HEIDENHAIN


# TNC 640

Contouring Control for
Machining Centers and
Milling/Turning Machines

**September 2011**

# Contents

The TNC 640...

# **Versatile**
## – The TNC Contouring Control for Milling and Milling/Turning Machines

For more than 35 years, TNC controls from HEIDENHAIN have been proving themselves in daily use on milling, drilling and boring machines, as well as machining centers. While the controls have undergone continuous development during this period, the basic operational technique has remained the same.

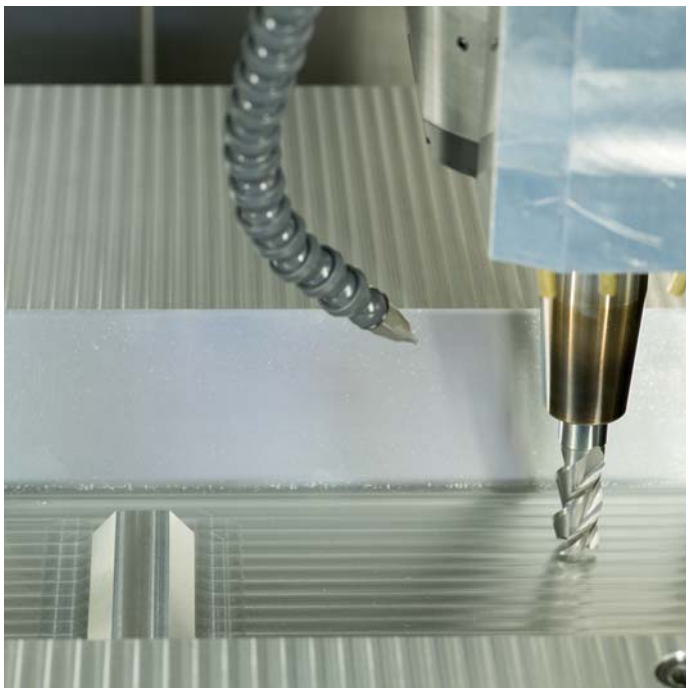
You will find also these principles implemented in the TNC 640, the HEIDENHAIN contouring control for milling and milling/turning operations: shop-oriented programmability with graphic support, many field-proven cycles and an operational design you'll recognize from other HEIDENHAIN controls.

**Shop-oriented programming**
You program conventional milling and drilling operations, and with the TNC 640 also turning operations, yourself at the machine, in plain language dialog—the workshop-oriented programming language from HEIDENHAIN. The TNC 640 provides you with optimum support with practical prompts, questions and expressive graphical aids—for turning operations, too.

Standard operations and even complex applications are on call as a large variety of real-world machining cycles or coordinate transformations.

**Simple operation**
For simple work, such as face milling or face turning, you need not write a program on the TNC 640. It is just as easy to operate the machine manually by pressing the axis keys or—for maximum sensitivity—using the electronic handwheel.

**Offline program creation**
The TNC 640 can be programmed remotely just as well. Your Fast Ethernet interface guarantees very short transfer times, even of long programs.



**Universally applicable**
The TNC 640 is particularly capable in operations on milling/turning machines. The TNC 640 is also intended for HSC and 5-axis operations on machines with up to 18 axes.

The TNC 640 is especially attractive for the following areas of application:

**Milling/turning machines**
- Simple, program-controlled switchover between milling and turning
- Comprehensive turning cycle package
- Constant surface speed
- Tool-tip radius compensation

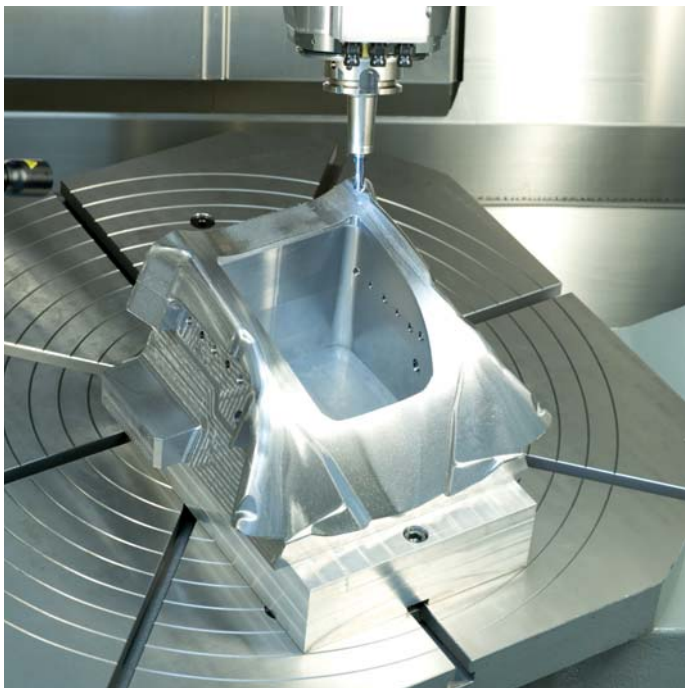
**Universal milling machine**
- Shop-floor programming in HEIDENHAIN conversational format
- Fast presetting with a HEIDENHAIN touch probe
- Electronic handwheel

**High speed milling**
- Fast block processing
- Short control-loop cycle time
- Motion control low in jerk
- High spindle speed
- Fast data transfer

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

- **Five-axis machining with swivel head and rotary table**
- Tilting the working plane
- Cylindrical surface machining
- Tool Center Point Management (TCPM)
- 3-D tool compensation
- Fast execution through short block processing times

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



# Well Designed and User Friendly
## – The TNC 640 in Dialog with the User

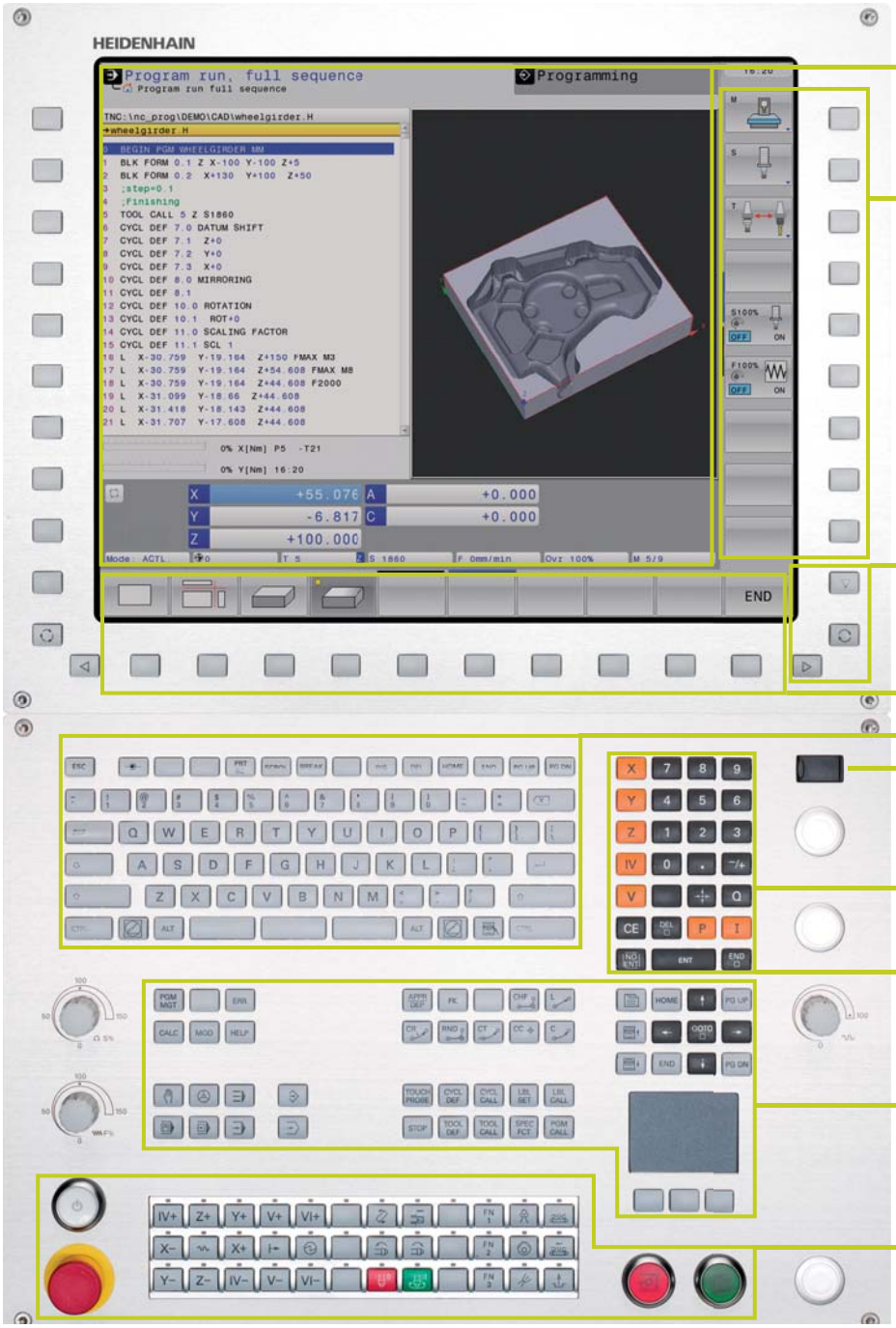
**The screen**
The large TFT 19-inch color monitor shows a clear overview of all relevant information for programming, operating and inspecting the machine tool and control, such as program blocks, comments and error messages. More information is provided through graphic support during program entry, 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 on tool position, the current program, active cycles and coordinate transformations, and other data. The TNC 640 even shows the current machining time.

**The keyboard**
As with all TNCs from HEIDENHAIN, the keyboard is tailored to the programming process. The well-thought-out arrangement of keys in a clear division into **functions groups** programming modes, machining modes, management/TNC functions and navigation support 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), mode of operation and for shifting 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 TNC 640 features a special protection coating and is therefore highly resistant to soiling and wear.

**Smooth**
The rectangular, slightly rounded keys are pleasant for your fingers and reliable in operation. Their inscriptions stay free of wear even under extreme workshop conditions

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

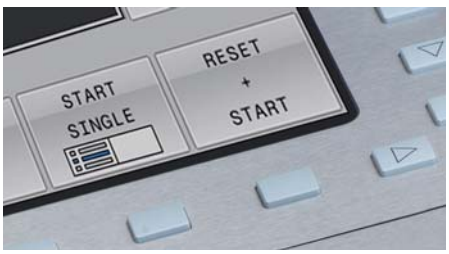
**Reliable**
The elevated key bed of the machine operating panel prevents accidental actuation. LEDs serve for status display of each key by clearly indicating the active machine functions.

