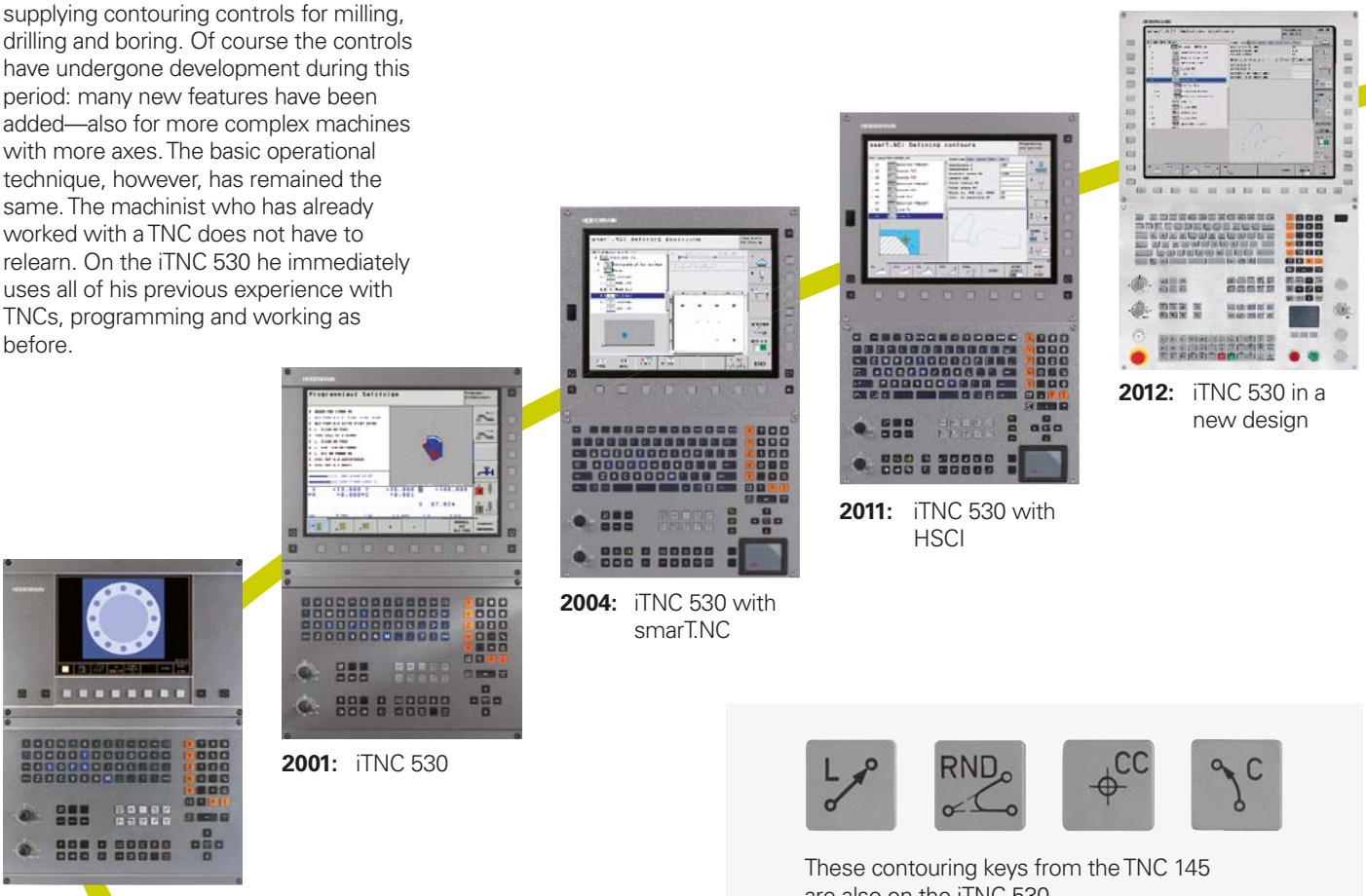
Versatile
Soft keys both for the programming and the machine functions always show only the currently available selections.

Sensitive
With the handy control knobs you can individually adjust the feed-rate, rapid traverse and spindle speed.

Communicative
The fast USB 2.0 interface lets you connect storage media or pointing devices to the keyboard simply and directly.

Consistently upward compatible
– A promising future with HEIDENHAIN contouring controls

For over 35 years, HEIDENHAIN has been supplying 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 already worked with a 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.



2012: iTNC 530 in a new design

2011: iTNC 530 with HSCI

2004: iTNC 530 with smarT.NC

2001: iTNC 530

1997: TNC 426 M
TNC 430

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

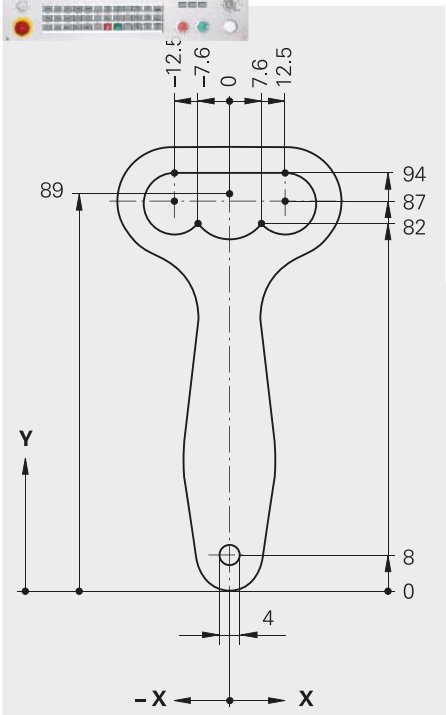
1993: TNC 426 C/P

1988: TNC 407
TNC 415

1987: TNC 355

1984: TNC 155

2012: TNC 640 for milling/turning machines



Inside contour—

"Old" programs also run on newer TNC controls

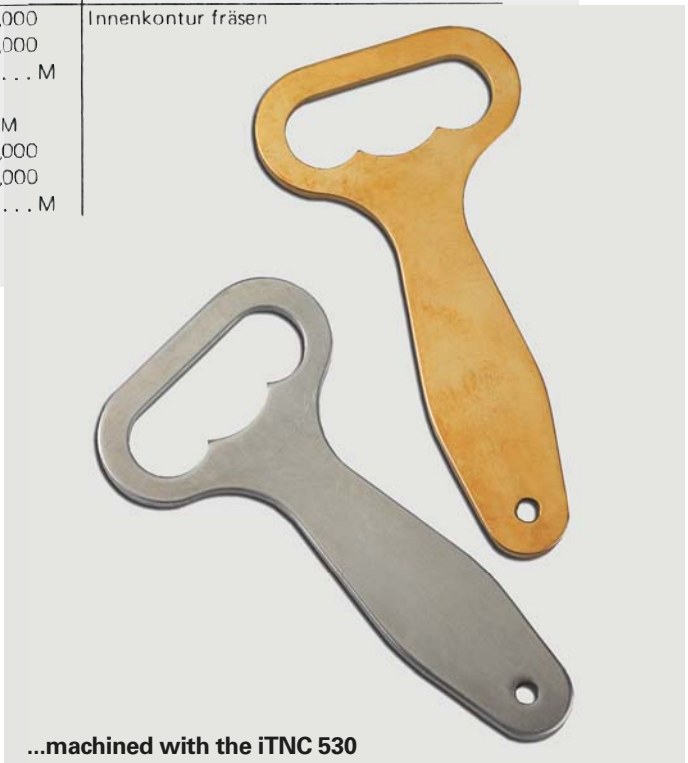
Part programs from your NC program archive that were written on older TNC contouring controls can also be run on an iTNC 530 with very little effort. This ensure the greatest possible flexibility in machine utilization and provides enormous benefits when you have to manufacture "old" parts again. HEIDENHAIN contouring controls put you in the position to manufacture a replacement part fast and economically even now—after more than 30 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 step up to a new control you need not relearn its operation and programming. Instead you only need to become familiar with those functions that are new. So you can apply your professional knowledge as a machinist immediately even on a new TNC control.

```
31  CC X+0,000      Y+8,000
32  C  X+0,000      Y+0,000
                DR+ RRF ... M98
-------------------------------------------------------------------
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

1983: TNC 150

1981: TNC 145, the first contouring control from HEIDENHAIN

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 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 motion control, its contour precalculation and its algorithms for jerk limitation, the iTNC 530 has the right features for the desired 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-smoothed 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 looks ahead and thinks with you. 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.

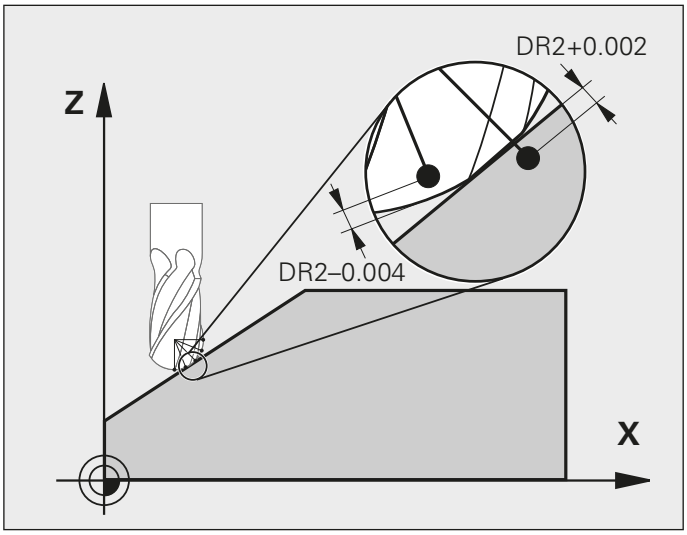
For NC programs with normal vectors, such as those generated by CAM systems, the iTNC 530 automatically calculates a 3-D tool compensation (option) selectably for end mills, ball-nose cutters, or toroid cutters.

– Compensating form errors of tools with 3D-ToolComp (option)

3D-ToolComp is a new and powerful option for three-dimensional tool radius compensation. A compensation-value table is used to define angle-dependent delta values that describe the tool deviation from the ideal circular form (see graphic).

The iTNC then corrects the radius value defined for the tool's current point of contact with the workpiece. In order to determine the point of contact exactly, the NC program must be have been created with surface-normal blocks (LN blocks) by a CAM system. The surface-normal blocks specify the theoretical center point of the radius cutter, and in some cases also the tool orientation relative to the workpiece surface.

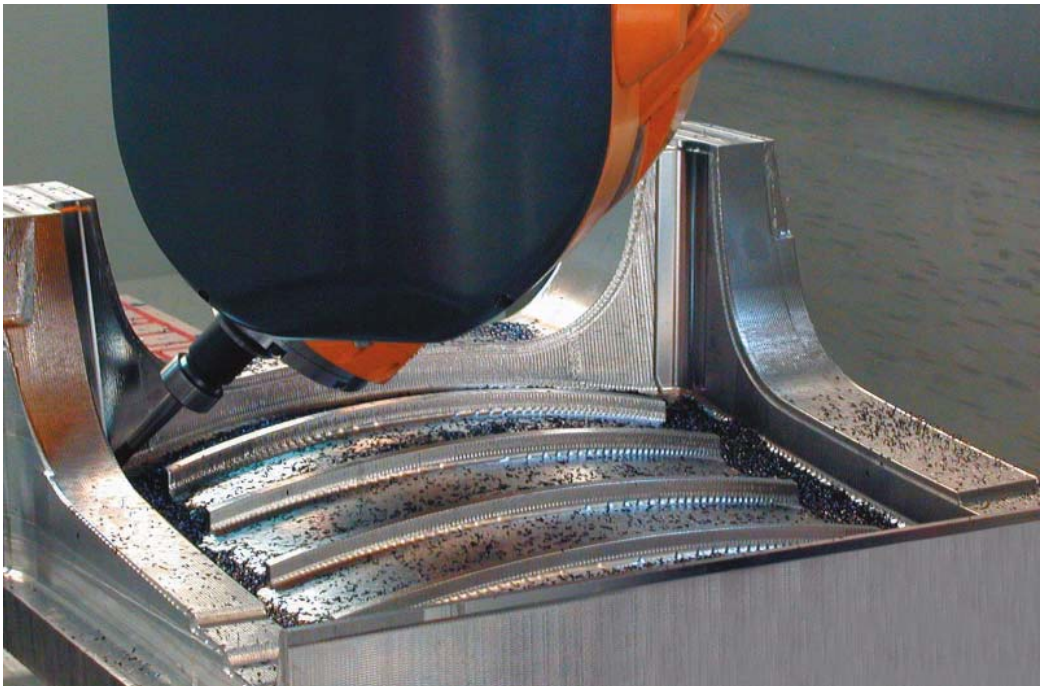
Ideally, the compensation-value table is generated fully automatically by way of a special cycle that uses a laser system to measure the form of the tool so that the iTNC can use this table directly. If the form errors of the tool used are available as a calibration chart from the tool manufacturer, then you can create the compensation-value table manually.



Machining with five axes
– Guided tool tip

CAM systems use postprocessors to generate 5-axis programs. In principle, such programs contain either all coordinates of the machine's existing NC axes, or NC blocks with surface normal vectors. During 5-axis machining with three linear axes and two additional tilting axes* the tool is always normal to the workpiece surface or is kept at a specific angle to it (inclined tool machining).

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** (TCPM) feature—an improvement upon the proven TNC function M128—provides optimal tool guidance and prevents contour gouging.



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

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

TCPM defines the **interpolation between the start and end positions:**

- During **face milling**—machining mainly with the 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**, the workpiece is machined 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.

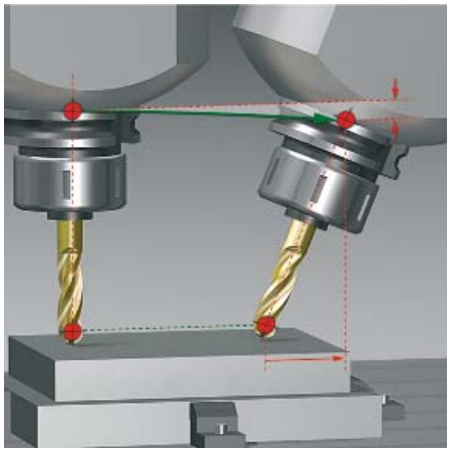
TCPM defines the **effect of the programmed feed rate** as desired as one of the following:

- 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.
- 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.

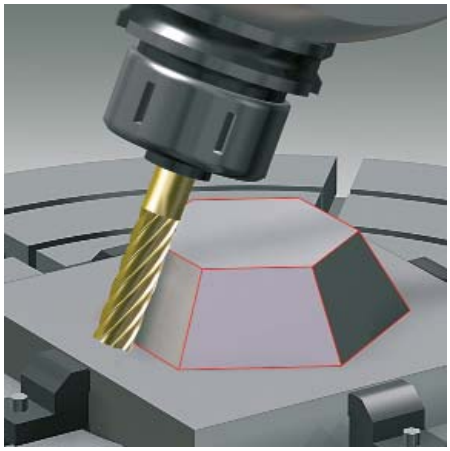
With TCPM you can also define the **effect of the inclination angle** for more uniform cutting passes when working with an inclined radius cutter:

- Angle of inclination defined as axis angle
- Angle of inclination defined as spatial angle

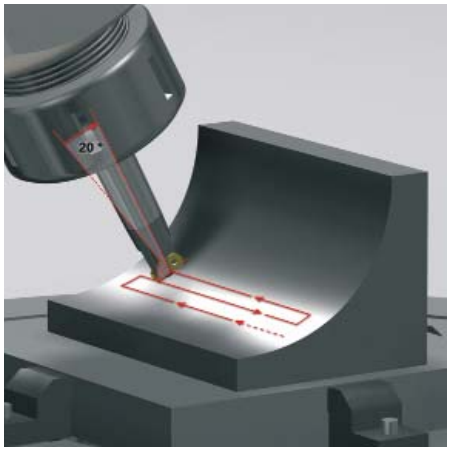
The iTNC takes the inclination angle into account in all 3-D machining—even with 45° swivel heads or tilting tables. You either specify the angle of inclination in the NC program via a miscellaneous function, or adjust it manually with an electronic handwheel. The iTNC 530 makes sure that the tool remains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



Superimposed compensation movement



Peripheral milling



Face milling

Machining with five axes
—Swivel head and rotary table controlled by iTNC

Many 5-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 CAM system.

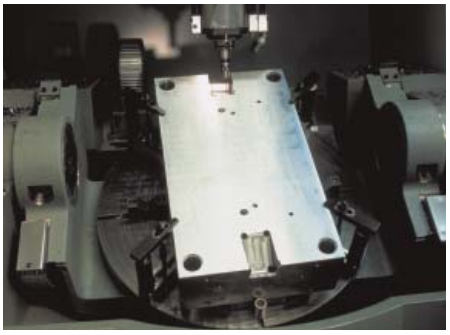
Tilting the working plane*
Programs for contours and holes on inclined surfaces are often very complex and require time-consuming computing and programming work. Here the iTNC 530 helps you to save a great deal of programming time.

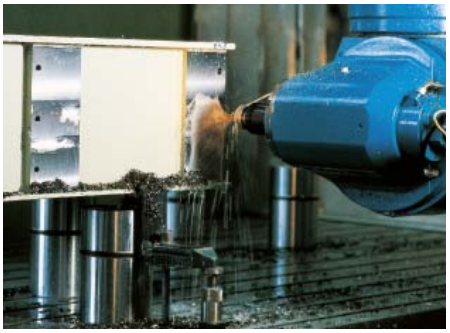
You program the machining operation as usual in the main plane, for example in X/Y. The machine runs the program in a plane that has been tilted by one or more rotary axes with respect to the main plane.

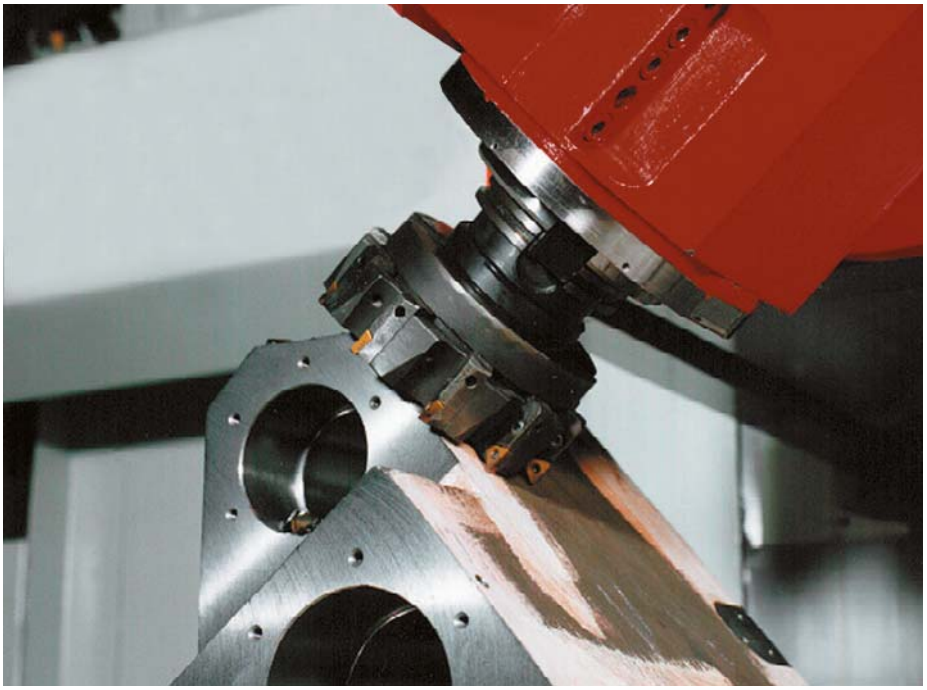
The PLANE feature makes it easy to define a tilted working plane: You can specify tilted working planes in seven different ways, depending on the information on the workpiece drawing. In order to keep the use of these complex functions as simple as possible, a separate animation is available for each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

You can also use the PLANE function to define the positioning behavior for tilting 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 in the machine and iTNC by the machine tool builder.







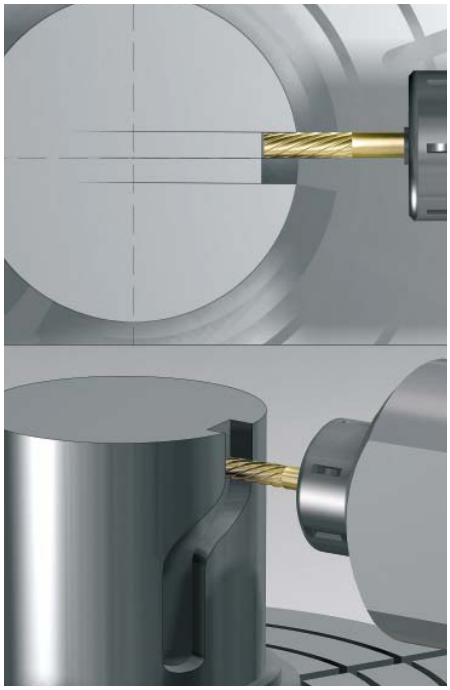
Machining cylindrical surfaces*
With the iTNC 530 it is quite easy to program contours (which consist of straight lines and arcs) on cylindrical surfaces using rotary and tilting tables: You simply program the contour in a plane as if the cylinder surface were unrolled. The iTNC 530 then executes the 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 in the machine and iTNC 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 move the tool in the current direction of the tool axis through an external direction key 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 5-axis machining program,
- use the handwheel or external direction keys to perform an operation in Manual mode with an inclined tool,
- move the tool with the handwheel in the active tool axis direction during machining.

Feed rate for rotary axes and tables in mm/min*
In the standard version, the feed rate of rotary axes is programmed in degrees/minutes. However, the iTNC 530 can interpret this feed rate in mm/min as well. 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 the normally high traversing speeds of 5-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.

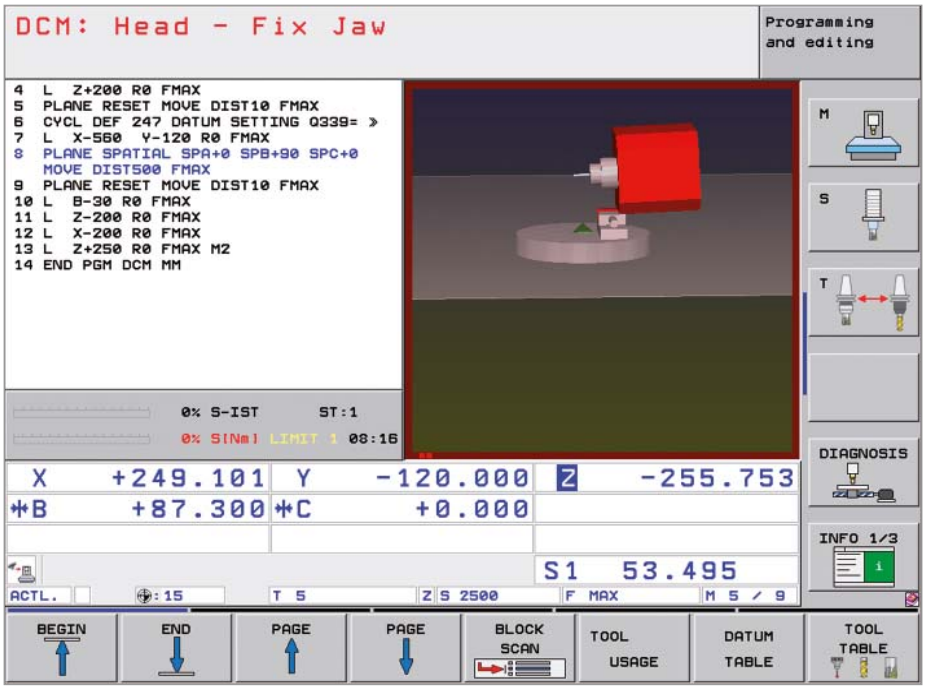
In cases such as these, the machine operator is supported by the **dynamic collision monitoring (DCM)*** feature of the iTNC 530. The control interrupts machining whenever a collision is imminent, thereby increasing the safety for the machine and its operator. This helps to prevent machine damage, which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

With NC functions you can activate or deactivate previously saved fixture situations in automatic operation. Specific fixtures can also be activated for each NC program from pallet tables. This increases safety and reliability in automatic production.

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 or fixture in the working space, the iTNC 530 detects it, stops axis movement, and issues an error message.

Before actually machining a part, you can also check for collisions in the Test Run mode, with a real datum and real tools.

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



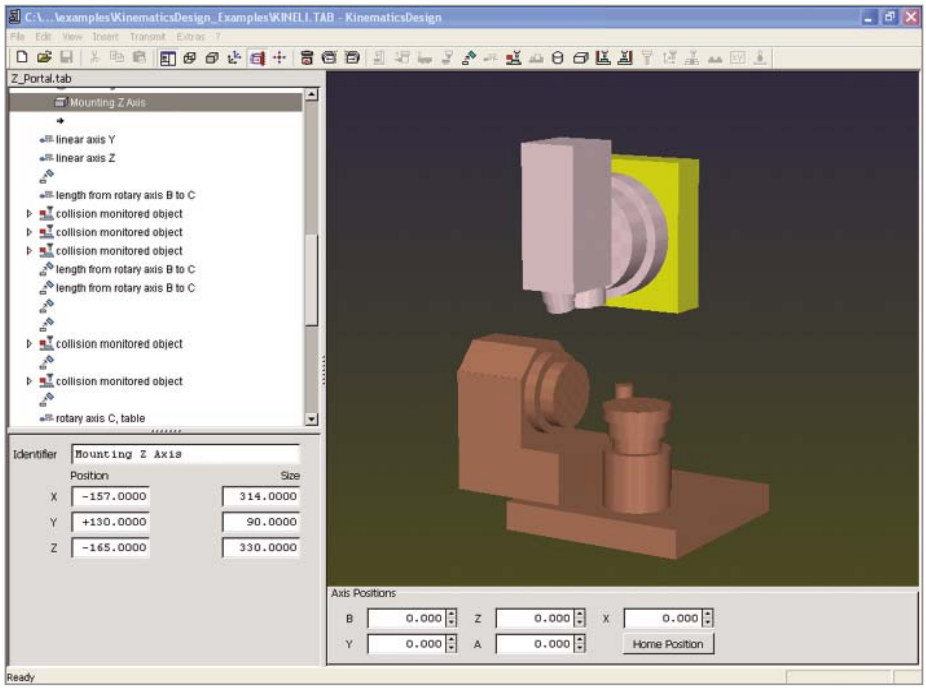
Of course the iTNC 530 also shows the machine operator—both with an error message and graphically—which machine components are endangered. If a collision warning is displayed, the TNC permits retracting the tool only in those directions which increase the clearance between the colliding objects.

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 defines 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, DCM cannot fully eliminate it.