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

**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.







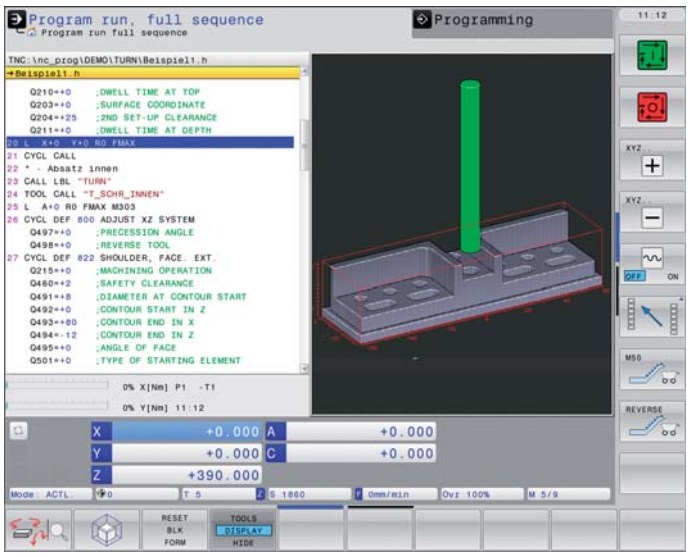
# Well Designed and User Friendly
## – The Functional User Interface

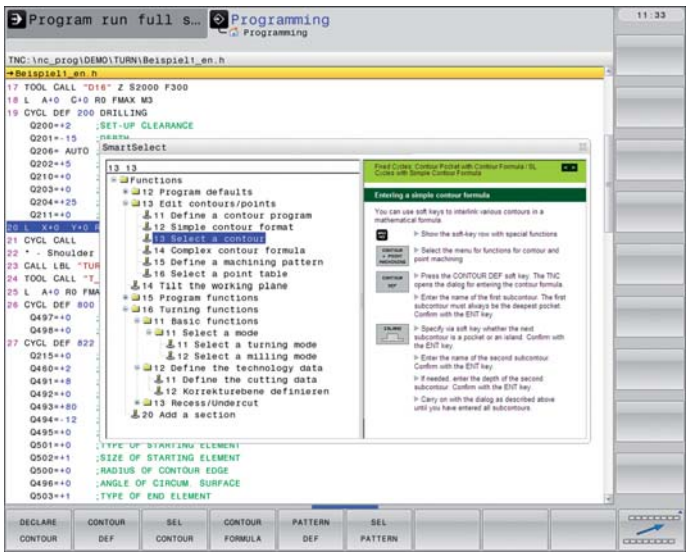
The combination of the straightforward and ergonomically correct keyboard and the well-designed screen layout are the essence of reliable and fatigue-free operation. These are principles that HEIDENHAIN controls have always embodied. However, the TNC 640 also offers a number of features that make working with the control even easier and user-friendlier than ever.

**Attractive view**
The user interface of the TNC 640 has a modern appearance, with lightly rounded forms, color gradients and a homogeneously designed font. The individual screen areas are clearly distinct, the two main operating modes Machining and Programming and Editing are indicated by dialog texts and the respective operating mode signals.

To better distinguish between the priority of error messages, the TNC 640 displays them in color-coded categories. A color-coded warning triangle is also displayed.

**Fast function overview**
With **smartSelect** you enjoy dialog guidance for selecting functions quickly and easily that up to now were accessible only through the soft-key structure. As soon as you open smartSelect, it displays a tree structure with all subordinate functions that can be defined in the control's current condition. Moreover, in the right part of the smartSelect window, the TNC displays the integrated help. With the cursor or a mouse click, you immediately access detailed information on the respective function. smartSelect enables you to define fixed cycles, touch probe cycles, special functions (SPEC FCT), and quickly access the parameter programming.



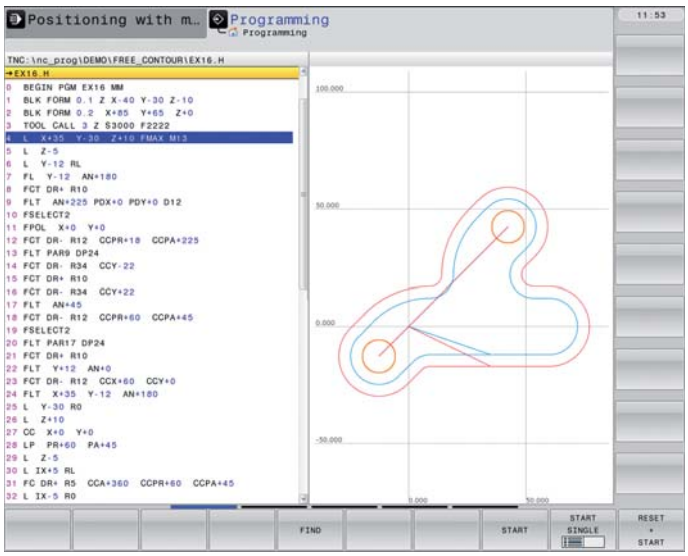



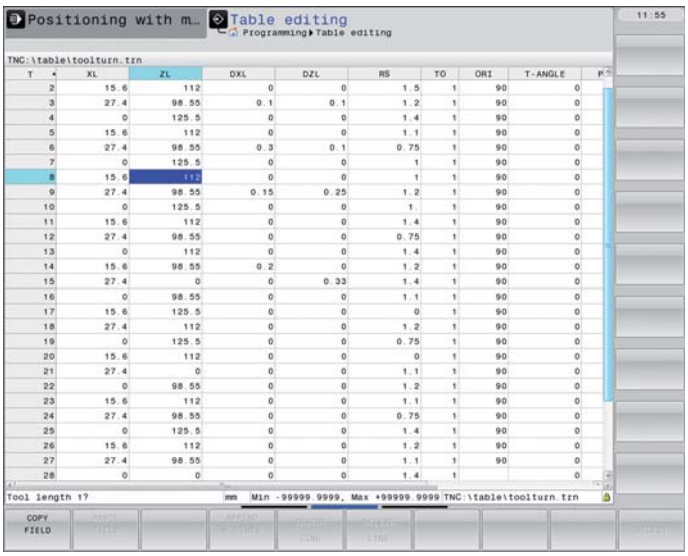
**Color-structured programs**
The content of a program line can be quite comprehensive: line number, program function, input values, comment. To help you always find your way even in complex programs, the individual program elements on the TNC 640 are shown in different colors. The color syntax highlighting improves your overview when editing NC programs. It enables you to see at a glance, for example, where the editable input values are.

**Uniform table editor**
Regardless of which table you are editing—whether the tool table, preset table or pallet table—the function and operation of the table editor are always the same.

**Info line**
In the info line, the TNC 640 shows the respective submode condition and helps you to orient yourself. The function is comparable with the history function in web browsers.

**MOD function**
The additional mode MOD was thoroughly overhauled and now offers a myriad of possible settings in a standardized layout regardless of the operating mode.



# Multi-Operation Machining
## – Milling and Turning on One Machine (Option)

Does your workpiece, after complex milling operations, also need to be set up on a lathe for several working steps? Do you have to plan for machine capacity, make tools, set up and fix the workpiece, and measure the finished part? The TNC 640 helps you to save time: on a milling/turning machine with TNC 640 you machine the complete workpiece on one machine: milling, turning, milling, in whatever sequence. After performing all operations on one machine, you measure the finished workpiece with a HEIDENHAIN touch probe.

The TNC 640 offers you powerful functions that enable you to switch the NC program as desired between turning and milling under program control. This enables you to decide with complete freedom how and when you want to combine the two machining methods. And of course, the operations switch back and forth regardless of the machine and its axis configuration. During switchover, the TNC 640 assumes all necessary internal changes, such as switching to diameter display, setting the datum in the center of the rotary table, and even machine-dependent functions such as clamping the tool spindle.*

**Programming as accustomed**
You can program the turning operations—as always—conveniently under dialog guidance in HEIDENHAIN plain-language. Besides the standard path functions you can also use FK free contour programming to easily create contour elements not otherwise dimensioned for NC. Beyond this, you also have the contour elements recessing and undercutting for turning operations, which are supported by expressive help illustrations.

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

**Cycles for milling and turning**
HEIDENHAIN controls have always been known for their comprehensive and technologically sophisticated package of cycles. Frequently recurring operations that comprise several steps are also stored in the TNC 640 as cycles. You program them under conversational guidance and are supported by valuable help graphics that clearly illustrate the required input parameters. Besides the well known TNC milling and drilling cycles, the TNC 640 also offers a wide variety of turning cycles, for example for roughing, finishing, recessing and thread turning. The field-proven HEIDENHAIN lathe controls provided the software basis for the turning functions. They enable you to very easily program even complex turning operations at the machine.

In the more sophisticated contour turning cycles, the TNC 640 uses the same techniques as are used for milling. Here, too, there is no need for the TNC programmer to learn new ways of programming—he can continue to rely on what he already knows and quickly find his way into the world of turning on a milling machine.

# Quick and Reliable Machining with High Contour Fidelity
– Uniformly Digital Control Design

Thanks to its digital design, the TNC 640 has control over the machine's entire drive system. Not only does the field-proven digital drive technology from HEIDENHAIN make high contour fidelity and rapid machining at high speeds possible, but also all control components of the TNC 640 are connected via digital interfaces.

**Digital drive technology**
The position controller, speed controller and, if required, the current controller are integrated in the TNC 640. The digital motor control makes it possible to attain very high feed rates. While interpolating simultaneously in up to five axes, the TNC 640 reaches the required cutting speeds by digitally controlling spindle speeds up to **60 000 rpm**.

**High contour fidelity**
The TNC 640 dynamically calculates the contour in advance. This enables it to adapt the axis velocities early enough to the contour transitions. It controls the axes with special algorithms that ensure path control with the required limits to velocity and acceleration.

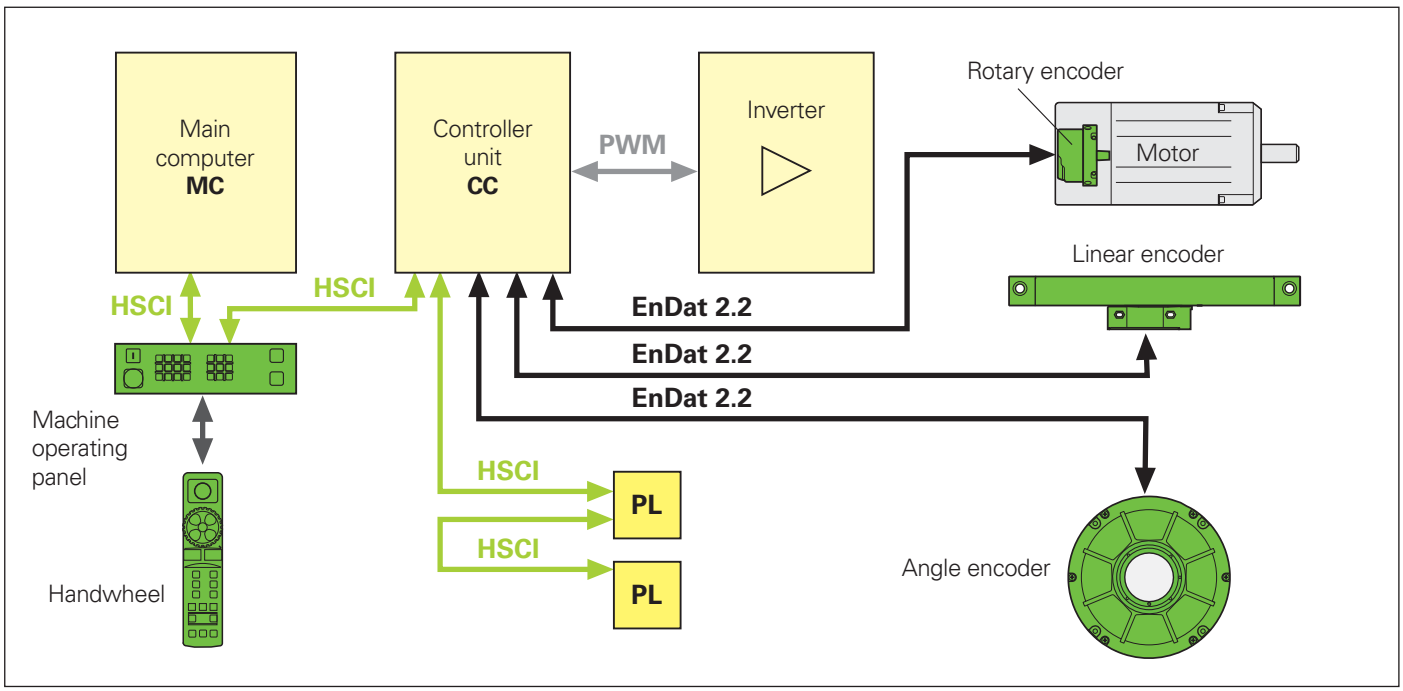
**Highest contour fidelity and surface quality**
Special filters specifically suppress machine-specific natural vibration. The desired accuracy is attained with the very high surface quality. Thanks to the short block processing time of 0.5 ms, even highly accurate contours with very high resolution are not a problem.

**Fast machining at specified accuracy**
You as user specify the accuracy of the machine contour—apart from the NC program. You simply enter in the control through a cycle the maximum permissible deviations from the ideal contour. The TNC 640 automatically adapts the machining to the tolerance that you define. No contour damage occurs with this method.

**High availability**
In the uniformly digital control concept of the TNC 640, all components are connected to each other via purely digital interfaces: The control components are connected via HSCI (HEIDENHAIN Serial Controller Interface), the real-time protocol from HEIDENHAIN for Fast Ethernet, and the encoders are connected via EnDat2.2, 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—from the main computer to the encoder.

The uniformly digital design from HEIDENHAIN guarantees not just very high accuracy and surface quality, but high machining speeds as well—regardless of whether you are milling or turning.



# Machining with Five Axes
## – Swivel Head and Rotary Table Controlled by the TNC

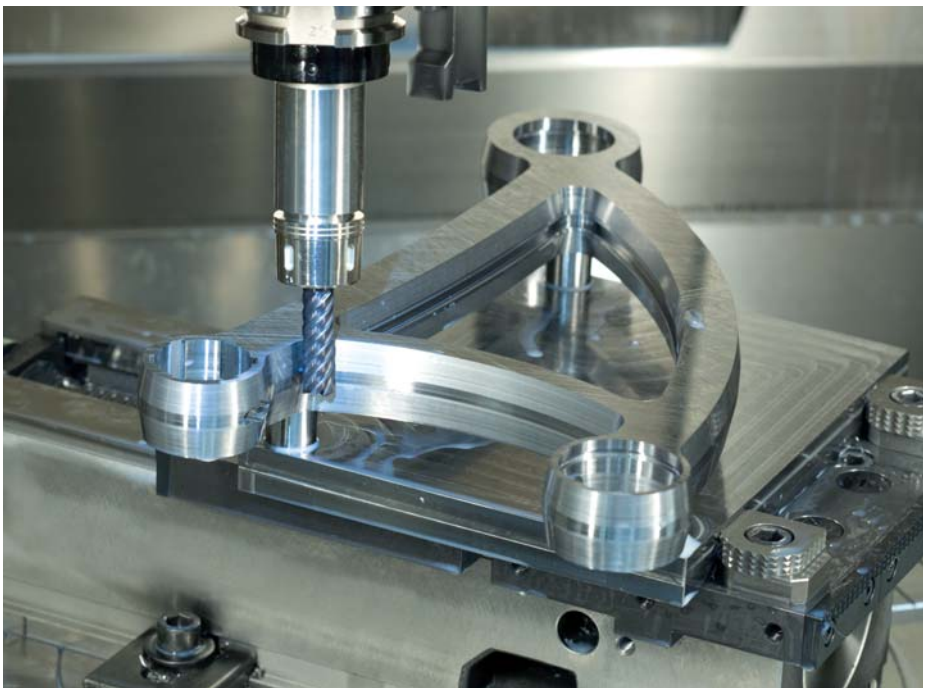
Many five-axis operations that at first glance may seem very complex can be reduced to conventional 2-D movements that are simply tilted about one or more rotary axes or wrapped onto a cylindrical surface. The TNC supports you with application-oriented functions to help you write and edit such programs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

**Tilting the working plane* (option 8)**
Programs for contours and holes on inclined surfaces are often very complex and require time-consuming computing and programming work. Here the TNC 640 helps you to save a great deal of programming time.

You program the part as usual in the working plane (e.g. the X/Y plane), but it is machined in a plane that is rotated in one or more axes about the main plane.

The PLANE feature makes it easy to define a tilted working plane: You can specify tilted working planes in seven different ways, depending on the information on the workpiece drawing. Clearly arranged support graphics assist you during input.

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



# – Optimal Tool Guidance

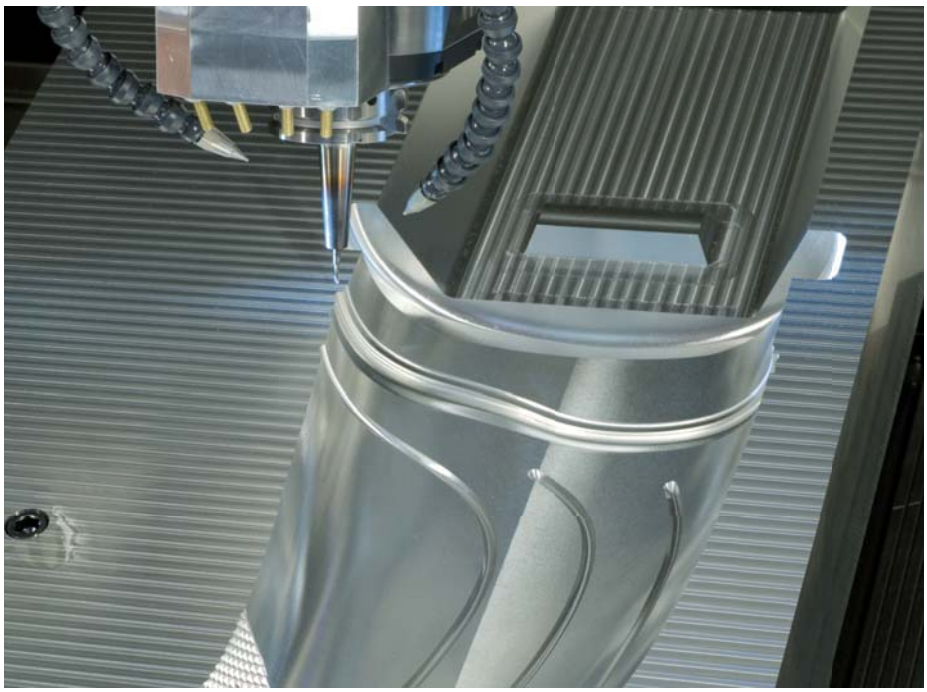
**Five-axis machining (option 9)**
When machining with five axes (three linear axes and two tilting axes)*, the tool can stay perpendicular, or if desired, inclined at a predetermined angle to the workpiece surface. To generate an NC program, the CAD system only needs to calculate the points on the workpiece surface and the angles of the tilting axes. The TNC 640 automatically compensates the machine's geometry* and the tool length, and it compensates the tool radius in 3-D. Also, it calculates the feed rate so that it remains constant at the tool tip.

**Feed rate for rotary tables in mm/min* (option 8)**
In the standard version, the feed rate of rotary axes is programmed in degrees/minute. However, the TNC 640 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.

**Cylindrical surface machining* (option 8)**
With the TNC 640 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 TNC 640 then executes the operation on the surface of the cylinder.

The TNC 640 features three 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

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



# Minimize Setup Times
## – The TNC 640 Makes Setup Easy

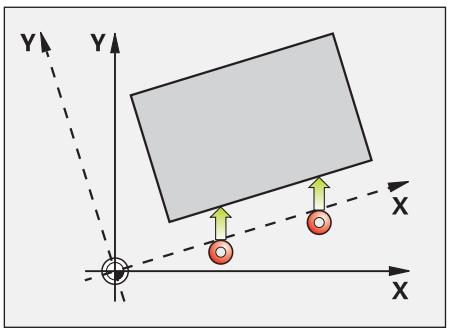
Before you can begin machining, you must first clamp the tool and set up the machine, find the position and orientation of the workpiece on the machine, and set the workpiece reference point. This is a time-consuming but indispensable procedure. After all, any error directly reduces the machining accuracy.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

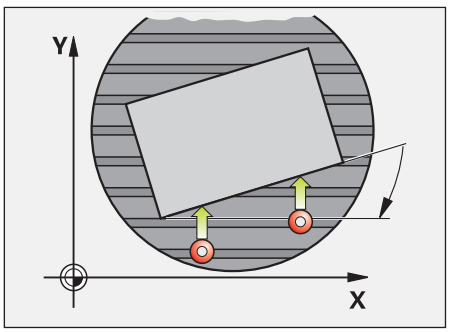
The TNC 640 features application-oriented, real-world setup functions. They support the user, help to reduce non-productive time, and make overnight, unattended production possible. Together with the **touch probes,** the TNC 640 offers numerous probing cycles for automatic alignment of the workpieces, 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 35). 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 touch probes (see page 32) and the probing functions of the TNC 640, you can forgo any tedious manual alignment of the workpiece:
- Clamp the workpiece in any position.
- The touch probe ascertains the workpiece misalignment by probing a surface.
- The TNC 640 compensates the misalignment with a "basic rotation," which means that in the NC program the part is rotated by the measured misalignment.