- Only the machine manufacturer can define machine components. The operator creates fixtures from fixture templates, which are provided by HEIDENHAIN or by the machine manufacturer.
- Collisions between machine components (such as swivel heads) and the workpiece cannot be detected.
- DCM cannot be used during operation in following error mode (i.e. without feedforward).
- In the Test Run operating mode you can check for collision before actually machining the workpiece.



– 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. They allow 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. A clearly structured form is provided for this. After program start the iTNC then moves, if necessary, to a new position with a positioning logic influenced by you.

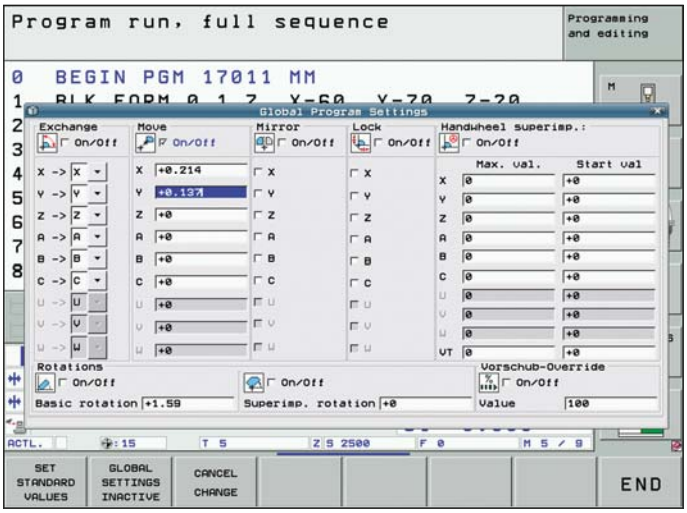
The following functions are available:

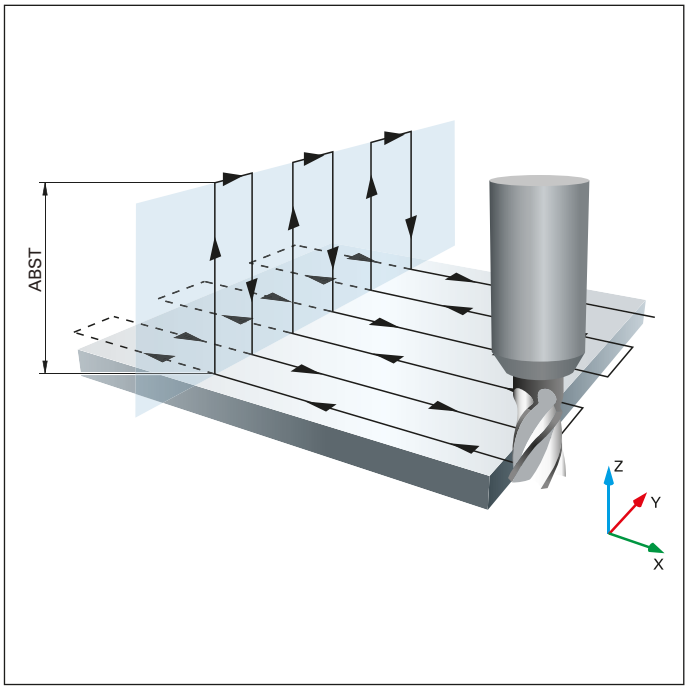
- Swapping axes
- Additional, additive datum shift
- Superimposed mirroring
- Axis locking
- Handwheel superimposition, with axis-specific memory of paths covered per handwheel, in **virtual axis direction**
- Superimposed basic rotation
- Superimposed rotation
- Globally valid feed-rate factor
- Limiting the machining area (limit plane)

Limit plane

The powerful **limit plane** function enables you to easily limit the working space. This enable you to:

- define ranges for machining, e.g. for repairs
- avoid limit-switch warnings, e.g. when a CAM program is run on a machine with a smaller traverse range, and
- edit the limit height, for example to machine a programmed surface in multiple steps by limiting the depth



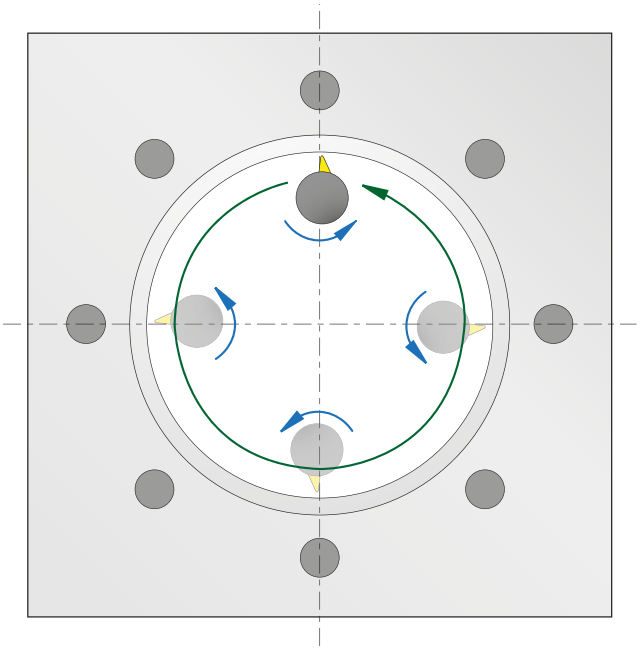


– Interpolation turning (option)

During interpolation turning the tool edge describes a circular motion where the orientation of the cutting edge is always oriented to the circle center. By varying the circle radius and the axial position, any rotationally symmetric objects can be produced in any working plane.

With the interpolation turning cycle, the iTNC 530 can create a rotationally symmetric shoulder in the active machining plane that is defined by its starting and ending point. The center of rotation is the tool location in the working plane at the time the cycle is called. The rotational surfaces can be inclined or rounded relative to each other.

This cycle can only be used for finishing. Roughing operations with multiple steps are not possible. The machining strategy can be chosen flexibly: from the outside in or vice versa, and also from top to bottom or vice versa. This results in four different machining strategies, which are distributed over the four quadrants.



Intelligent machining
– Dynamic Efficiency

dynamic + efficiency

With the concept of **Dynamic Efficiency**, HEIDENHAIN offers innovative TNC functions that help the user to make heavy machining and roughing more efficient while also enhancing its process reliability. The software functions support the machine operator but also make the manufacturing process itself faster, more stable and more predictable—in short, more efficient. Dynamic Efficiency permits higher removal rates and therefore increases productivity without making the user resort to special tools. At the same time, it prevents any tool overloading and the concomitant premature cutter wear. All of this means that with Dynamic Efficiency you can manufacture more economically while increasing process reliability.

Dynamic Efficiency comprises three software functions:

- **Active Chatter Control (ACC)** – The ACC option reduces chatter tendencies and permits greater infeeds
- **Adaptive Feed Control (AFC)** – The AFC option controls the feed rate depending on the machining situation
- **Trochoidal milling**—a function for the roughing of slots and pockets that eases the load on the tool and the machine

Each solution in itself offers decisive advantages in the machining process. But the combination of these TNC features, in particular, exploits the potential of the machine and tool and at the same time reduces the mechanical load. Changing machining conditions, such as interrupted cuts, various material plunging procedures or simple clear-out also show that these features pay for themselves. In practice, removal rates can be increased by 20 to 25 percent.

– Active Chatter Control option (ACC)

dynamic + efficiency

Strong forces come into play during roughing (power milling). Depending on the tool spindle speed, the resonances in the machine tool and the chip volume (metal-removal rate during milling), the tool can sometimes begin to "chatter." This chattering places heavy strain on the machine, and causes ugly marks on the workpiece surface. The tool, too, is subject to heavy and irregular wear from chattering. In extreme cases it can result in tool breakage.

To reduce the inclination to chattering, HEIDENHAIN now offers an effective control function with its Active Chatter Control option (ACC). The use of this control function is particularly advantageous during heavy cutting. ACC makes substantially higher metal removal rates possible. This makes it possible to increase your metal removal rate by up to 25 % and more, depending on the type of machine. You reduce the mechanical load on the machine and increase the life of your tools at the same time.

Heavy machining without ACC (above) and with ACC (below)

Intelligent machining
– Adaptive Feed Control option (AFC)

dynamic + efficiency

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.

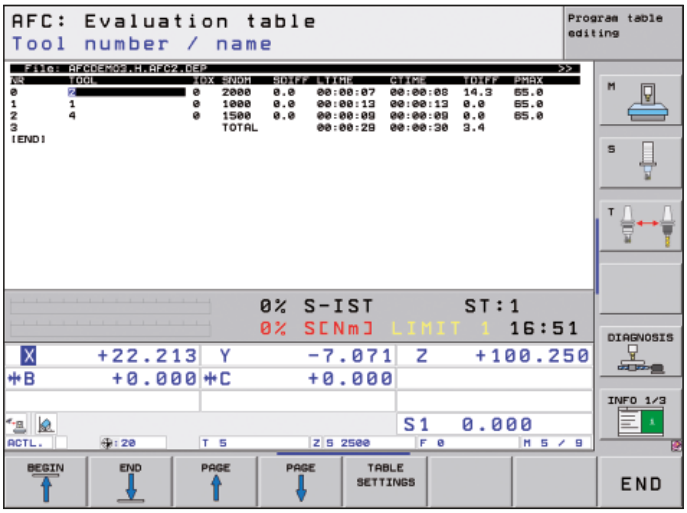
Adaptive feed rate control (AFC) automatically regulates the feed rate of the TNC, taking into consideration the respective spindle power and other process data. In a teach-in cut, the iTNC records the maximum spindle power. Then, before actual machining, you define in a table the respective limit values 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.

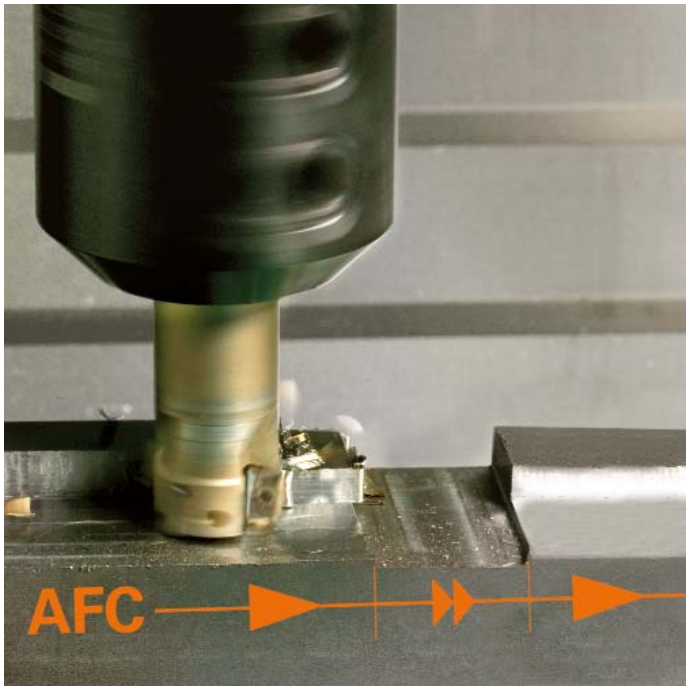
Adaptive feed rate control offers various advantages:

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 continuously 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 helps to prevent further damage after a tool breaks or is worn out.

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.





– Machining any contour slots with trochoidal milling

dynamic + efficiency

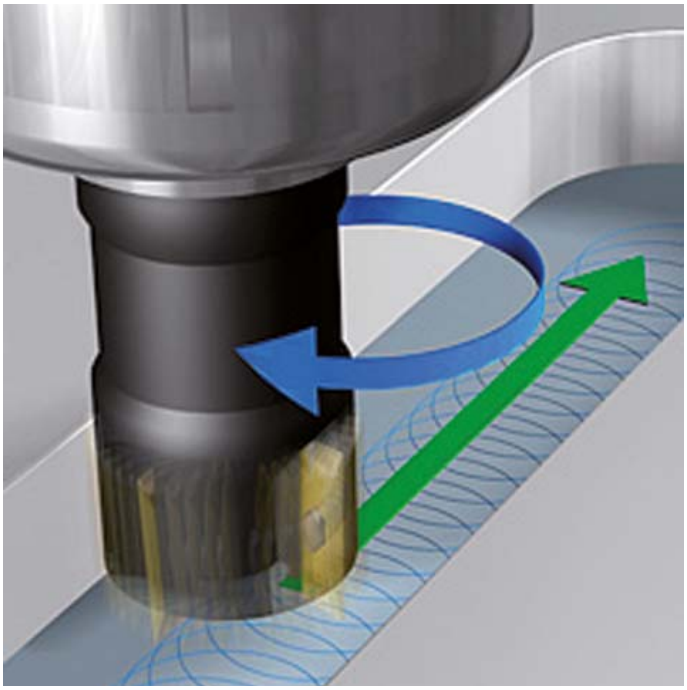
The benefit of trochoidal milling is its ultra-efficient machining of slots of all kinds. The roughing process is a circular motion superimposed on a forward linear motion. This procedure is referred to as trochoidal milling. It is used particularly for milling high-strength or hardened materials, where the high loads placed on the tool and machine usually only permit small infeeds.

With trochoidal milling, on the other hand, large cutting depths are possible since the prevailing cutting conditions do not increase the wear and tear on the tool. On the contrary, the entire length of a plain cutter's cutting edges can be used. This enables you to achieve a greater chip volume per tooth. Circular plunging into the material places less radial force on the tool. This reduces the mechanical load on the machine and prevents vibration. Enormous time savings can be realized by combining this milling method with the integrated adaptive feed control (AFC) option.

The slot to be machined is described in a contour subprogram as a contour train. You define the dimensions of the slot and the cutting data in a separate cycle. Any residual material remaining can then easily be removed with a subsequent finishing cut.

The benefits include:

- Engagement of the entire cutter length
- Higher chip volume
- Relief from mechanical load on the machine
- Less vibration
- Integrated finishing of the side wall



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.

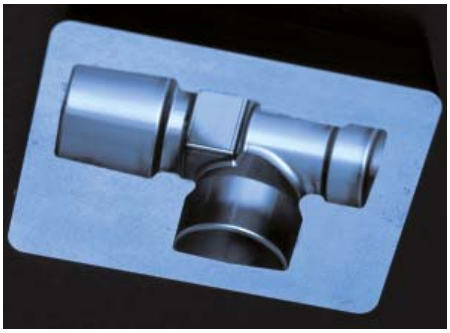
Very short block processing times
Block processing speeds are progressively being pushed into the background by sophisticated feedforward servo methods. Nevertheless, short block processing times remain a precondition for certain machining situations. One example is the machining of highly accurate contours with very high resolution. No problem for the iTNC 530. With block processing times below one millisecond, the iTNC 530 provides ideal control.

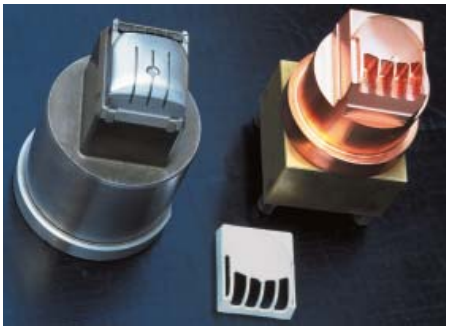
Very high contour accuracy
The iTNC 530 calculates the contour for up to 1024 blocks in advance. This enables it to adapt the axis velocities to the contour transitions. It controls the axes with special algorithms that ensure path control with the required limits to velocity and acceleration. The integrated filters specifically suppress machine-specific natural vibration. Of course, the desired accuracy of the surface is maintained.

Spline interpolation
If your 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.

Fast machining at specified accuracy
You as user specify the accuracy of the machined 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.