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



**Workpiece presetting**
You can use a reference point to assign a defined value in the TNC display to any workpiece position. Finding this point quickly and reliably reduces nonproductive time and increases machining accuracy. The TNC 640 features probing cycles for automatic presetting. Once found, you can save reference points:

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

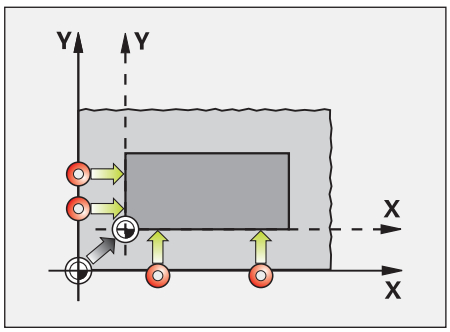
**Preset table: The TNC's central reference point management**
The preset table makes flexible machining, shorter setup times and increased productivity possible. In other words, it makes it much easier to set up the machine.

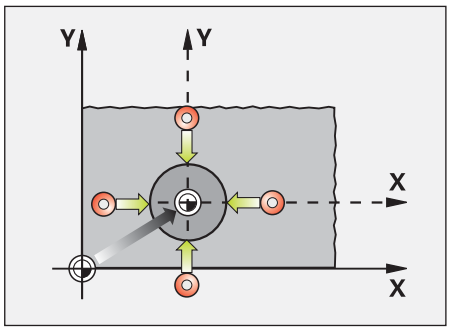
In the preset table you can save **any number of reference points** and assign an individual basic rotation to each one.

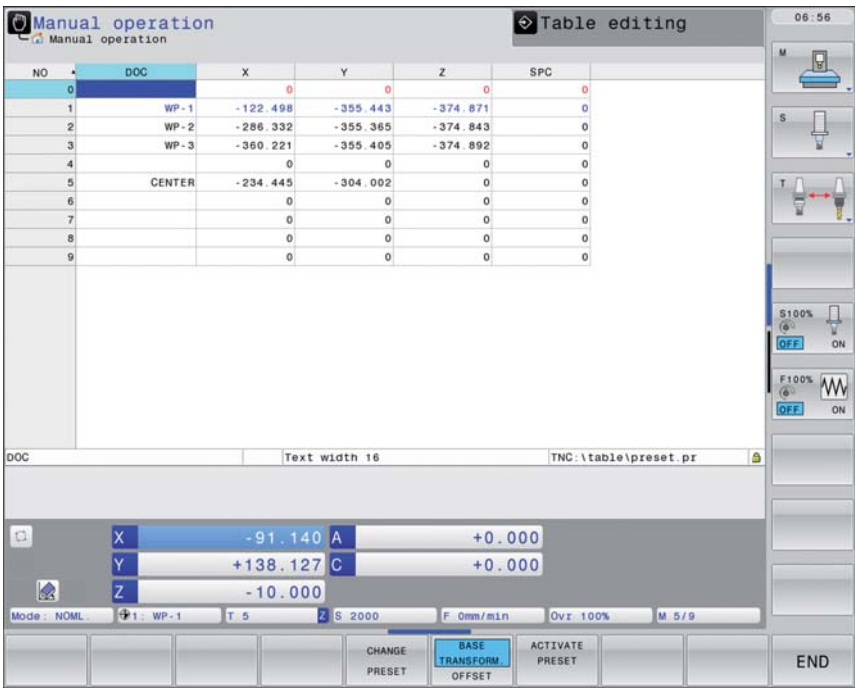
There are three ways to save reference points in the preset table:

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

**Workpiece presetting**
At a corner, for example, or in the center of a circular stud



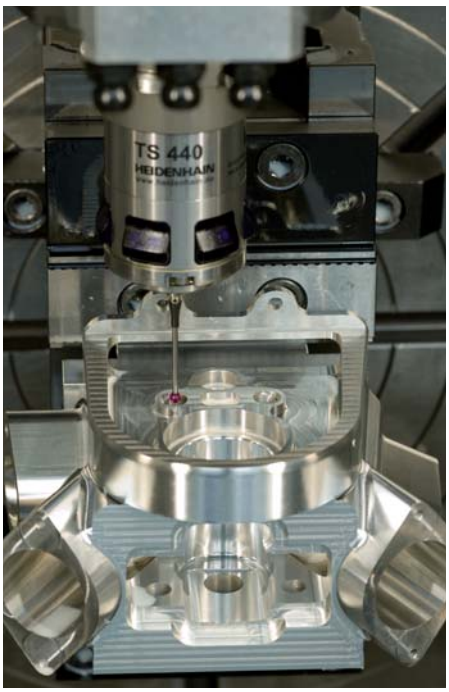
# Automated Machining
## – The TNC 640 Measures, Manages and Communicates

The difference in requirements placed on the classical machine for tool and mold-making and machining centers are becoming ever less distinct. Of course, the TNC 640 is capable of controlling automated manufacturing 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.

**Inspecting workpieces for proper machining and dimensional accuracy**
The TNC 640 features a number of measuring cycles for checking the geometry of the machined workpieces. To run the measuring cycles, you insert a touch probe from HEIDENHAIN (see page 32) into the spindle in place of a tool. This enables you to

- 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 the workpiece geometry and sort the parts,
- log measured data, and
- ascertain the machining error trend.

**Milling cutter measurement and automatic compensation of tool data**
Together with the TT 140 touch probe for tool measurement (see page 33) the TNC 640 can automatically measure milling cutters while they are in the machine. The TNC 640 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 TNC 640 locks the tool and automatically inserts a replacement tool.

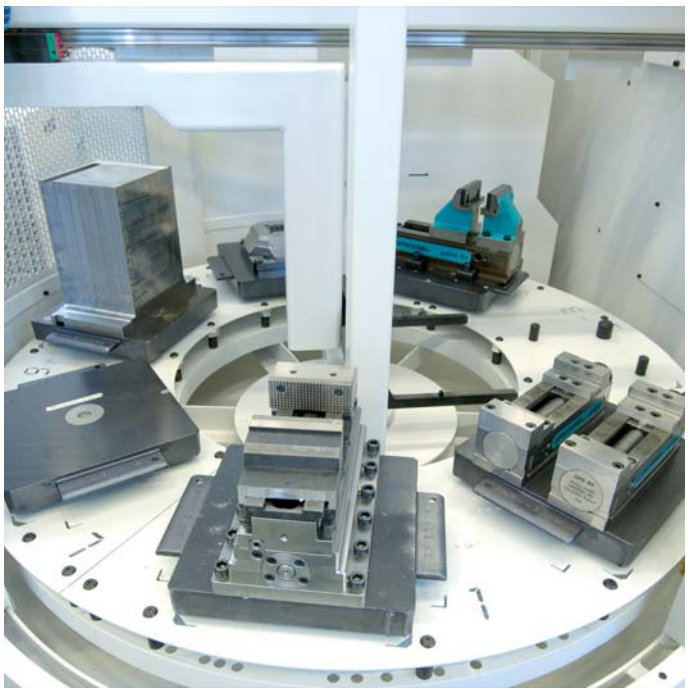



**Tool management**
For machining centers with automatic tool changers, the TNC 640 offers a central tool memory for any number of milling and turning tools. The tool memory is a freely configurable file and can therefore be optimally fitted to your needs. You can even have the TNC 640 manage your tool 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 TNC 640 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 TNC 640 automatically calls the correct part program. This permits automatic machining of a variety of parts in any sequence.

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



# **Programming, Editing, Testing**
## – The TNC 640 Opens Endless Possibilities

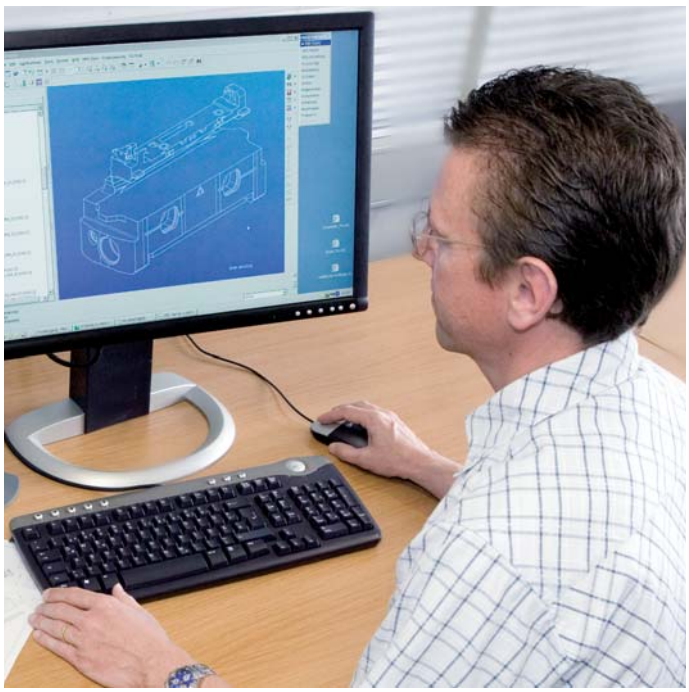
The TNC 640 is just as universal in application as it is flexible in machining and programming.

**Positioning with manual data input**
You can start working with the TNC 640 even before writing a complete part program. Simply machine a part step by step—switching as you want between manual operation and automatic positioning.

**Programming at the machine**
HEIDENHAIN controls are workshop oriented, which means that they were conceived for programming right at the machine. With conversational programming you can forget about memorizing G codes. Instead you use dedicated keys and soft keys to program line segments, circular arcs and cycles. With a keystroke, you initiate a HEIDENHAIN plain language dialog, and the TNC begins immediately to support you actively in your work. Unambiguous questions and prompts help you enter all the required information.

Whether plain-language prompts, dialog guidance, programming steps or soft keys, all texts are available in numerous languages.

**Creating programs offline**
The TNC 640 is also well equipped for offline programming. It can be integrated through its interfaces into networks and connected with programming stations or other data storage devices. The TNC 640 can also run programs that were written in DIN/ISO format.



# – Graphic Support in Any Situation

**Interactive programming graphics**
The two-dimensional programming graphics give you additional security: while you are programming, the TNC 640 draws every entered traverse command on the screen.