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. Of course, the iTNC 530 can interpolate simultaneously in up to five axes. To reach the required cutting speed, the iTNC 530 digitally controls spindle speeds up to **60 000 rpm**.





– Dynamic Precision

dynamic + precision

The hypernym **Dynamic Precision** stands for a number of HEIDENHAIN solutions for milling that can dramatically improve the dynamic accuracy of a machine tool. It is the result of a new perspective on the competing demand for accuracy, high surface quality and short machining times. The dynamic accuracy of machine tools can be seen in position errors at the tool center point (TCP), which depend on motion quantities such as velocity and acceleration (also jerk) and result from vibrations of machine components and other causes.

All the deviations are together responsible for dimensional errors and faults in the workpiece surface. They therefore have a decisive influence on quality and, when poor-quality parts are scrapped, also on productivity.

Because the stiffness of machine tools is limited for reasons of design and economy, problems such as compliance and vibration within the machine design are very difficult to avoid. Dynamic Precision counteracts these problems with intelligent control technology to enable designers to further improve the quality and dynamic performance of machine tools. That saves time and money in production.

The machine tool builder can use the options comprised by **Dynamic Precision** either individually or in combination:

- **CTC** – Compensation of position errors due machine elasticity between the encoder and the TCP. This increases accuracy during acceleration phases
- **AVD** – Active vibration damping improves surfaces
- **PAC** – Position-dependent adaptation of controller parameters
- **LAC** – Load-dependent adaptation of control parameters enhances accuracy regardless of load and aging
- **MAC** – Motion-dependent adaptation of control parameters

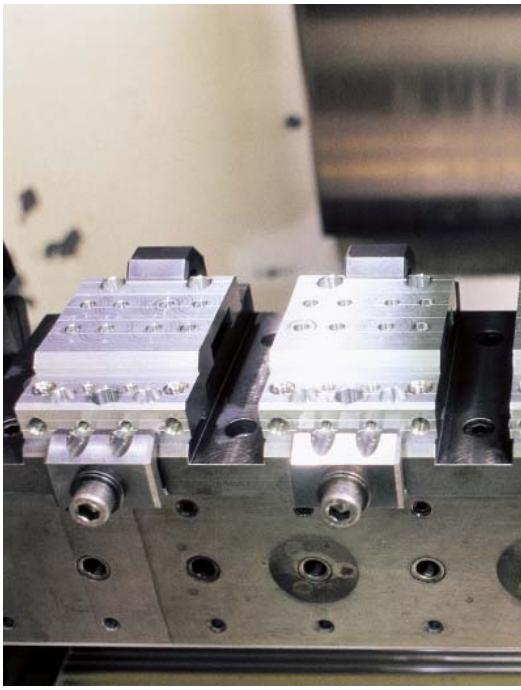
Automated machining
– The iTNC 530 manages, measures and communicates

The difference in requirements placed on the classical machine for tool and mold-making and machining centers is 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.

With the optionally available expanded tool management you can also graphically prepare and display any data.*

Pallet management
The iTNC 530 can assign the appropriate part program and datum shift to parts mounted on pallets and brought to the machine in any 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.



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

Tool-oriented machining
In 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.

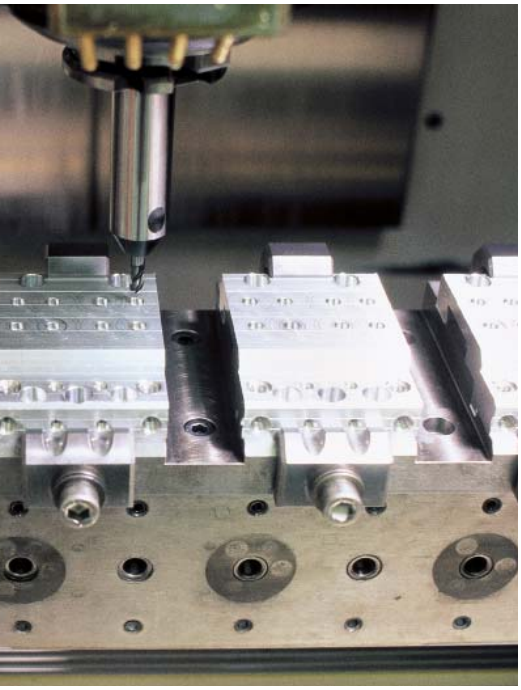
The iTNC 530 supports you with convenient input forms with which you can assign a tool-oriented machining operation to a pallet with several workpieces on several fixtures. You can write the program, however, in the familiar workpiece-oriented sequence.

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. For the measuring cycles you insert a 3-D touch probe from HEIDENHAIN (see page 48) into the spindle in place of a tool. The following measuring cycles are available:

- Recognize a workpiece and call the appropriate part program
- Check whether all machining operations were conducted correctly
- Determine infeeds for finishing
- Detect and compensate tool wear
- Check workpiece geometry and sort parts
- Log measured data
- Ascertain the machining error trend

Tool measurement and automatic compensation of tool data
Together with the TT or TL systems for tool measurement (see page 49), the iTNC 530 makes it possible to measure tools automatically while they are in the machine spindle. 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.

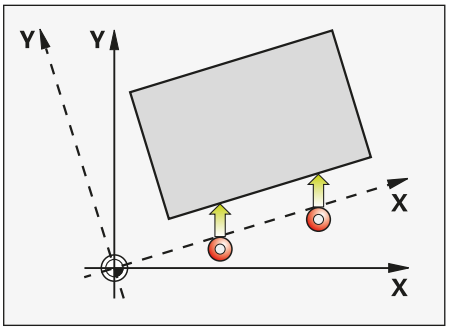
The iTNC 530 features application-oriented, real-world setup functions. They support the user, help to reduce non-productive time, and make overnight, unattended production possible. Together with the **3-D touch probes,** the iTNC 530 offers numerous probing cycles for automatic alignment of the workpieces,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

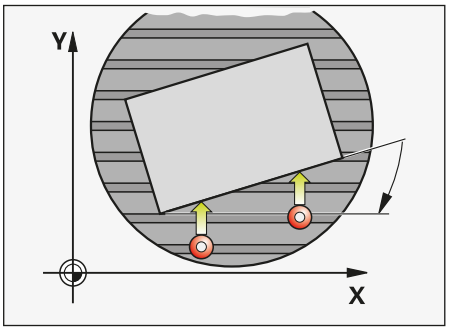
Delicate manual traverse
For setup, you can use the direction keys to move the machine axes manually or in incremental jog. A simpler and more reliable way, however, is to use the electronic handwheels from HEIDENHAIN (see page 51). Particularly with the portable handwheels you are always close to the action, enjoy a close-up view of the setup process, and can control the infeed responsively and precisely.

Workpiece alignment
With HEIDENHAIN 3-D touch probes (see page 48) 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 determines the actual workpiece position 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





Setting datums
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 datum management
- in a datum table, or
- by directly setting the displayed value.

Datum management
The datum management 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 datum management you can save **any number of datums** 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.

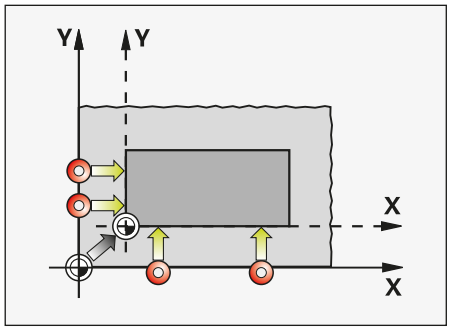
On machines with an **automatic spindle head changer**, the reference point remains unchanged after a head exchange, even if they differ kinematically (i.e. in their dimensions).

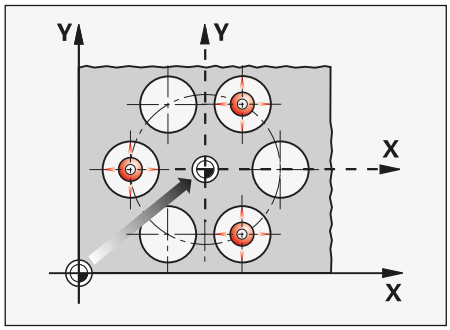
The iTNC automatically creates separate preset tables for individual **traverse ranges** (such as for alternating table machining). When changing traverse ranges, the iTNC activates the correct table with the most recently active reference point.

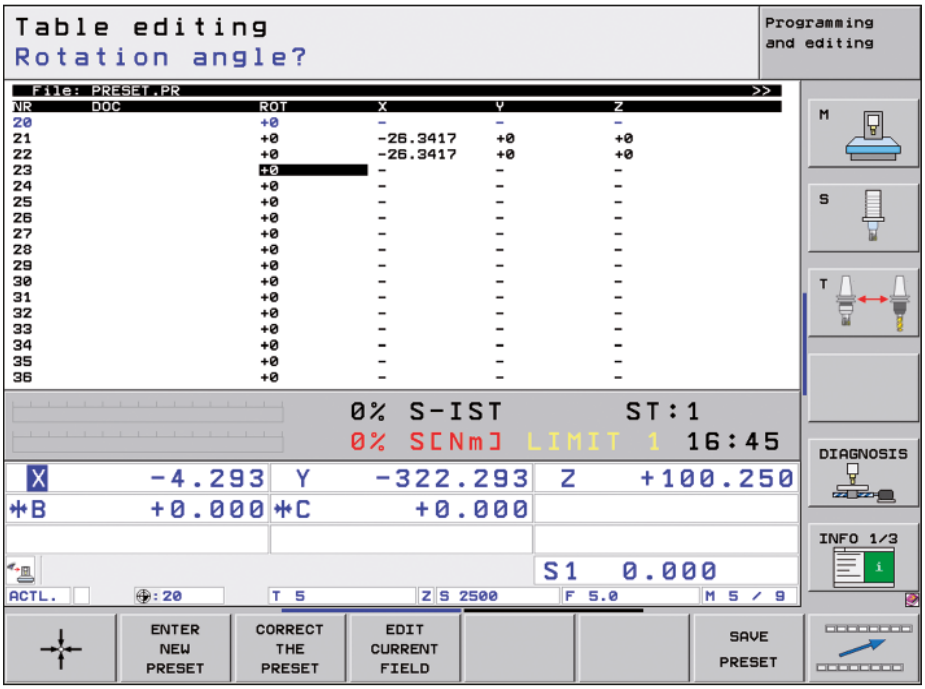
There are three ways to save datums rapidly in the datum management:

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

Setting a datum
You can set a datum 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.

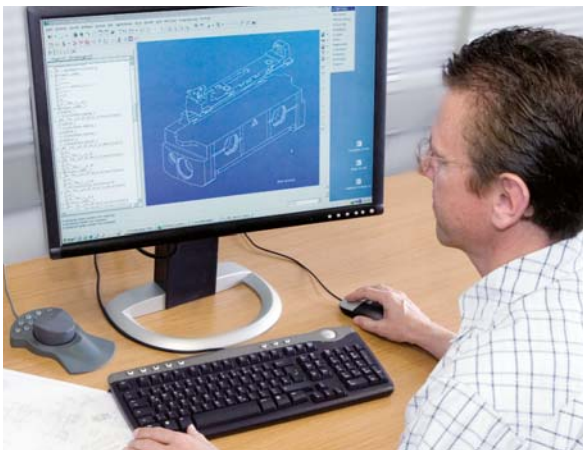
Programming at the machine
HEIDENHAIN controls are workshop oriented, which means that they were conceived for programming right at the machine. The iTNC 530 supports you with two user interfaces:

For over 30 years, HEIDENHAIN **Klartext** conversational programming has been the standard programming language for all TNC controls and for shop-floor programming in general. The **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 conversational prompts, dialog guidance, programming steps or soft keys, all texts are available in numerous languages.

Even if you are used to **ISO programming,** however, the iTNC is still the right control—you can enter ISO programs over the alphanumeric keyboard.

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. It 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.

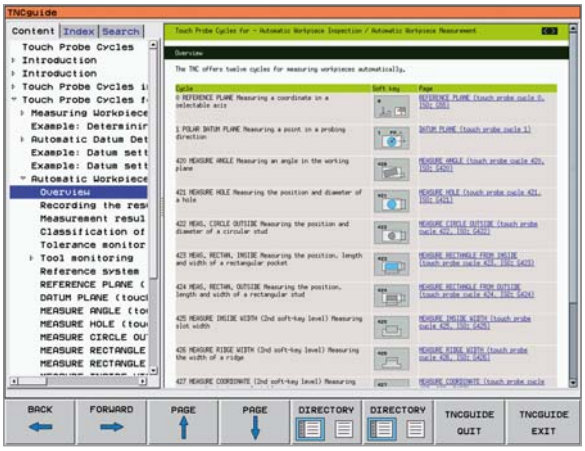
You can activate TNCguide by simply pressing the help key on the iTNC keyboard or by clicking any soft key with a pointer in the shape of a question mark. You switch the cursor by simply clicking the help symbol that is always visible on the TNC screen ( ).

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, which is especially useful when programming a function. This function is particularly helpful with the soft keys. The method and effect of operation are explained in detail.

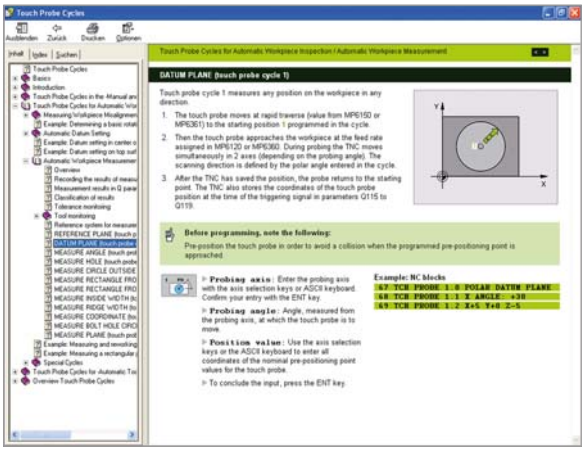
The iTNC 530 is shipped with integrated documentation for the respective NC software in English and German. Other languages are available for download free of charge as soon as the translations become available. After download, you can save the national language files in the corresponding language directory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

The following User's Manuals are available in the help system:
- Conversational Programming
- smarT.NC (pocket guide format)
- Cycle Programming
- DIN/ISO programming
- iTNC 530 programming station (included only in the programming station)

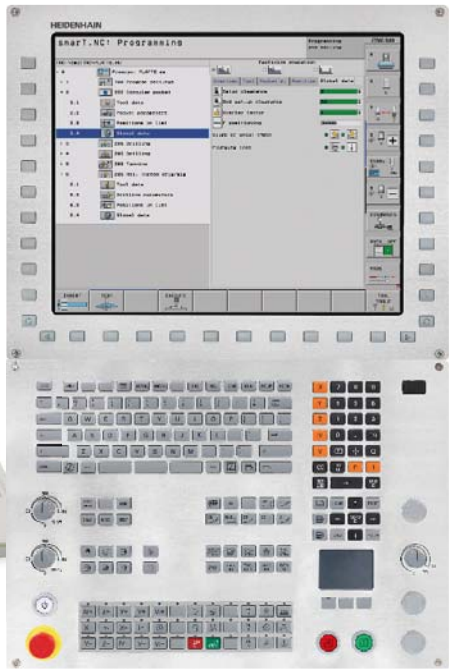
Also, the iTNC can display standard formats (PDF, BMP, GIF, JPG etc.).



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



... or at the programming station.

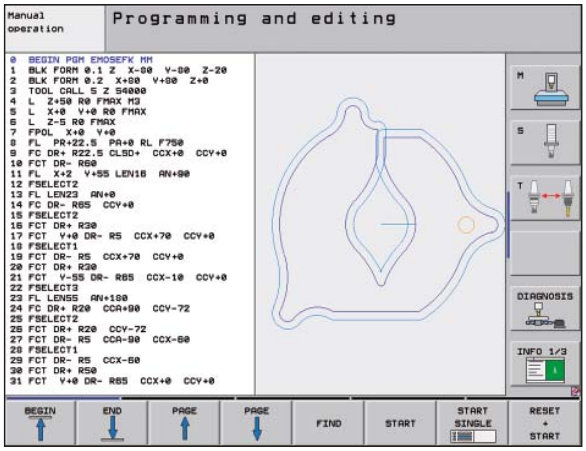


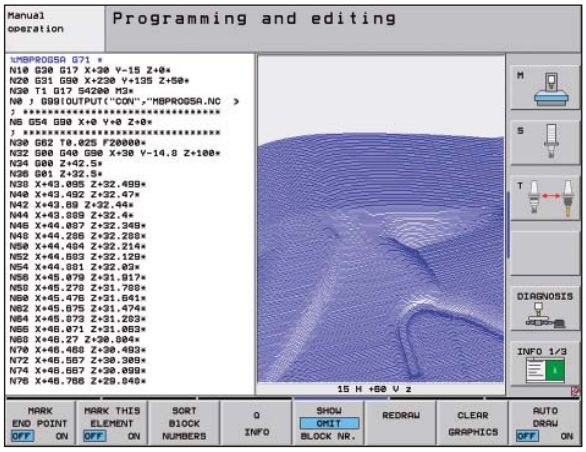
Programming, editing, testing
– Graphic support in any situation

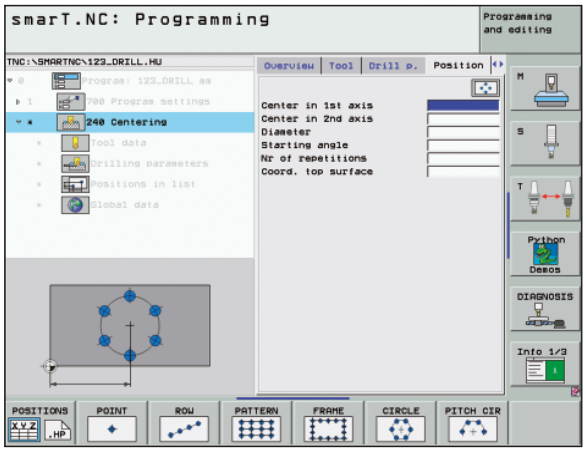
Programming graphics
The two-dimensional programming graphics give you additional security: while you are programming, the iTNC 530 draws every entered traverse command on the screen.

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 Klartext 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 a separate help graphic for each required input.







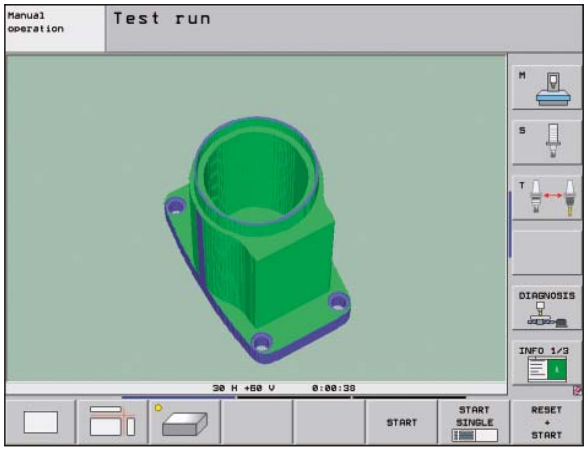
Program verification graphics
To play it safe before running a program, the iTNC 530 can graphically simulate the machining of the workpiece. It can display the simulation in various ways:

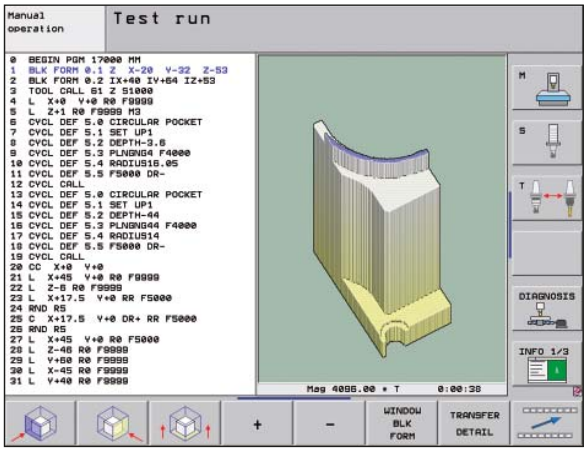
- In a plan view with different shades of depth
- In three planes (as in the workpiece drawing)
- In a solid model, 3-D view