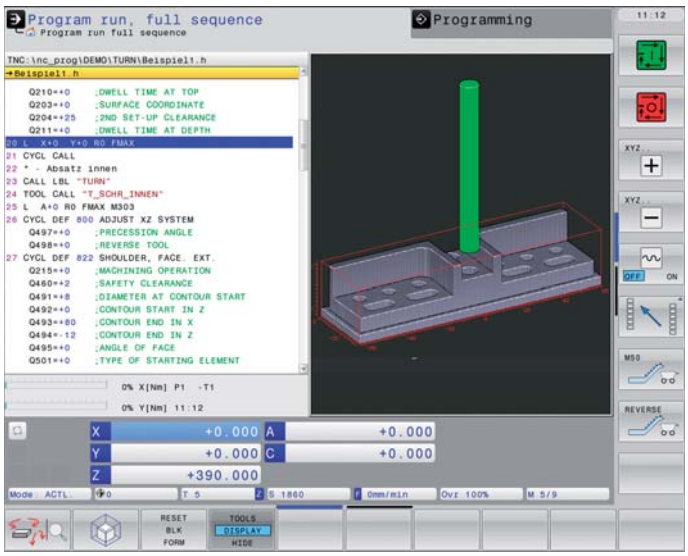
**Test run graphics**
To play it safe before running a program, the TNC 640 can graphically simulate the milling of the workpiece. The TNC 640 can display the simulation in the following ways:

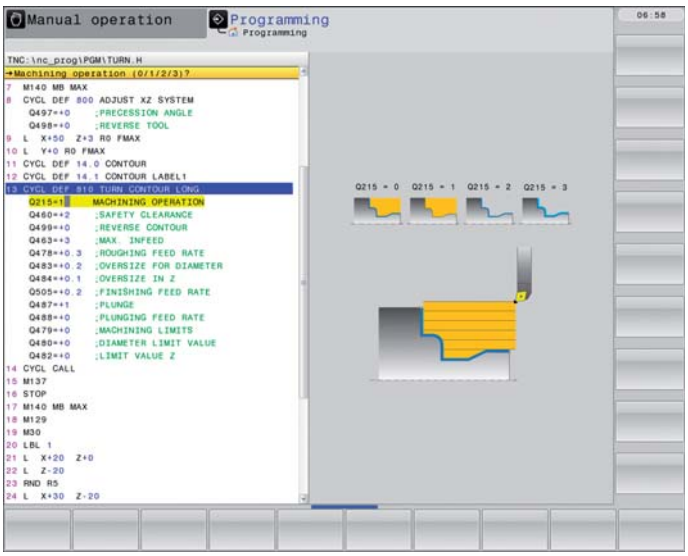
- In a plan view with different shades of depth
- In three planes (as in the workpiece drawing)
- In a solid model, 3-D view

Details can be displayed in magnification. In addition, the TNC 640 indicates the calculated machining time in hours, minutes and seconds.

**Program-run graphics**
On the TNC 640, you can run the program-run graphics or verification graphics even while the workpiece is being machined. Also, it shows a real-time graphic of the milling 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 milling.

**Help graphics**
During cycle programming in the plain-language dialog, the TNC shows a separate illustration for each parameter. This makes it easier to understand the function and accelerates programming. The TNC 640 also supports you with useful help graphics when programming the PLANE function and the contour elements for turning.



# Programming in the Workshop
## – Straightforward Function Keys for Complex Contours

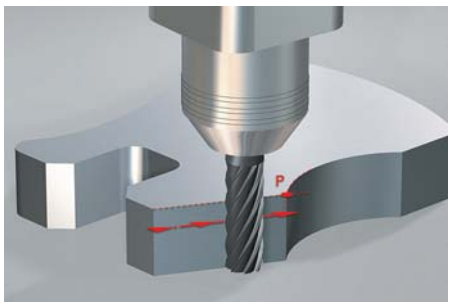
### Programming 2-D Contours

Two-dimensional contours are the daily bread of the modern machine shop. The TNC 640 offers a variety of possibilities here. And—regardless of whether you are programming a milling or turning contour—you always use the same tools. For you this means that you do not have to relearn, just continue to program as usual.

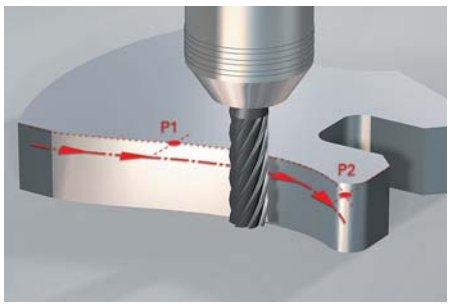
**Programming with path function keys**
If contours are dimensioned for NC, which means that the end points are specified in Cartesian or polar coordinates, then you can program them directly with the path function keys.

**Straight and circular contour elements**
To program a line segment, for example, simply press the key for linear traverse. The TNC 640 asks for all information required for a complete programming block, such as target coordinates, feed rate, tool 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.

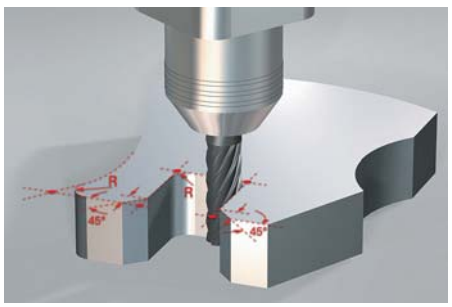
The TNC 640 can look ahead over a radius-compensated contour for up to 99 blocks to watch for back cutting and avoid contour damage such as can occur when roughing a contour with a large tool.




L
Straight line defined by its end point

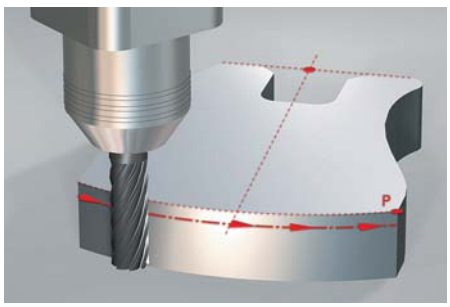



CT
Circular path defined by its end point, with a smooth (tangential) departure from the previous contour element

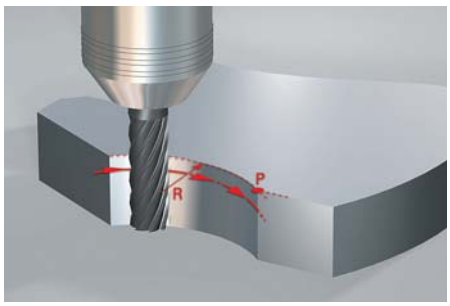



RND Rounding: circular path defined by radius and corner point, with a smooth (tangential) transition to its adjoining contour elements

CHF Chamfer defined by the corner point and chamfer length




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



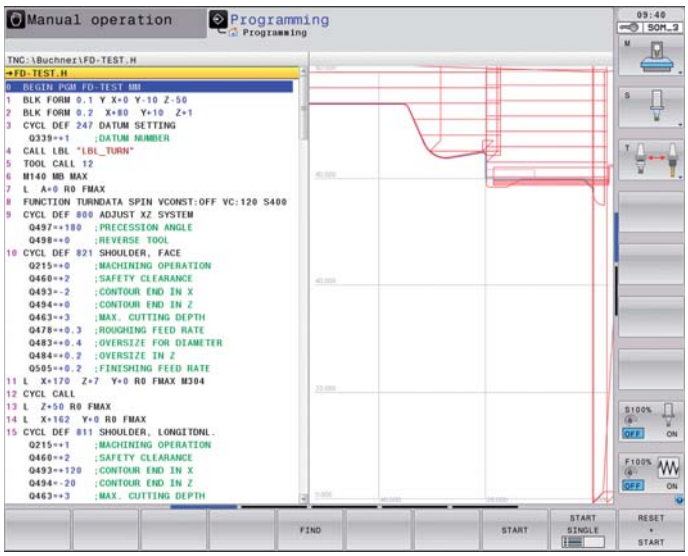

CR
Circular path defined by its radius, end point and rotational direction

# – Programming Contours Unconventionally

**FK free contour programming**
Not all workpieces are dimensioned for conventional NC programming. Thanks to FK, the control'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 TNC 640 programming graphics present the possible variants for your selection.

**Lathe-specific contour elements (option)**
The TNC 640 provides special contour elements to enable you to define recesses and undercuts. Axial or radial recesses can be defined over the GRV (groove) function. With the aid of dialog guidance and help graphics, you use the proper parameters to define the desired recess.

Undercuts can be defined using the UDC function. Here the forms E, F, H, K and U are available as well as thread undercuts.




# 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 TNC 640 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 and tapping (with or without floating tap holder), there are optional cycles for thread milling, reaming, boring and for hole patterns, as well as milling cycles for clearing plane surfaces, and for roughing and finishing pockets, slots and studs.

**Cycles for complex contours**
Clearing pockets with combined contours is aided greatly by **Subcontour List cycles** (SL). 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.

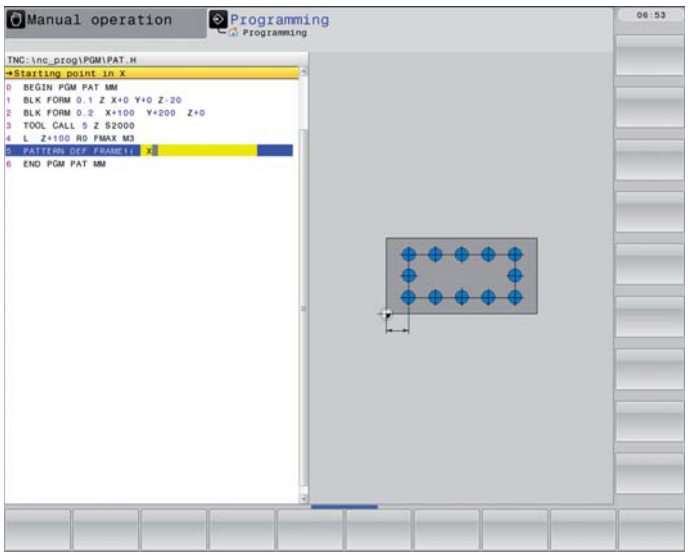
Up to twelve **subcontours** can be superimposed for machining. The control automatically calculates the resulting contour and the tool paths for roughing or clearing the surfaces. Subcontours can be pockets or islands. Different components are combined to form a single pocket in which the tool avoids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

The TNC 640 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.

**Stay simple and flexible when programming machining patterns**
Machining positions are often arranged in patterns on the workpiece. With the TNC 640, 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.

**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.

**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 TNC 640. 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 TNC 640 to suit your own needs.