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 multiside machining can be displayed. In addition, the iTNC 530 indicates the calculated machining time in hours, minutes and seconds.

Program-run graphics
With the iTNC 530, the programming graphics or verification graphics are available 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

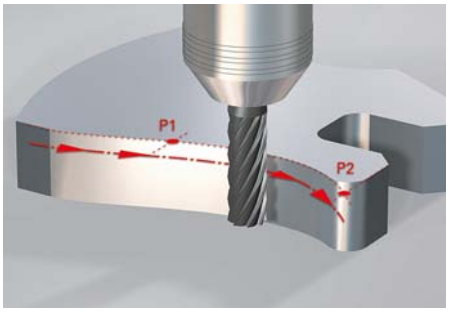
Two-dimensional contours are the bread and butter of the modern machine shop. Here the iTNC 530 offers a variety of possibilities.

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.

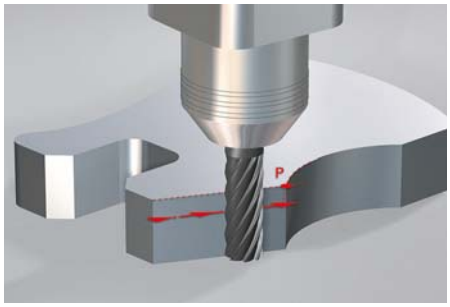
You simply specify the starting or end point of the contour and the approaching or departing radius of the cutter edge—the control does the rest for you.

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. Contour damage can occur, for example, when roughing a contour with a large tool.



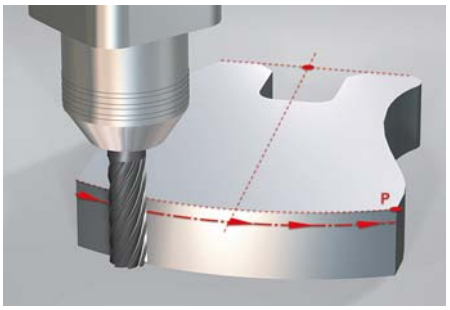
CT

Circular path with smooth, (tangential) connection with the preceding contour element, defined by end point



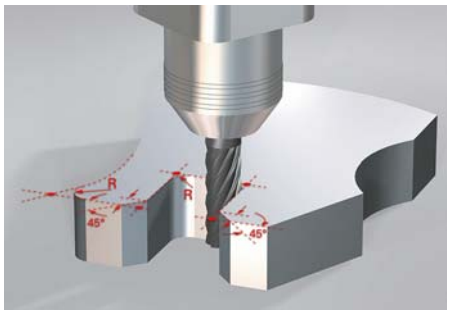
L

Straight line defined by its end point



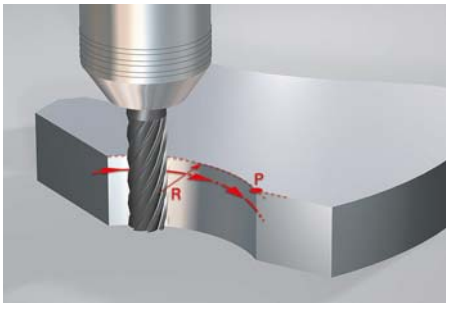
CC C

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



RND Corner rounding: circular path with smooth (tangential) connection on both sides, defined by radius and corner point

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

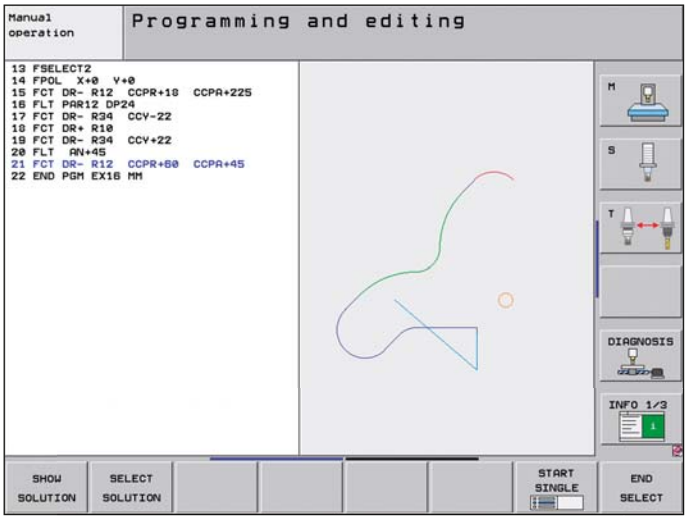


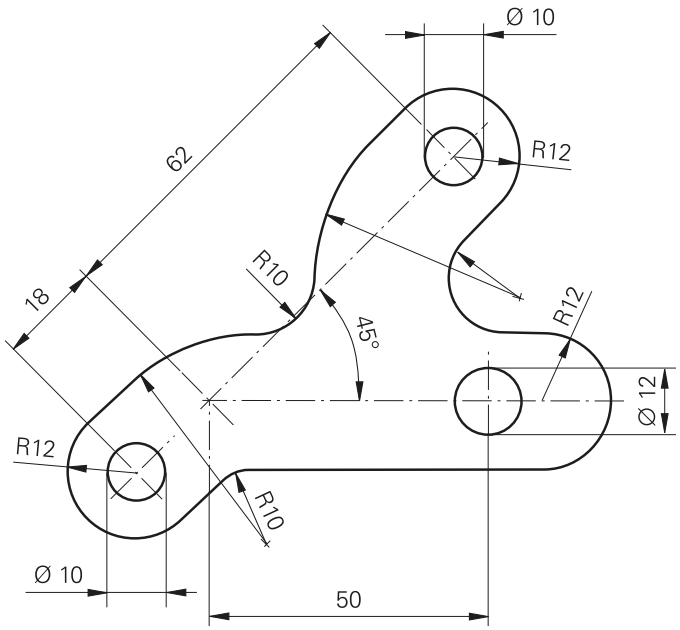
CR

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

– Programming contours unconventionally

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

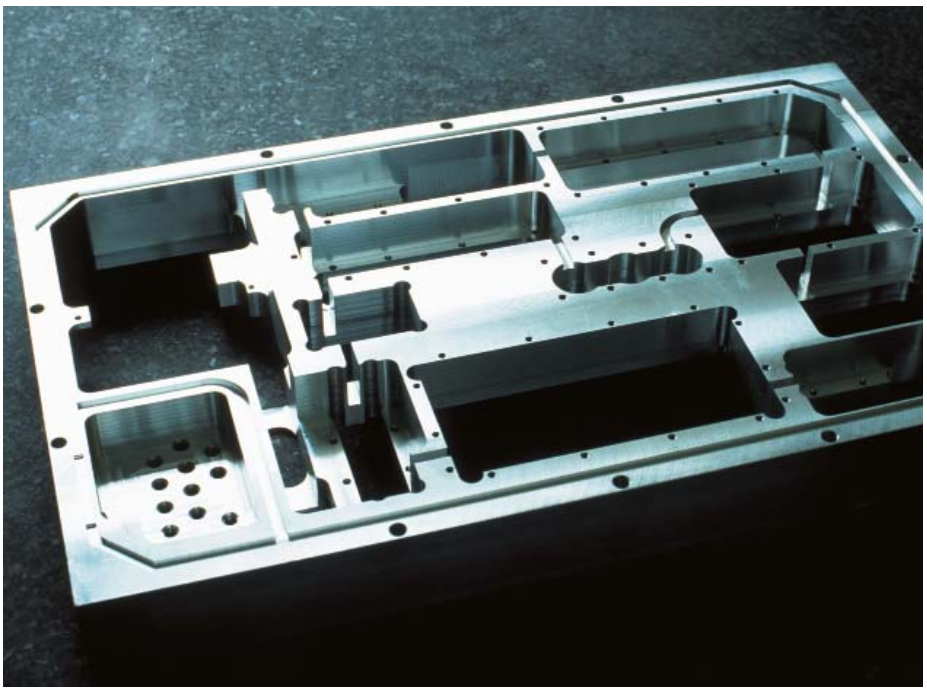
Frequently recurring operations that comprise several working steps are stored in the iTNC 530 as cycles. You program them under conversational guidance and are supported by graphics that clearly illustrate the required input parameters.

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
The **Subcontour List cycles** (SL) 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 avoids 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.

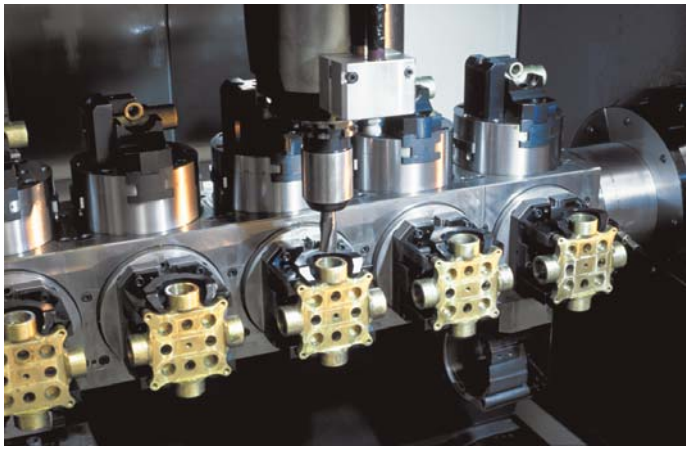
You can also program **"open" contours** with the SL cycles. This enables the iTNC 530 to observe allowances for 2-D contours, to move the tool in alternating directions after each infeed, to avoid contour damage at undercuts and to maintain the defined milling direction (climb or conventional) after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

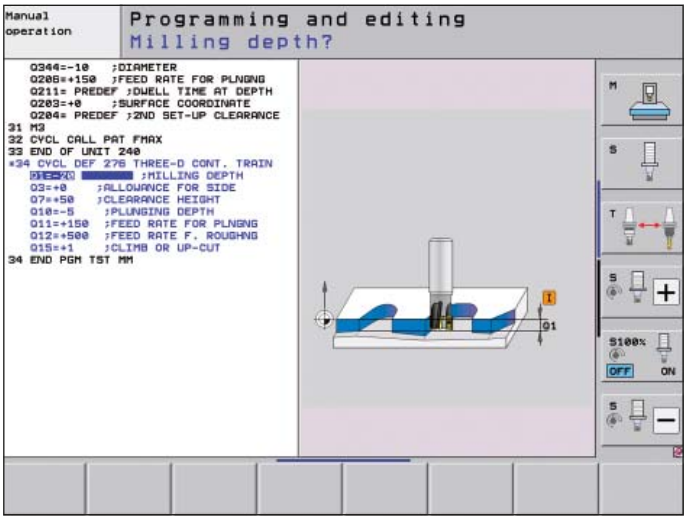


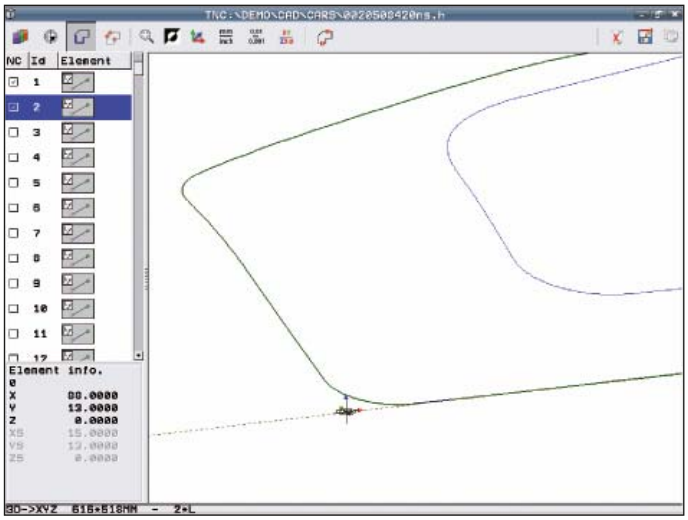
– Peripheral milling with the contour train cycle

Special demands are in force during peripheral milling of cutting and bending tools, particularly for large-scale mold making for the automobile industry. The outlines are generated from CAM systems and usually include not only the coordinates in the working plane, but also those in the tool axis direction. What is special is that the cutting or bending edge does not run at a constant Z height, but differs greatly depending on the component.

With the Contour Train cycle you can machine such 3-D contours very simply: you define the contour to be machined in a subprogram. You specify the approach behavior, machining mode and radius compensation with a separate cycle. You can machine the 3-D contour train with or without infeed, depending on whether an infeed has been defined.

The 3-D contours to be machined can be created very easily if they can be loaded from existing NC programs that were generated by a postprocessor. This applies in particular if a smaller tool must be used to rework specific areas. An enhancement was implemented for this in the DXF converter, making it possible to load contours or parts of contours from conversational programs.





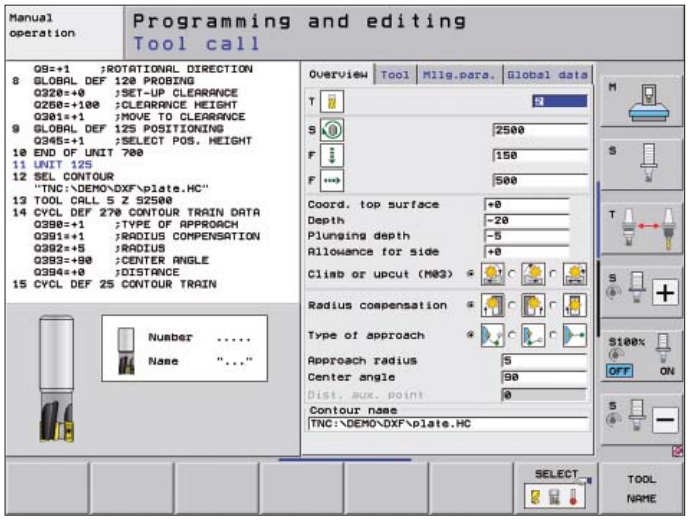
Well thought out, simple and flexible
– smarT.NC—the alternative operating mode

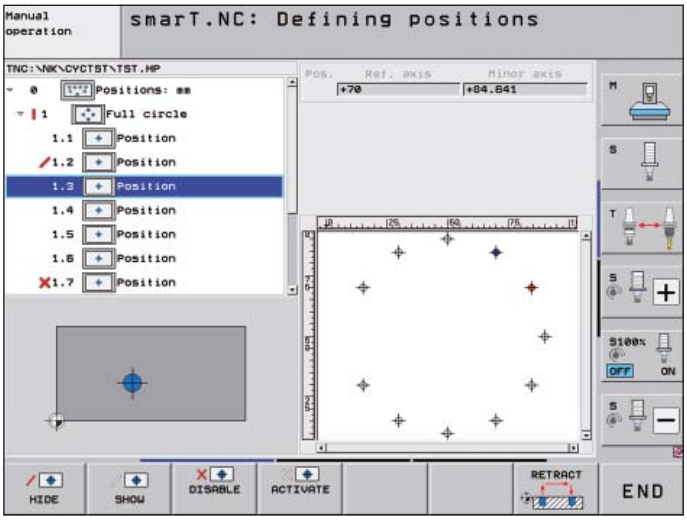
TNC controls from HEIDENHAIN have always been user friendly: Thanks to their simple programming in HEIDENHAIN Klartext conversational language, field-proven cycles, unambiguous function keys, and clear and vivid graphic functions, they now count among the most popular shop-floor programmable controls.

The alternative smarT.NC operating mode makes programming even easier. Particularly as a TNC beginner, with the well thought-out input forms you can create your NC program in a few quick steps. Of course you'll be aided during programming 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.

And even the TNC specialist profits from smarT.NC. The smarT.NC wizard completely unifies the worlds of smarT.NC and conversational programming. The strengths of both worlds are now available in a single user interface. The full flexibility of conversational programming, based on NC blocks, can now be combined at any location with the fast, fillable-form based work-step programming method of smarT.NC.





– smarT.NC—the alternative operating mode (continued)

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

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.

Programming contours
You define contours in the same way as you do part programs—using fillable forms with graphic guidance. The individual contour elements are likewise displayed in the outline; the associated data are shown in a form.

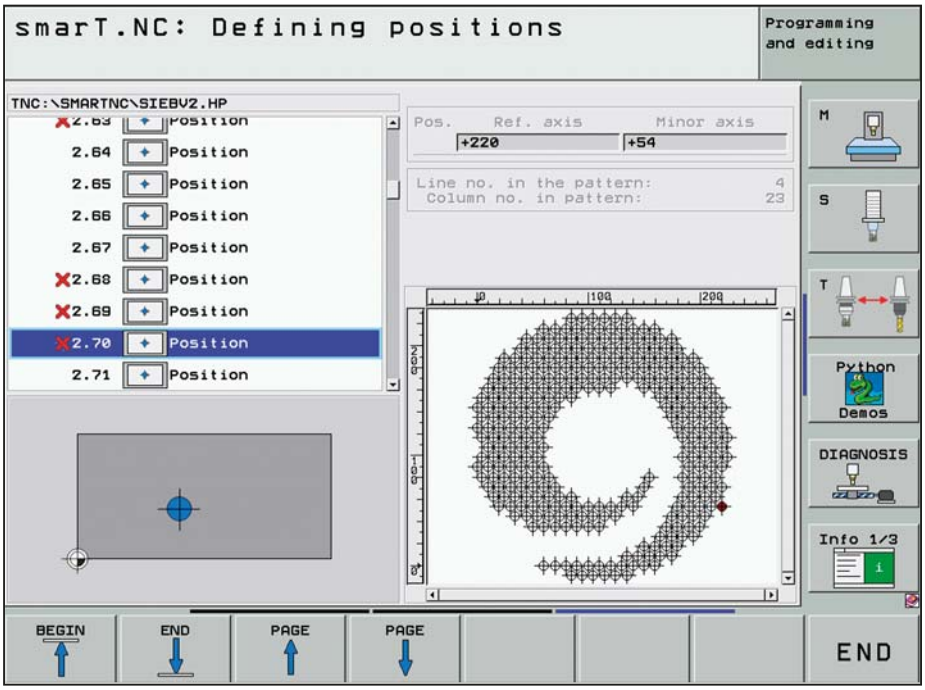
If a workpiece is not dimensioned for conventional NC programming, smarT.NC also features the powerful HEIDENHAIN FK free contour programming.

If you have access to the contour as a DXF file, you can transfer it directly from the respective editing form with a few mouse clicks.

Simple and flexible programming of 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.



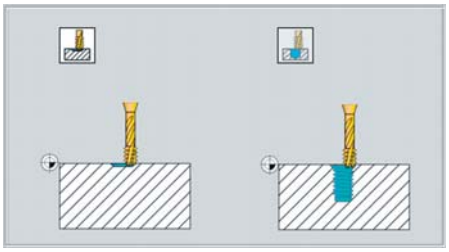
Well-designed for fast operation
With the split-screen, smarT.NC provides an **easily understandable program structure.** On the left screen, you can navigate quickly in a variable tree structure. On the right, clearly arranged input forms immediately show you the defined machining parameters. The soft-key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

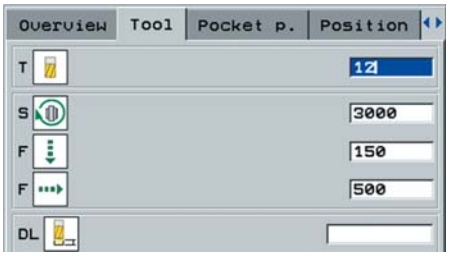
smarT.NC allows **fast editing**: With the new navigation keys, you can quickly reach any machining parameter in an input form. With a separate key you can switch directly between the form views.

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

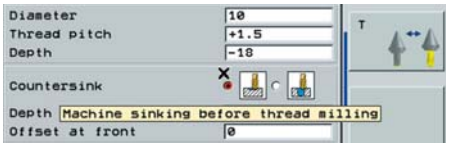
Clearly arranged **help graphics** illustrate all required input.

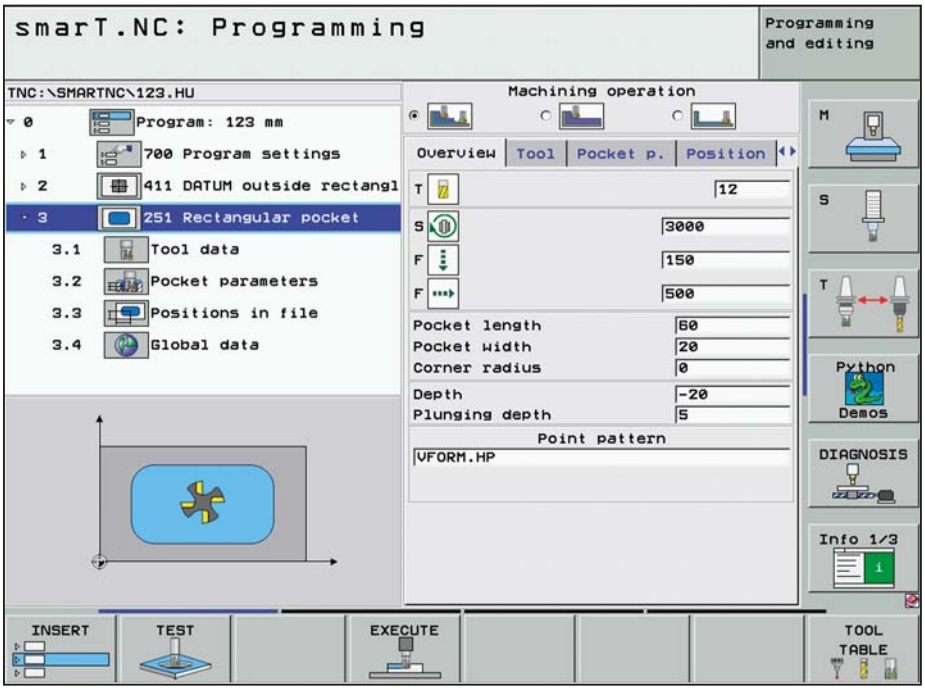


Graphic symbols increase concept recognition when the same type of input is required for different operations.



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? You can open DXF files directly on the iTNC 530 in order to extract contours or machining positions from it. This not only saves time otherwise spent on programming and testing, you can also be sure that the finished contour is exactly according to the design engineer'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.

In the DXF converter can also open conversational or ISO programs that were created externally in CAM systems. The DXF converter graphically displays the tool paths generated in the CAM system. You can select parts of the contour and save them as separate NC programs or in the buffer memory. This is especially helpful, for example, when you need to rework parts of contours with a smaller tool, or even if you just want to rework some parts of a 3-D shape. This function saves you a trip to the CAM office, and can be performed at any time directly on the iTNC. You can then machine this newly created NC program directly or in combination with the iTNC's contour train cycles.

As a rule, DXF files contain multiple layers, with which the design engineer organizes the drawing. So that as little unnecessary information as possible appears on the screen during selection of the contours, with a keystroke you can hide all **excessive layers** contained in the DXF file. This requires a keyboard with touchpad or an external pointing device. The iTNC can select a contour train even if it has been saved in **different layers.**

The iTNC also supports you when **defining the workpiece datum**. The datum of the drawing for the DXF file is not always located in a manner that lets you use it directly as a 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:

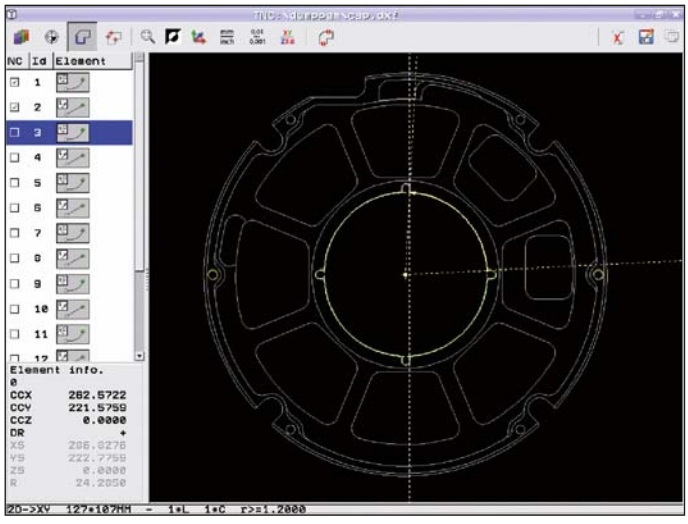
- The beginning, end or mid-point of a line
- The beginning, end or center point of a circular arc
- Quadrant transitions or center point of a circle
- Intersection of two lines, regardless of whether it is located inside or outside the programmed segments
- Intersection of a line and a circular arc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

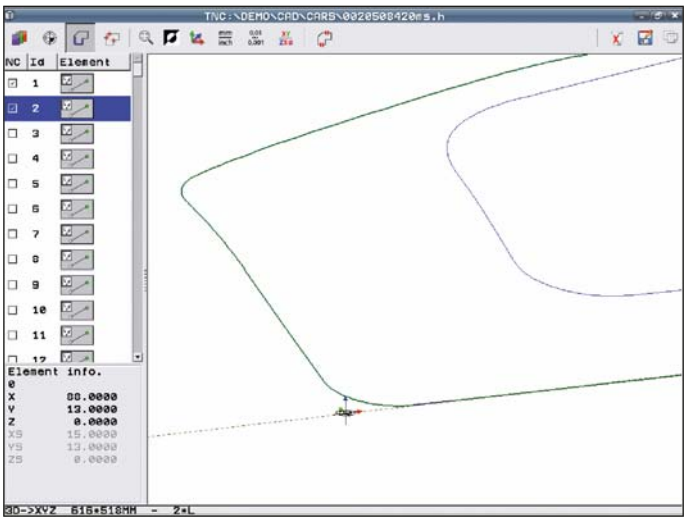