# Programming in the Workshop
## – Field-Proven Turning Cycles (Option)

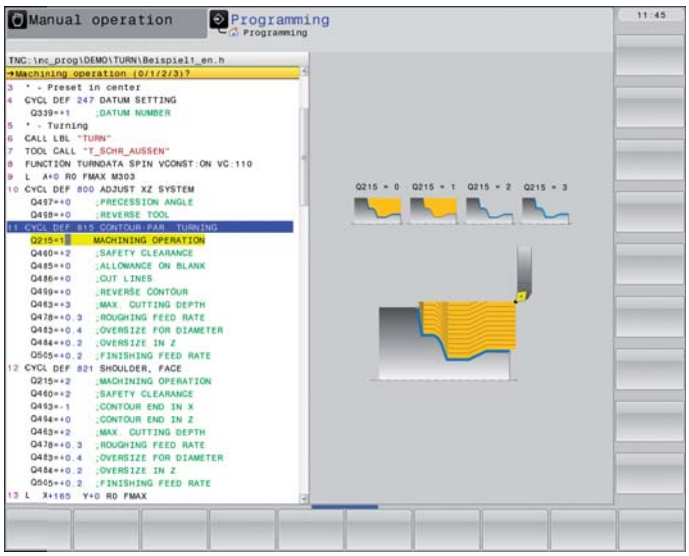
In the area of milling cycles, too, the TNC 640 offers a comprehensive and technologically ambitious package. The are equivalent to the proven and fully developed kernel functions of the HEIDENHAIN lathe controls. The user interface, however, is inspired in its look and functionality by the familiar and proven plain-language dialog. Cycle parameters that come into use both for milling and turning are, of course, used with the same number. For turning operations, as well, you are supported during programming as accustomed with explanatory graphics.

**Machining simple contours**
Various cycles are available for machine simple contours in longitudinal and transverse direction. The surface to be machined might also be inclined and can require a plunging movement. Naturally, the TNC 640 fully automatically takes the angle of the turning tool into account.

**Machining any desired contours**
If the contours to be machines become more complex and can no longer be defined with simple cycle parameters, you can describe them using contour subprograms. The process is completely identical to the procedure when using SL cycles in milling: with Cycle 14 you define the subprogram in which the finished contour is described, and in the respective turning cycle you specify the technological parameters.

During contour description, too, you use exactly the same conversational functions as when defining a milling contour, which of course includes FK free contour programming. Moreover, the turning-specific contours elements recess and undercut are available, which you can insert between contour elements like chamfers and rounding arcs. Besides radial and axis recesses, undercuts are available of the forms E, F, H, K, U and thread undercuts.

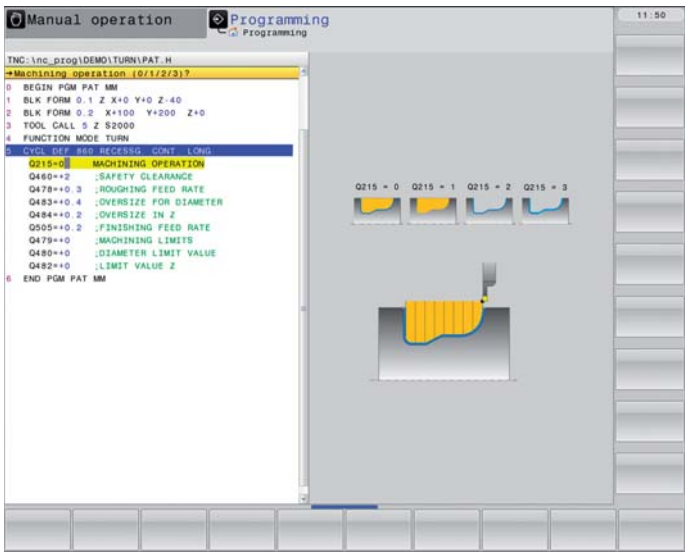
Depending on the cycle, the TNC 640 machines parallel to the axis or the contour. You define the machining operations (roughing, finishing) or oversize under dialog guidance through the corresponding parameters.

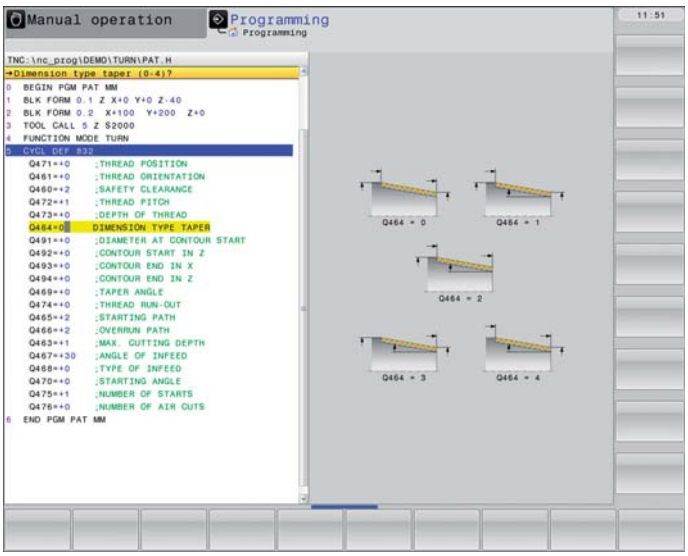


**Recessing**
In this area, as well, the TNC 640 distinguishes itself with ample flexibility and functionality. Simple recessing operations in longitudinal and transverse direction are just as possible as contour recessing, in which the cycle is machined along any desired contour. Here, too, the TNC considers the technological constraints (width of recessing tool from the tool table) and executes the operations quickly and reliably.

**Thread machining**
Simple and expanded cycles are available for longitudinal and transverse machining of cylindrical or tapered threads. You can use cycle parameters to define the manner in which the thread is produced. This enables you to machine a wide variety of materials.

**Orientation of the turning tool**
Milling/turning machines sometime require that the tool be held at an angle during turning or that the side from which the machining begins be changed The TNC provides a simple cycle for such cases. This makes it possible to use an outside turning tool as an inside tool without having to adjust the tool tip and/or the angle of orientation on the tool table.



# Programming in the Workshop
– Reusing Programmed Contour Elements

**Coordinate transformation**
If you should need a contour that has already been programmed at another position or in a different size, the TNC 640 offers you a simple solution: coordinate transformation.
Depending on the machining task, you can, for example **rotate** (milling), **mirror** (milling) or **shift the datum** (milling and turning) in the coordinate system. With a **scaling factor** (milling) you can enlarge or reduce contours to respect shrinkage allowance or oversizes.

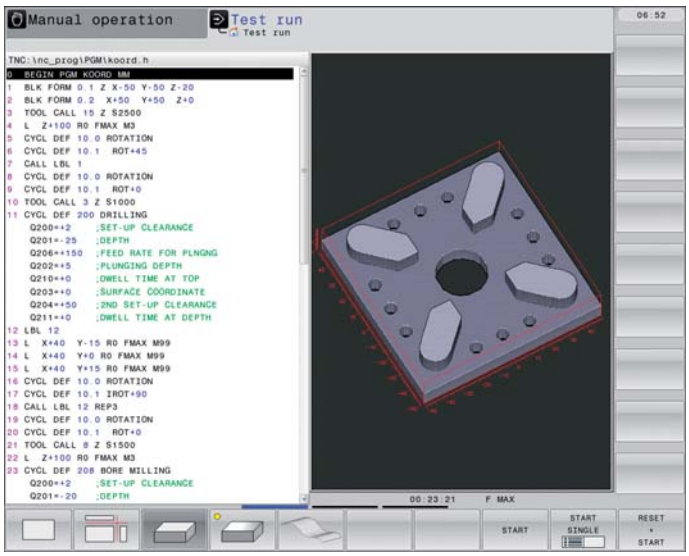
**Program section repeats, subprograms, program calls**
Many machining operations repeat themselves either on the same workpiece or on different workpieces. Once you have programmed a detail there is no reason to have to program it again. With its subprogramming feature, the TNC can save you a great deal of programming time.

In **program section repetition,** you label a section of the program and during program run the TNC repeats the section successively as many times as required.

You can mark a program section as a **subprogram** and then call it at any point in the program and as often as you want.

With the **program call** function you can even use a completely separate program at any place in your current program. This gives you convenient access to pre-programmed, frequently needed working steps or contours.

Of course you can also combine these programming techniques as often as desired.



# – Fast Availability of All Information

Do you have questions on a programming step, but your User's Manual is not at hand? No problem: Both the TNC 640 and the TNC 640 programming station feature **TNCguide,** a convenient **help system** that can display the user documentation in a separate window.

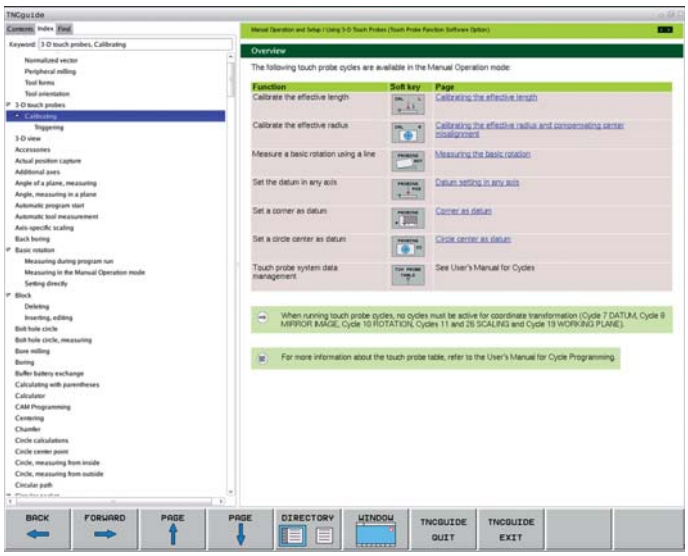
You can activate TNCguide by simply pressing the help key on the TNC keyboard or by clicking any soft key with a cursor in the shape of a question mark. You switch the cursor by simply clicking the help symbol ( ) that is always visible on all TNC screens.

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

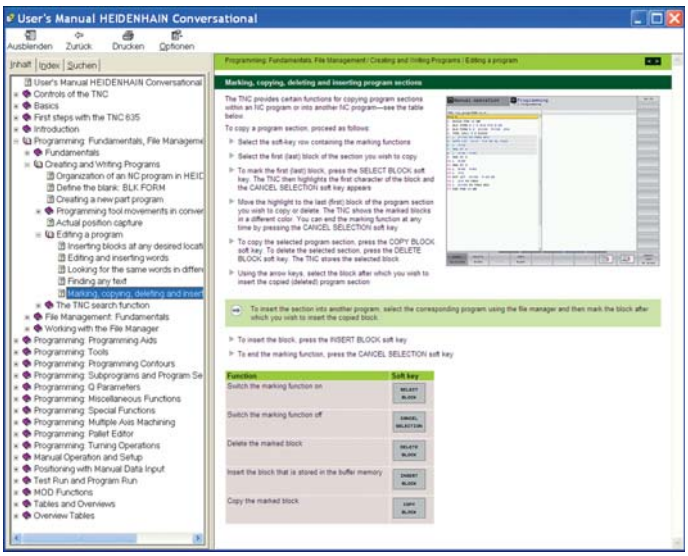
You can download the documentation in the desired language from the HEIDENHAIN homepage into the corresponding language directory on the TNC hard disk.

The following manuals are available in the help system:

- User's Manual for Conversational Programming
- User's Manual for Cycle Programming
- DIN/ISO Programming User's Manual
- User's Manual for the TNC 640 Programming Station (only included in the programming station)




TNCguide integrated in the control, e.g. on the TNC 640 ...




… or at the programming station

# Open for Communication
## – Fast Data Transfer with the TNC

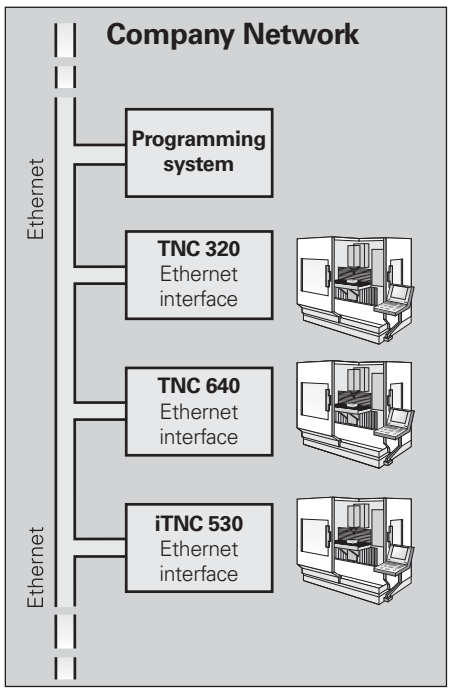
**The networked TNC 640**
The TNC 640 can be integrated into networks and connected with PCs, programming stations and other data storage devices. Even in its standard version, TNC 640 features a latest-generation Fast Ethernet interface in addition to its RS-232-C/V.24 data interface. The TNC 640 communicates with NFS servers and Windows networks in TCP/IP protocol without needing additional software. The fast data transfer at rates of up to 100 Mbps guarantees very short transfer times.

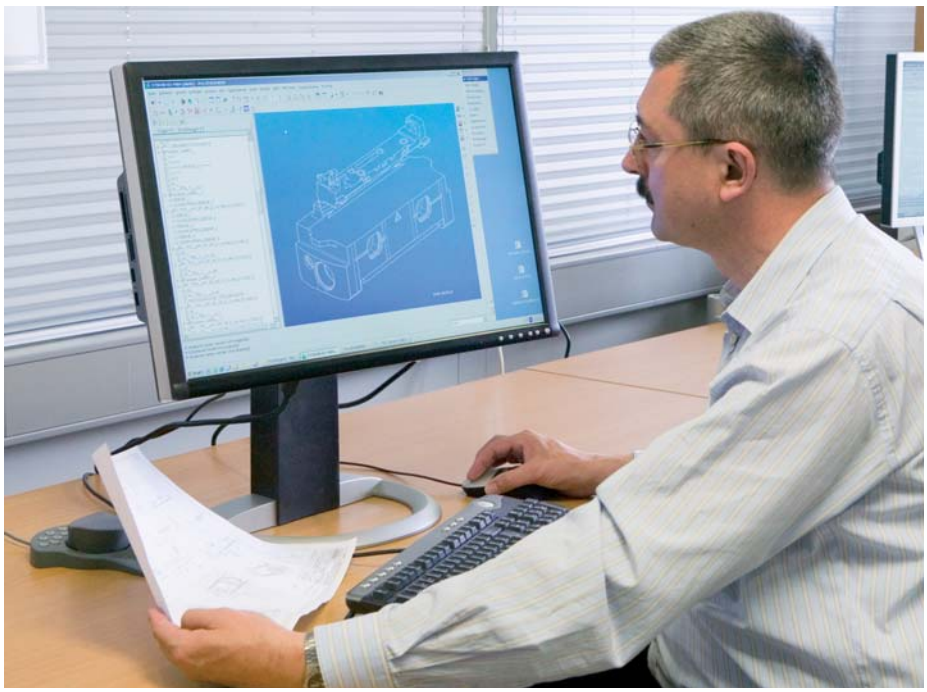
The transmitted programs are saved in the internal memory of the TNC 640 and are run from it at high speed.

For well-organized program management on your control, simply place the individual files in directories (folders). You can structure the respective directories through individual subdirectories.

**Programs for data transfer**
With the aid of the free PC program **TNCremo** from HEIDENHAIN and an Ethernet or other data interface you can
- transfer remotely stored part programs and tool or pallet tables in both directions and
- start the machine.

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



# – The TNC 640 Programming Station

**Why a programming station?**
It's well known that it is easy to create part programs on a TNC 640 at the machine, even while another part is being machined. Nevertheless, it can often happen that short reloading times and other machining tasks hinder any prolonged or concentrated programming work. With the TNC 640 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 HEIDENHAIN conversational or DIN/ISO programs 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 CAD/CAM system. The various views of the program verification graphics help you to easily spot contour damage and hidden details.

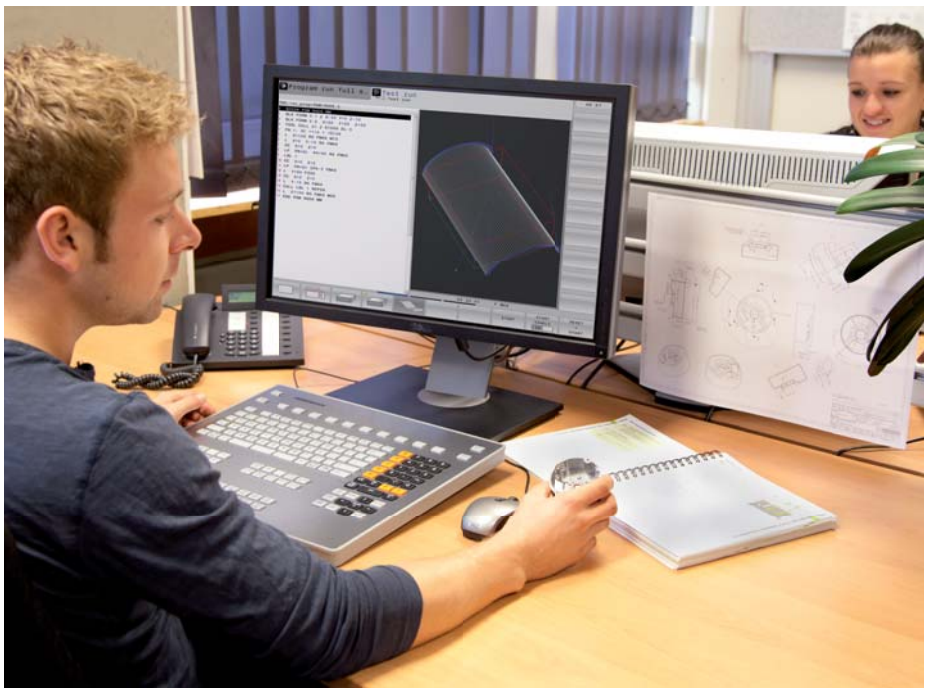
**Training with the programming station**
Because the TNC 640 programming station is based on the same software as the TNC 640, it is ideally suited for apprentice and advanced training. The program is entered on the original keyboard unit. Even the test run functions exactly as it does on the machine. This gives the trainee the experience needed to enable him to safely operate the machine later.

Because it can be programmed with smarT.NC, in plain language, and in ISO, the TNC 640 programming station can also be used in schools for TNC programming training.

**Your workstation**
The programming station software runs on a PC. The PC screen shows you the TNC user interface as on the control, and offers the familiar graphic support. Depending on the version of the programming station, there are several types of possibilities for using it.

The free **demo version** contains all functions of the TNC 640, and permits short programs to be saved. It is programmed over the PC keyboard. On the version with the **TNC operating panel** you then create your programs as always, on a keyboard with the same function keys as the control on the machine. It also has a PC keyboard for G-code programming, file names and comments.

But you can also work without the TNC operating panel: a **virtual keyboard** simulating the TE appears on the PC screen. It provides the TNC 640's most important dialog initiation keys.



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

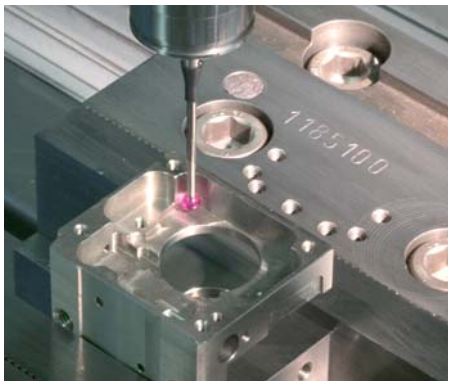
Workpiece touch probes from HEIDENHAIN help you to reduce costs in the workshop and in series production: Together with the TNC 640, touch probes can automatically perform setup, measuring and inspection functions.

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 TNC 640 by the machine tool builder.

Touch probe with **cable connection for signal transmission** for machines with manual tool change:
**TS 220** – TTL version
Touch probes with **infrared signal transmission** for machines with automatic tool change:
**TS 440** – Compact dimensions
**TS 444** – Compact dimensions, battery-free power supply through integrated air turbine generator over central compressed air supply
**TS 640** – Standard touch probe with wide-range infrared transmission
**TS 740** – High probing accuracy and repeatability, low probing force



**SE 640**


**TS 640**


**TS 440**

**TS 220**

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

# Measurement of Milling Cutters
## – Measuring Length, Radius and Wear Directly in the Machine

The tool is of course a definitive factor in ensuring a consistently high level of production quality. This means that an exact measurement of the tool dimensions and periodic inspection of the tool for wear and breakage, as well as the shape of each tooth, are necessary. An effective device for measuring milling cutters is the **TT 140*** touch trigger probe. It is installed directly in the machine's workspace, where it permits tool measurement either before machining or during interruptions.

The TT 140 tool touch probe captures the length and radius of the milling cutter. When probing the tool, either while rotating or at standstill (such as for measuring individual teeth), the contact plate is deflected and a trigger signal is transmitted to the TNC 640.