Contour selection is exceptionally user friendly. You select any element by clicking it with the mouse. As soon as you select a second element, the iTNC detects your desired direction of machining, and starts the **automatic contour detection.** The iTNC automatically selects all clearly identifiable 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 in order to use drilling positions or starting points for pocket machining. With drilling positions this can be done very easily: use the mouse to simply select the desired area. In a pop-up window with filter function, the TNC displays all hole diameters that are within the area you have selected. To select the desired hole diameters and restrict the number of hole positions, simply click the corresponding filter symbol to change the filter limits.

A zoom function and various possibilities for settings round out the functionality of the DXF converter. In addition, you can define the resolution of the contour program to be output in case you want to use it on older TNC controls, or a transition tolerance if 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 the iTNC

Frequently, 5-axis programs are written on offline CAM systems and then transferred to the control over a data line. The iTNC 530 offers many benefits for these applications as well. 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.

The iTNC 530 works very well together with all CAM systems. HEIDENHAIN intensively supports the manufacturers of postprocessors for the best possible use of the iTNC 530's powerful features.

Programs created offline
NC programs for 5-axis operations are normally written on 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:

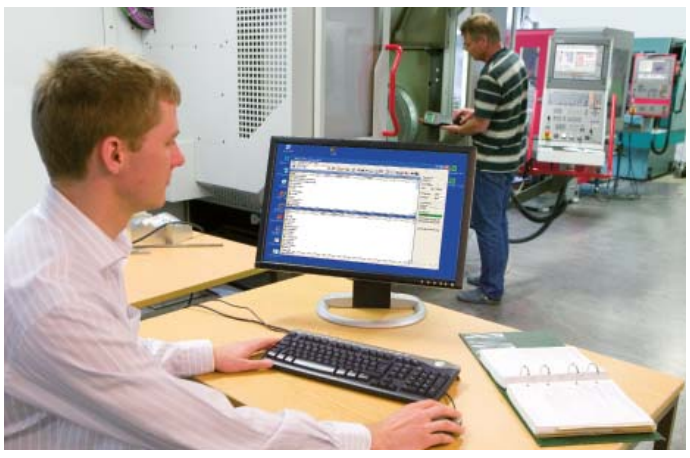
- Machine-specific NC programs take the respective machine configuration into account and include all coordinates of the NC axes on that machine.
- Machine-neutral NC programs define the contour and use vectors to define the respective tool position on the contour. The iTNC 530 then uses the information 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 G-code 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. Examples are:

- Tool Center Point Management (TCPM)
- Structuring function
- 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.

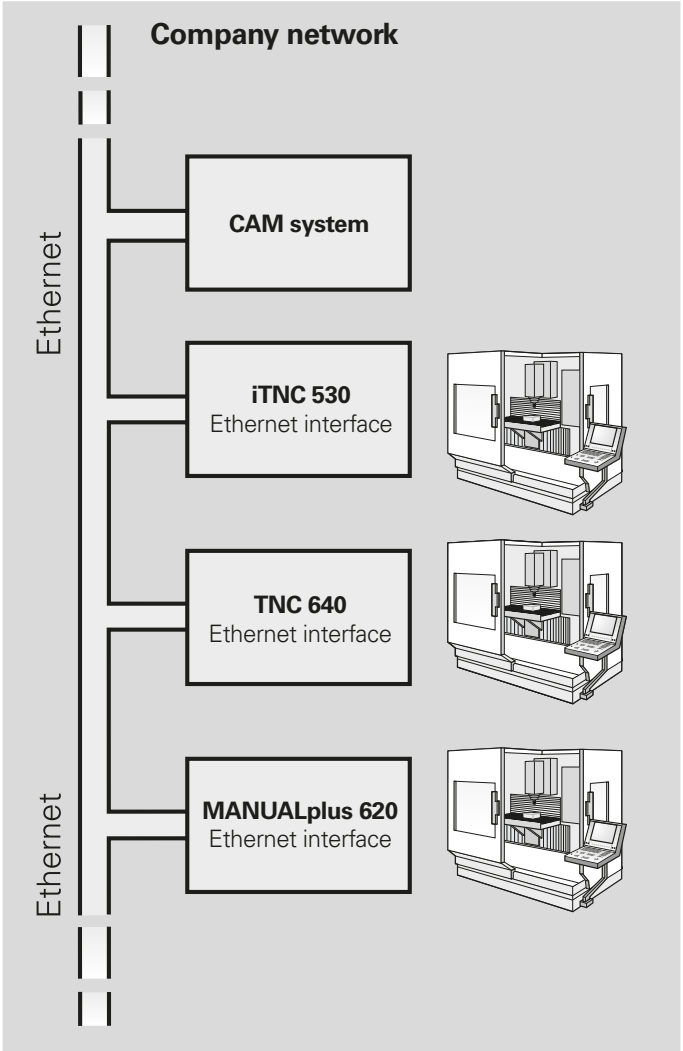
CAM systems don't always generate programs optimized for the machining process. Therefore, the iTNC 530 offers a point filter for smoothing externally created NC programs. The filter function creates a copy of the original program, and then adds any points required by the parameters that you set. This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



– 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, the iTNC 530 features a latest-generation Gigabit 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 with speeds of up to 1000 Mbit/s ensures very fast transmission times, even with large 3-D programs.

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.

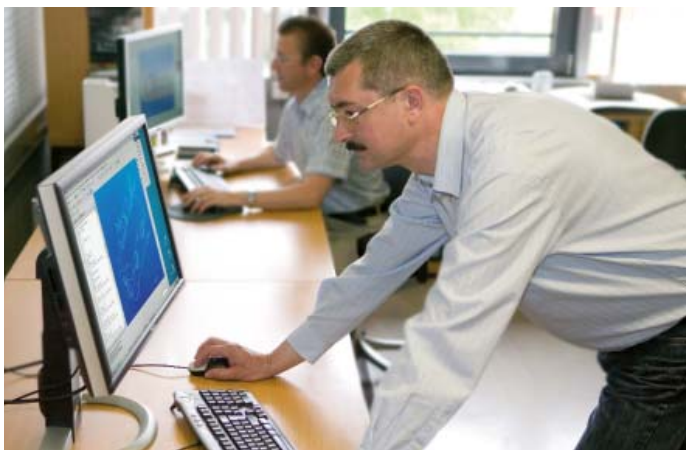


Programs for data transfer
With the aid of the free PC software **TNCremo** from HEIDENHAIN, you can

- transfer remotely stored part programs and tool or pallet tables in both directions,
- start the machine,
- make backups of the hard disk,
- and interrogate the operating status of the machine.

With the powerful **TNCremoPlus** PC software you can also transfer the screen contents of the control to your PC using the live-screen function.

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



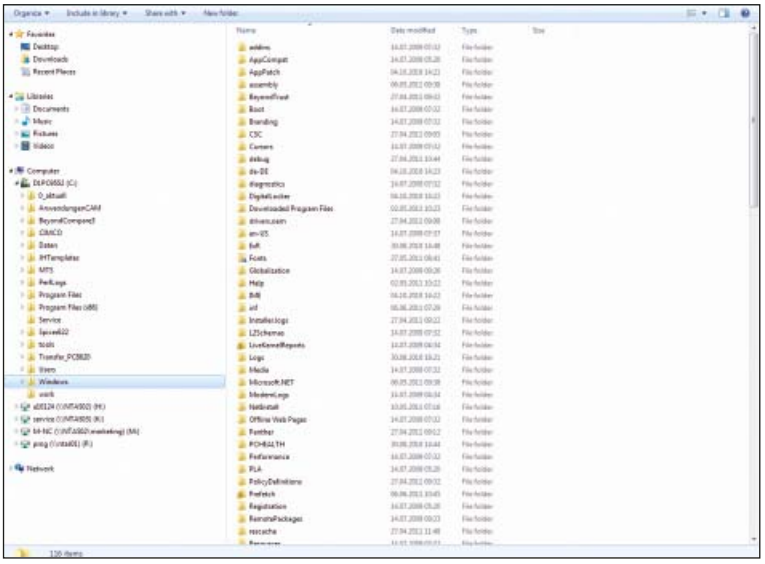
Open for communication
– Remote desktop manager (option)

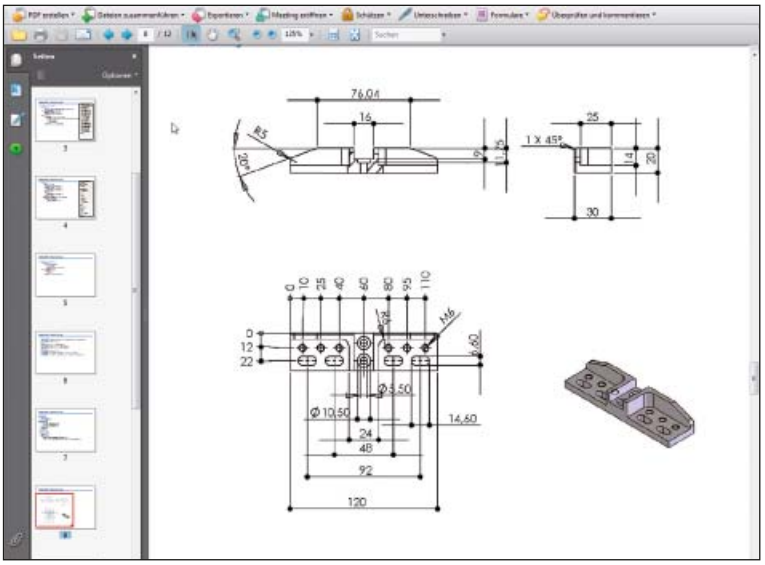
Windows applications on the iTNC 530*
For standard Windows applications, the iTNC 530 can be equipped with the Windows operating system. The main computer attends as always to the real-time tasks and the HEIDENHAIN operating system, while a second IPC computer stays available exclusively for the standard 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 must be prepared by the machine tool builder for this function.
Windows is a brand name of the Microsoft Corporation.

– The iTNC programming station

Why a programming station?
It's well known that it is easy to create part programs with an iTNC at the machine, even while another part is being machined. Nevertheless, short reloading times and other machining tasks can often 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 of programs created offline
Of course you can also test programs that were written on a CAM system. The high-resolution, program verification graphics help you to easily spot contour damage and hidden details even in complex 3-D programs.

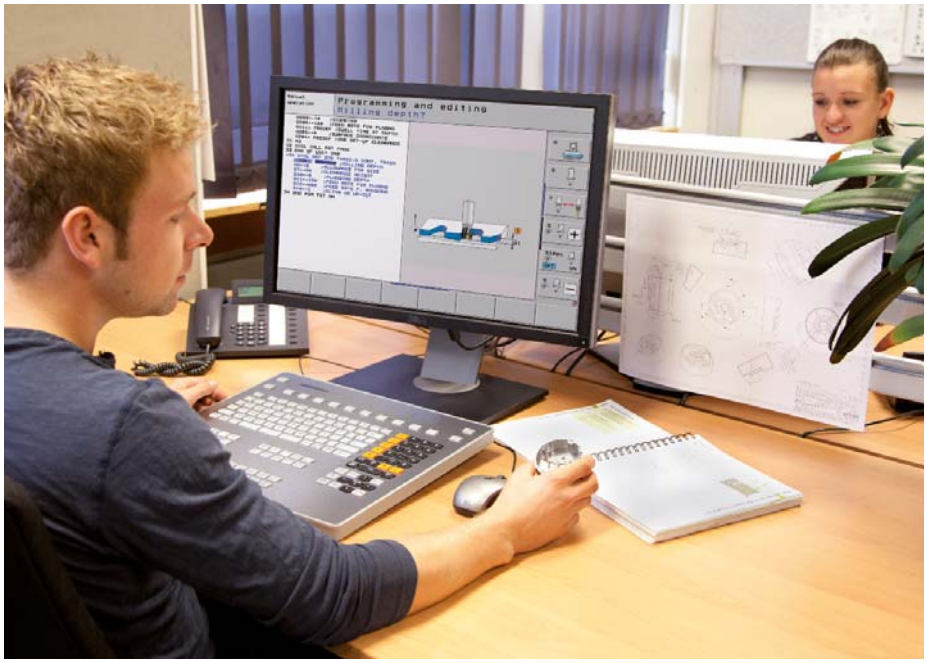
Training with the iTNC programming station
Because the iTNC programming station is based on the same software as the iTNC 530, it is ideally suited for apprentice and advanced training. The program is entered on the original keyboard unit. Even the Test Run mode functions exactly as it does on the machine. 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 HEIDENHAIN conversational 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 is 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 displays the familiar TNC user interface.

Or as an alternative, you can even operate the programming station without an iTNC keyboard. You can use your mouse on a convenient virtual keyboard instead—it is displayed together with the iTNC control panel and features the most important dialog initiation keys of the TNC.



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 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 touch probe* is inserted directly into the machine tool spindle. It can be equipped with various shanks depending on the machine. The ruby ball tips are available in several diameters, and the styli in different lengths.

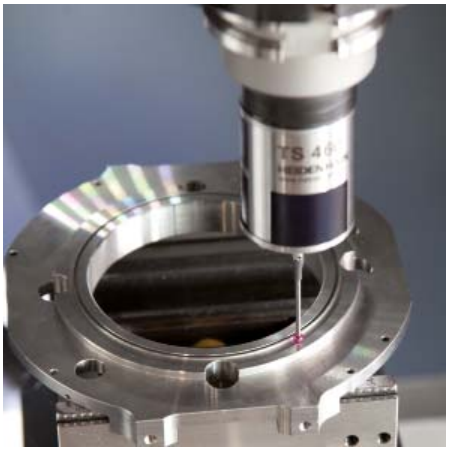
* The touch probes must be interfaced to the iTNC 530 by the machine tool builder.

Touch probes with **cable connection for signal transmission** for machines with manual tool change and for grinding machines and lathes:

TS 260 – New generation, axial or radial cable

Touch probe with **radio or infrared signal transmission** for machines with automatic tool change:

TS 460 – New generation standard touch probe for radio and infrared transmission, with compact dimensions
TS 444 – Battery-free voltage supply through integrated air turbine generator from compressed air, for infrared transmission, with compact dimensions
TS 740 – High probing accuracy and repeatability, low probing force, with infrared transmission



TS 260

TS 460

SE 660

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

Tool measurement
– Measuring length, radius and wear directly in the machine

The tool is of course a decisive factor in ensuring a consistently high level of production quality. This means that an exact measurement of the tool dimensions and periodic inspection of the tool for wear and breakage, as well as the shape of each tooth, are necessary. HEIDENHAIN offers the TT trigger tool touch probes 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 before machining as well as during interruptions.

The **TT tool touch probes** measure the tool length and radius. When probing the rotating or stationary tool, e.g. during individual tooth measurement, the contact plate is deflected and a trigger signal is transmitted directly to the iTNC 530.

The **TT 160** uses signal transmission by cable, whereas the **TT 460** operates with wireless signal transmission over radio or an infrared beam. It is therefore particularly suitable for use on rotary and tilting tables.

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 errors of individual teeth along with the tool length and radius.

TL Micro

TT 460

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

Inspecting and optimizing machine accuracy
– Calibrating rotary axes simply with KinematicsOpt (option)

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

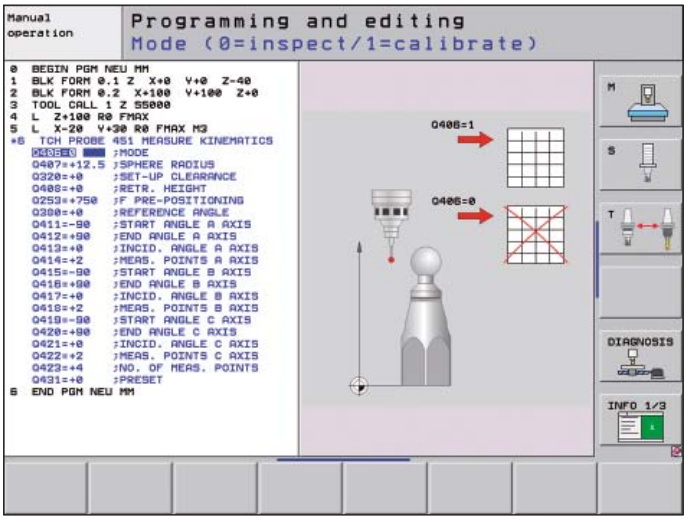
The TNC function **KinematicsOpt** is an important component to help you meet these high requirements: With a HEIDENHAIN touch probe inserted, a 3-D touch probe cycle measures your machine's rotary axes fully automatically. The results of measurement are the same regardless of whether the axis is a rotary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

To measure the rotary axes, a calibration sphere 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 range that you want to measure.

From the measured values, the TNC calculates the static 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 kinematics table.

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

An especially rigid calibration sphere is necessary for optimum use of KinematicsOpt. This helps to reduce deformations that occur as the result of probing forces. That is why HEIDENHAIN offers calibration spheres with highly rigid holders that are available in various lengths.



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 520 and HR 550 portable handwheels
The HR 520 and HR 550 are particularly helpful for when you have to work close to the machine's working space. The axis keys and certain functional keys are integrated in the housing. In this way you can switch axes and set up the machine at any time—regardless of where you happen to be standing. As a wireless handwheel, the HR 550 is ideal for use on large machine tools. If you no longer need the handwheel, just attach it to the machine somewhere by its built-in magnets.

The following functions are available:

HR 520, HR 550
- 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
- Selection of axes via keys or soft keys
- Keys for continuous traverse of the axes
- Emergency stop button
- Actual position capture
- NC start/stop
- Spindle on/off
- Soft keys for machine functions defined by the machine tool builder

HR 550

... and if there's a problem?
– Diagnostics for HEIDENHAIN controls

The operating reliability of machine tools and controls has improved continually in recent years. 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.

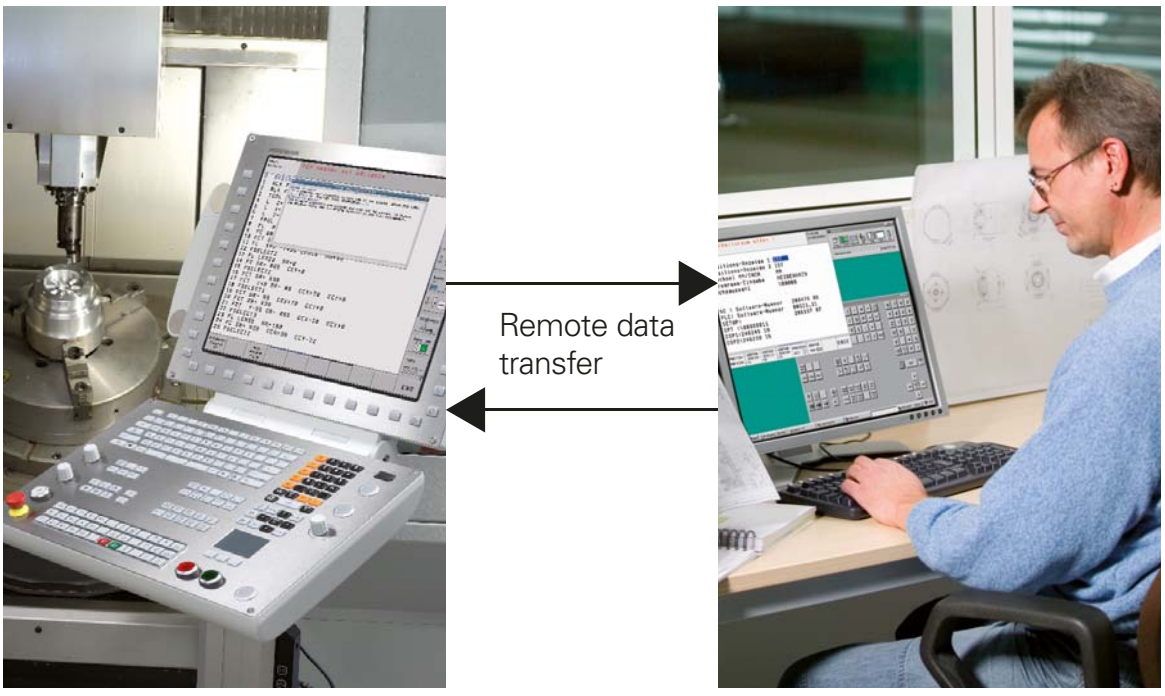
HEIDENHAIN offers the **TeleService** PC software for remote diagnosis. They enable an extensive search for problems on the control as well as in the inverter system, and even in the motors themselves. TeleService also features comprehensive remote operation and remote monitoring of the control.*

* 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.

Error diagnostics on the control
The TNC provides integrated functions with which the service technician—in the case of failure—can quickly and simply detect faults in the area of the drives or in the hardware.



Overview
– User functions

User functions	Default	Option	
Short description	•		Basic version: 3 axes plus spindle
		○	4th NC axis plus auxiliary axis
			or
		0-7 77 78	} A total of 14 additional NC axes or 13 additional NC axes plus second spindle
	•		Digital current and speed control
Program entry	•		With smarT.NC, in HEIDENHAIN Klartext conversational format and according to DIN/ISO
		42	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	•		Point filter for smoothing externally created NC programs
Position entry	•		Nominal positions for lines and arcs in Cartesian coordinates or polar coordinates
	•		Incremental or absolute dimensions
	•		Display and entry in mm or inches
	•		Display of the handwheel path during machining with handwheel superimpositioning
Tool compensation	•		Tool radius in the working plane and tool length
	•		Radius compensated contour look ahead for up to 99 blocks (M120)
		9	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)
	•		Entry of cutting speed as alternative 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 a program with graphical support while another program is being run
3-D machining	•		Motion control with smoothed jerk
		9	3-D tool compensation through surface normal vectors
		9	Using the electronic handwheel to change the angle of the swivel head during program run without affecting the position of the tool point (TCPM = Tool Center Point Management)
		9	Keeping the tool normal to the contour
		9	Tool radius compensation normal to the tool direction
		9	Spline interpolation
		9	Manual traverse in the active tool-axis system
		92	Compensating form errors of tools with 3D-ToolComp
Rotary table machining		8	Programming of cylindrical contours as if in two axes
		8	Feed rate in distance per minute
Adaptive Feed Control		45	AFC: Adaptive Feed Control adjusts the contouring feed rate to the current spindle power
Collision monitoring		40	Dynamic Collision Monitoring (DCM)
		40	Graphic depiction of the active collision objects
		40	Fixture monitoring
		40	Tool holder monitoring
		40	DCM in the Test Run mode

Overview
– User functions (continued)

User functions	Default	Option	
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
	•		Via circular arc
FK free contour programming	•		FK free contour programming in HEIDENHAIN conversational format with graphic support for workpiece drawings not dimensioned for NC
Program jumps	•		Subprograms
	•		Program section repeats
	•		Calling any program as a subprogram
Fixed cycles	•		Drilling, conventional and rigid tapping, rectangular and circular pockets
	•		Peck drilling, reaming, boring, counterboring, centering
	•		Milling internal and external threads
	•		Clearing level and oblique surfaces
	•		Multi-operation machining of rectangular and circular pockets, rectangular and circular studs
	•		Multioperation machining of straight and circular slots
	•		Cartesian and polar point patterns
	•		Contour train (incl. 3-D), contour pocket—also with contour-parallel machining
	•		Contour slot with trochoidal milling
		96	Interpolation turning
	•		Engraving cycle: Engrave text or numbers in a straight line or on an arc
	•		OEM cycles (special cycles developed by the machine tool builder) can be integrated
Coordinate transformation			*Programmable:*
	•		Shifting, rotating, mirroring, scaling (axis specific)
		8	Tilting the working plane, PLANE function
			Manually definable:
		44	Global program settings make it possible to manually define shifts, rotations, and handwheel superimpositioning
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 (=, = /, <, >)
	•		Calculating with 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	•		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; context-sensitive
	•		Graphic support for programming cycles
	•		Comment and structure blocks in the NC program
Teach-In	•		Actual positions can be transferred directly into the NC program

User functions	Default	Option	
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 Magnification of details
3-D line graphics	•		For verification of programs created offline
Programming graphics	•		In the Programming and Editing mode, the contours of the NC blocks are drawn on screen while they 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 modes
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 return
Datum management	•		One table per traverse range for storing datums
Datum tables	•		Several datum tables for storing workpiece-related datums
Pallet tables	•		Tool-oriented or workpiece-oriented execution of pallet tables with any number of entries for selection of pallets, NC programs and datums
Touch probe cycles	• • • • • •	 48	Calibrating the touch probe 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. Toggle between showing the measurement results in the coordinate system of the workpiece or the machine Automatic measurement and optimization of machine kinematics
Conversational languages	•		English, German, Czech, French, Italian, Spanish, Portuguese, Swedish, Danish, Finnish, Dutch, Polish, Hungarian, Russian (Cyrillic), Chinese (traditional, simplified), Slovenian, Slovak, Norwegian, Korean, Turkish, Romanian

Overview
– Accessories
– Options

Accessories	
Electronic handwheels	• One **HR 520** portable handwheel or • One **HR 550** portable wireless handwheel or • One **HR 130** panel-mounted handwheel or • Up to three **HR 150** panel-mounted handwheels through an HRA 110 handwheel adapter
Workpiece measurement	• **TS 260:** 3-D touch trigger probe with cable connection or • **TS 460:** 3-D touch trigger probe with radio or infrared transmission or • **TS 444:** 3-D touch trigger probe with infrared transmission or • **TS 740:** 3-D touch trigger probe with infrared transmission
Tool measurement	• **TT 160:** 3-D touch trigger probe or • **TT 460:** 3-D touch trigger probe with radio or infrared transmission • **TL Nano:** Laser system for contact-free tool measurement or • **TL Micro:** Laser system for contact-free tool 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 (operation per PC keyboard—free of charge)
Software for PCs	• **TeleService:** Software for remote diagnosis, monitoring, and operation • **CycleDesign:** Software for creating your own cycle structure • **TNCremo:** Data transmission software—free of charge • **TNCremoPlus:** Software for data transfer with live-screen function

Option number	Option	As of NC software 60642x-	ID	Remark
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	01	367591-01	**Rotary table machining** • Programming of cylindrical contours as if in two axes • Feed rate in distance per minute **Interpolation:** Circular in 3 axes with tilted working plane **Coordinate transformations:** Tilting the working plane, PLANE function
9	Software option 2	01	367590-01	**3-D machining** • 3-D tool compensation through surface normal vectors • Using the electronic handwheel to change the angle of the swivel head during program run without affecting the position of the tool point (TCPM = Tool Center Point Management) • Keeping the tool normal to the contour • Tool radius compensation normal to the tool direction • Manual traverse in the active tool-axis system **Interpolation** • Linear in 5 axes (subject to export permit) • Spline: Execution of splines (third-degree polynomials)
18	HEIDENHAIN DNC	01	526451-01	Communication with external PC applications over COM component

Option number	Option	As of NC software 60642x-	ID	Remark
40	DCM collision	01	526452-01	Dynamic Collision Monitoring (DCM)
42	DXF converter	01	526450-01	Load and convert DXF contours
44	Global PGM Settings	01	576057-01	Global program settings
45	Adaptive Feed Control (AFC)	01	579648-01	Adaptive Feed Control
46	Python OEM process	01	579650-01	Python application on the TNC
48	KinematicsOpt	01	630916-01	Touch probe cycles for automatic measurement of rotary axes
52	KinematicsComp	01	661879-01	Spatial compensation of error in rotary and linear axes
77	4 additional axes	01	634613-01	4 additional control loops
78	8 additional axes	01	634614-01	8 additional control loops
92	3D-ToolComp	01	679678-01	Compensating tool form error
93	Extended tool management	01	679938-01	Extended tool management
96	Adv. spindle interp.	02	751653-01	Advanced functions for an interpolated spindle
98	CAD viewer	02	800553-01	Opening CAD files directly on the iTNC
133	Remote Desktop Manager	02	894423-01	Display and remote operation of external computer units (e.g. a Windows PC)
141	Cross Talk Comp.	02	800542-01	CTC: Compensation of axis couplings
142	Pos. Adapt. Control	02	800544-01	PAC: Position-dependent adaptation of the control parameters
143	Load Adapt. Control Control	02	800545-01	LAC: Load-dependent adaptation of the control parameters
144	Motion Adapt. Control	03	800546-01	MAC: Motion-dependent adaptation of control parameters
145	Active Chatter Control	03	800547-01	ACC: Active suppression of chatter
146	Active Vibration Damping	03	800548-01	AVD: Active suppression of vibration

Overview
– Specifications

Specifications	Default	Option	
Components	•		MC main computer
	•		CC controller unit
	•		TE control panel (suitable for 15.1-inch or 19-inch screens)
	•		BF TFT flat-panel display with soft keys (15.1-inch or 19-inch)
Operating system	•		HEROS real-time operating system for machine control
NC program memory	•		RAM memory: ≥ 2 GB
	•		For version with HDR hard disk: approx. 144 GB For version with SSDR solid state disk: approx. 21 GB
Input resolution and display step	•		Linear axes: up to 0.1 µm
	•		Angular axes: to 0.0001°
Input range	•		Maximum 99 999.999 mm (3.937 inches) or 99 999.999°
Interpolation	•		Linear in 4 axes
		9	Linear in 5 axes (subject to export permit)
	•		Circular in 2 axes
		8	Circular in 3 axes with tilted working plane
	•		Helical: superimposition of circular and straight paths
		9	Spline: Execution of splines (third-degree polynomials)
Block processing time	•		0.5 ms (3-D straight line without radius compensation)
Axis feedback control	•		Position loop resolution: Signal period of the position encoder/1024
	•		Cycle time of position controller: 200 µs
	•		Cycle time of speed controller: 200 µs
	•		Cycle time of current controller: minimum 50 µs
Range of traverse	•		Maximum 100 m (3937 inches)
Spindle speed	•		Maximum 60 000 rpm (with 2 pole pairs)
Error compensation	•		Linear and nonlinear axis error, backlash, reversal peaks during circular movements, reversal error, thermal expansion
	•		Static friction, sliding friction
Data interfaces	•		One each RS-232-C/V.24 and RS-422/V.11, max. 115 kbit/s
	•		Extended data interface with LSV2 protocol for remote operation of the iTNC 530 over the data interface with the HEIDENHAIN software TNCremo or TNCremoPlus
	•		2 x Gigabit Ethernet interface 1000BASE-T
	•		4 x USB (1 x front USB 2.0; 3 x back panel USB 3.0)
		18	HEIDENHAIN-DNC for communication between a Windows application and TNC (DCOM interface)
Diagnostics	•		Fast and simple troubleshooting through integrated diagnostic aids
Ambient temperature	•		Operation: 0 °C to +50 °C
	•		Storage: –20 °C to +60 °C

– Comparison of controls

Comparison of controls	**TNC 620** NC-SW 81760x-03	**TNC 640** NC-SW 34059x-06	**iTNC 530** NC SW 60642x-04
Area of application	**Standard milling**	**High-end milling/turning**	**High-end milling**
• Simple machining centers (up to 5 axes + 1 spindle)	●	●	●
• Machine tools/machining centers (up to 18 axes + 2 spindles)	–	●	●
• Milling/turning operations (up to 18 axes + 2 spindles)	–	Option	–
Program entry			
• HEIDENHAIN Klartext conversational	●	●	●
• According to ISO	●	●	●
• DXF converter	Option	Option	Option
• CAD viewer	●	●	Option
• FK free contour programming	Option	●	●
• Extended milling and drilling cycles	Option	●	●
• Turning cycles	–	Option	–
NC program memory	1.8 GB	> 21 GB	> 21 GB
5-axis and high-speed machining	Option	Option	Option
Block processing time	1.5 ms	0.5 ms	0.5 ms
Input resolution and display step (standard/option)	0.1 µm/0.01 µm	0.1 µm/0.01 µm	0.1 µm/–
New design of the screen and keyboard	15-inch screen	15/19-inch screen	15/19-inch screen
Optimized user interface	●	●	–
Adaptive Feed Control (AFC)	–	Option	Option
Active Chatter Control (ACC)	Option	Option	Option
Dynamic Collision Monitoring (DCM)	–	Option	Option
KinematicsOpt	Option	Option	Option
KinematicsComp	–	Option	Option
Touch probe cycles	Option	●	●
Pallet editor	Option	●	●
Parallel axis function	●	●	–

● Standard
– Not available

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 8669 31-0
FAX +49 8669 32-5061
E-mail: info@heidenhain.de

www.heidenhain.de

Vollständige und weitere Adressen siehe www.heidenhain.de
For complete and further addresses see www.heidenhain.de

DE **HEIDENHAIN Vertrieb Deutschland**
83301 Traunreut, Deutschland
☏ 08669 31-3132
FAX 08669 32-3132
E-Mail: hd@heidenhain.de

HEIDENHAIN Technisches Büro Nord
12681 Berlin, Deutschland
☏ 030 54705-240

HEIDENHAIN Technisches Büro Mitte
07751 Jena, Deutschland
☏ 03641 4728-250

HEIDENHAIN Technisches Büro West
44379 Dortmund, Deutschland
☏ 0231 618083-0

HEIDENHAIN Technisches Büro Südwest
70771 Leinfelden-Echterdingen, Deutschland
☏ 0711 993395-0

HEIDENHAIN Technisches Büro Südost
83301 Traunreut, Deutschland
☏ 08669 31-1345

AR **NAKASE SRL.**
B1653AOX Villa Ballester, Argentina
www.heidenhain.com.ar

AT **HEIDENHAIN Techn. Büro Österreich**
83301 Traunreut, Germany
www.heidenhain.de

AU **FCR Motion Technology Pty. Ltd**
Laverton North 3026, Australia
E-mail: vicsales@fcrmotion.com

BE **HEIDENHAIN NV/SA**
1760 Roosdaal, Belgium
www.heidenhain.be

BG **ESD Bulgaria Ltd.**
Sofia 1172, Bulgaria
www.esd.bg

BR **DIADUR Indústria e Comércio Ltda.**
04763-070 – São Paulo – SP, Brazil
www.heidenhain.com.br

BY **GERTNER Service GmbH**
220026 Minsk, Belarus
www.heidenhain.by

CA **HEIDENHAIN CORPORATION**
Mississauga, OntarioL5T2N2, Canada
www.heidenhain.com

CH **HEIDENHAIN (SCHWEIZ) AG**
8603 Schwerzenbach, Switzerland
www.heidenhain.ch

CN **DR. JOHANNES HEIDENHAIN (CHINA) Co., Ltd.**
Beijing 101312, China
www.heidenhain.com.cn

CZ **HEIDENHAIN s.r.o.**
102 00 Praha 10, Czech Republic
www.heidenhain.cz

DK **TP TEKNIK A/S**
2670 Greve, Denmark
www.tp-gruppen.dk

ES **FARRESA ELECTRONICA S.A.**
08028 Barcelona, Spain
www.farresa.es

FI **HEIDENHAIN Scandinavia AB**
01740 Vantaa, Finland
www.heidenhain.fi

FR **HEIDENHAIN FRANCE sarl**
92310 Sèvres, France
www.heidenhain.fr

GB **HEIDENHAIN (G.B.) Limited**
Burgess Hill RH15 9RD, United Kingdom
www.heidenhain.co.uk

GR **MB Milionis Vassilis**
17341 Athens, Greece
www.heidenhain.gr

HK **HEIDENHAIN LTD**
Kowloon, Hong Kong
E-mail: sales@heidenhain.com.hk

HR Croatia → **SL**

HU **HEIDENHAIN Kereskedelmi Képviselet**
1239 Budapest, Hungary
www.heidenhain.hu

ID **PT Servitama Era Toolsindo**
Jakarta 13930, Indonesia
E-mail: ptset@group.gts.co.id

IL **NEUMO VARGUS MARKETING LTD.**
Tel Aviv 61570, Israel
E-mail: neumo@neumo-vargus.co.il

IN **HEIDENHAIN Optics & Electronics India Private Limited**
Chetpet, Chennai 600 031, India
www.heidenhain.in

IT **HEIDENHAIN ITALIANA S.r.l.**
20128 Milano, Italy
www.heidenhain.it

JP **HEIDENHAIN K.K.**
Tokyo 102-0083, Japan
www.heidenhain.co.jp

KR **HEIDENHAIN Korea LTD.**
Gasan-Dong, Seoul, Korea 153-782
www.heidenhain.co.kr

MX **HEIDENHAIN CORPORATION MEXICO**
20290 Aguascalientes, AGS., Mexico
E-mail: info@heidenhain.com

MY **ISOSERVE SDN. BHD.**
43200 Balakong, Selangor
E-mail: sales@isoserve.com.my

NL **HEIDENHAIN NEDERLAND B.V.**
6716 BM Ede, Netherlands
www.heidenhain.nl

NO **HEIDENHAIN Scandinavia AB**
7300 Orkanger, Norway
www.heidenhain.no

PH **Machinebanks` Corporation**
Quezon City, Philippines 1113
E-mail: info@machinebanks.com

PL **APS**
02-384 Warszawa, Poland
www.heidenhain.pl

PT **FARRESA ELECTRÓNICA, LDA.**
4470 - 177 Maia, Portugal
www.farresa.pt

RO **HEIDENHAIN Reprezentanţă Romania**
Braşov, 500407, Romania
www.heidenhain.ro

RS Serbia → **BG**

RU **OOO HEIDENHAIN**
115172 Moscow, Russia
www.heidenhain.ru

SE **HEIDENHAIN Scandinavia AB**
12739 Skärholmen, Sweden
www.heidenhain.se

SG **HEIDENHAIN PACIFIC PTE LTD.**
Singapore 408593
www.heidenhain.com.sg

SK **KOPRETINA TN s.r.o.**
91101 Trencin, Slovakia
www.kopretina.sk

SL **NAVO d.o.o.**
2000 Maribor, Slovenia
www.heidenhain.si

TH **HEIDENHAIN (THAILAND) LTD**
Bangkok 10250, Thailand
www.heidenhain.co.th

TR **T&M Mühendislik San. ve Tic. LTD. ŞTİ.**
34775 Y. Dudullu –
Ümraniye-Istanbul, Turkey
www.heidenhain.com.tr

TW **HEIDENHAIN Co., Ltd.**
Taichung 40768, Taiwan R.O.C.
www.heidenhain.com.tw

UA **Gertner Service GmbH Büro Kiev**
01133 Kiev, Ukraine
www.heidenhain.ua

US **HEIDENHAIN CORPORATION**
Schaumburg, IL 60173-5337, USA
www.heidenhain.com

VE **Maquinaria Diekmann S.A.**
Caracas, 1040-A, Venezuela
E-mail: purchase@diekmann.com.ve

VN **AMS Co. Ltd**
HCM City, Vietnam
E-mail: davidgoh@amsvn.com

ZA **MAFEMA SALES SERVICES C.C.**
Midrand 1685, South Africa
www.heidenhain.co.za

Zum Abheften hier falzen! / *Fold here for filing!*

895822-25 · 10 · 6/2016 · H · Printed in Germany