***** The touch probe must be interfaced to the TNC 640 by the machine tool builder.


**TT 140**

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

# Inspecting and Optimizing Machine Accuracy
## – Easy Calibration of Rotary Axes 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.

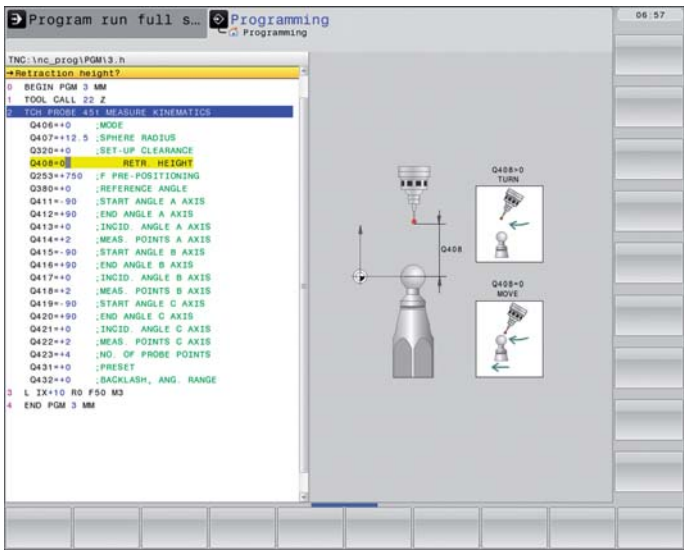
The new TNC function **KinematicsOpt** is an important component to help you meet these high requirements: With a HEIDENHAIN touch probe inserted, a cycle measures your machine's rotary axes fully automatically. The results of measurement are the same regardless of whether the axis is a rotary table,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

To set up the workpiece you can use the direction keys to move the machine axes manually or in incremental jog. A simpler and more sensitive way, however, is to use the electronic handwheels from HEIDENHAIN.

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

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

**HR 410 portable handwheel**
The portable HR 410 is 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. This way you can switch axes and set up the machine at any time—regardless of where you happen to be standing.

The following functions are available:
- Traverse direction keys
- Three keys with preset feed rates for latched traverse
- Actual-position-capture key
- Three keys for machine functions to be defined by the machine tool builder
- Permissive buttons/keys
- Emergency stop button

HR 410

# Overview
## – User Functions

| User functions | Standard | Option | |
|---|---|---|---|
| **Brief description** | • | | Basic version: 3 axes plus spindle |
| | | ○ | 4th NC axis plus auxiliary axis |
| | | | or |
| | | 0-7<br>77<br>78 | } a total of 14 additional NC axes or 13 additional NC axes plus 2nd spindle |
| | • | | Digital current and spindle speed control |
| **Programming** | • | | HEIDENHAIN conversational |
| | • | | DIN/ISO |
| **Position data** | • | | Nominal positions for lines and arcs in Cartesian coordinates or polar coordinates |
| | • | | Incremental or absolute dimensions |
| | • | | Display and entry in mm or inches |
| **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 |
| **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 minimum jerk |
| | | 9 | 3-D tool compensation through surface normal vectors |
| | | 9 | Keeping the tool normal to the contour |
| | | 9 | Tool radius compensation normal to the tool direction |
| **Rotary table machining** | | 8 | Programming of cylindrical contours as if in two axes |
| | | 8 | Feed rate in mm/min |
| **Contour elements** | • | | Straight line |
| | • | | Chamfer |
| | • | | Circular path |
| | • | | Circle center |
| | • | | Circle radius |
| | • | | Tangentially connecting circular arc |
| | • | | Corner rounding |
| | | 50 | Recess |
| | | 50 | Undercut |
| **Approaching and departing the contour** | • | | Via straight line: tangential or perpendicular |
| | • | | Via circular arc |

| User functions | Standard | Option | |
|---|---|---|---|
| **FK free contour programming** | • | | FK free contour programming in HEIDENHAIN conversational format with graphic support for workpiece drawings not dimensioned for NC |
| **Program jumps** | • | | Subroutines |
| | • | | Program-section repeat |
| | • | | Calling any program as a subroutine |
| **Fixed cycles** | • | | Drilling, conventional and rigid tapping, rectangular and circular pockets |
| | • | | Peck drilling, reaming, boring, counterboring, (centering) |
| | | 50 | Area clearance cycles, longitudinal and transverse, paraxial and contour parallel |
| | | 50 | Recessing cycles, radial/axial |
| | • | | Milling internal and external threads |
| | | 50 | Turning inside and outside threads |
| | • | | Clearing level and oblique surfaces |
| | • | | Multioperation machining of straight and circular slots |
| | • | | Multioperation machining of rectangular and circular pockets |
| | • | | Linear and circular point patterns |
| | • | | Contour train, contour pocket—also with contour-parallel machining |
| | • | | OEM cycles (special cycles developed by the machine tool builder) can be integrated |
| **Coordinate transformation** | • | | Datum shift, rotation, mirror image, scaling factor (axis-specific) |
| | | 8 | Tilting the working plane, PLANE function |
| **Q parameters** Programming with variables | • | | Mathematical functions =, +, –, *, /, sin α, cos α, tan α, arc sin, arc cos, arc tan, $a^n$, $e^n$, ln, log, $\sqrt{a}$, $\sqrt{a^2 + b^2}$ |
| | • | | Logical operations (=, = /, <, >) |
| | • | | 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 TNC 640 |
| | • | | Graphical support for programming cycles |
| | • | | Comment and structure blocks in the NC program |
| **Actual position capture** | • | | Actual positions can be transferred directly into the NC program |
| **Test run graphics** Display modes | • | | Graphic simulation before milling operations, even while another program is running |
| | • | | Plan view / projection in 3 planes / 3-D view, also in tilted working plane/3-D pencil-trace graphics |
| | • | | Magnification of details |
| **Interactive programming graphics** | • | | In the Programming and Editing mode, the contour of the NC blocks is drawn on screen while the blocks are being entered (2-D pencil-trace graphics), even while another program is running |
| **Program-run graphics** Display modes | • | | Real-time graphic simulation during execution of the milling program |
| | • | | Plan view / projection in 3 planes / 3-D view |

# Overview
## – User Functions (Continued)

| User functions | Standard | Option | |
|---|---|---|---|
| **Machining time** | •<br>• | | Calculation of machining time in the Test Run operating mode<br>Display of the current machining time in the Program Run operating modes |
| **Returning to the contour** | •<br>• | | Mid-program startup in any block in the program, returning the tool to the calculated nominal position to continue machining<br>Program interruption, contour departure and return |
| **Preset tables** | • | | One preset table for storing reference points |
| **Datum tables** | • | | Several datum tables for storing workpiece-related datums |
| **Pallet tables**<br><br>**Touch probe cycles** | •<br><br>•<br>•<br>•<br>• | | Pallet tables (with as many entries as desired for the selection of pallets, NC programs and datums) can be machined workpiece by workpiece<br><br>Touch probe calibration<br>Compensation of workpiece misalignment, manual or automatic<br>Datum setting, manual or automatic<br>Automatic tool and workpiece measurement |
| **Parallel secondary axes** | •<br>•<br><br>• | | Compensating movement in the secondary axis U, V, W through the principal axis X, Y, Z<br>Including movements of parallel axes in the position display of the associated principal axis (sum display)<br>Defining the principal and secondary axes in the NC program makes it possible to run programs on different machine configurations |
| **Conversational languages** | • | <br>41 | English, German, Chinese (traditional, simplified), Czech, Danish, Dutch, Finnish, French, Hungarian, Italian, Polish, Portuguese, Russian (Cyrillic), Spanish, Swedish<br>For more conversational languages, see *Options* |

# – Options

| Option number | Option | As of NC software 34059x- | ID | Comment |
|---|---|---|---|---|
| **0**<br>**1**<br>**2**<br>**3**<br>**4**<br>**5**<br>**6**<br>**7** | Additional axis | 01 | 354540-01<br>353904-01<br>353905-01<br>367867-01<br>367868-01<br>370291-01<br>370292-01<br>370293-01 | Additional control loops 1 to 8 |
| **8** | Software option 1 | 01 | 617920-01 | **Rotary table machining**<br>• Programming of cylindrical contours as if in two axes<br>• Feed rate in mm/min<br>**Interpolation:** Circular in 3 axes with tilted working plane<br>**Coordinate transformation:** Tilting the working plane, PLANE function |
| **9** | Software option 2 | 01 | 617921-01 | **Interpolation:** Linear in 5 axes<br>**3-D machining**<br>• 3-D tool compensation through surface normal vectors<br>• Keeping the tool normal to the contour<br>• Tool radius compensation normal to the tool direction |
| **18** | HEIDENHAIN DNC | 01 | 526451-01 | Communication with external PC applications over COM component |
| **23** | Display step | 01 | 632986-01 | **Display step** to 0.01 µm or 0.00001° |
| **41** | Additional languages | 01 | 530184-01<br>-02<br>-03<br>-04<br>-06<br>-07<br>-08<br>-09<br>-10 | • Slovenian<br>• Slovak<br>• Latvian<br>• Norwegian<br>• Korean<br>• Estonian<br>• Turkish<br>• Romanian<br>• Lithuanian |
| **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 |
| **50** | Turning | 01 | 634608-01 | Turning functions:<br>• Tool management for turning<br>• Tool-tip radius compensation<br>• Switching between milling and turning modes of operation<br>• Lathe-specific contour elements<br>• Turning cycle package |
| **77** | 4 Additional Axes | 01 | 634613-01 | 4 additional control loops |
| **78** | 8 Additional Axes | 01 | 634614-01 | 8 additional control loops |
| **93** | Extended Tool Management | 01 | 679938-01 | Extended tool management |
| **133** | Remote Desktop Manager | 01 | 894423-01 | Display and operation of external computer units (e.g. a Windows PC) |

# – Accessories

| Accessories | |
|---|---|
| **Electronic handwheels** | • One **HR 410:** Portable handwheel, or<br>• One **HR 130:** Panel-mounted handwheel or<br>• Up to three **HR 150:** Panel-mounted handwheels via HRA 110 handwheel adapter |
| **Workpiece measurement** | • **TS 220:** Touch trigger probe with cable connection or<br>• **TS 440:** Touch trigger probe with infrared transmission or<br>• **TS 444:** Touch trigger probe with infrared transmission or<br>• **TS 640:** Touch trigger probe with infrared transmission or<br>• **TS 740:** Touch trigger probe with infrared transmission |
| **Tool measurement** | • **TT 140:** Touch trigger probe |
| **TNC 640 programming station** | Control software for PCs for programming, archiving, and training<br>• Full version with control keyboard<br>• Full version with virtual keyboard<br>• Demo version (operated via PC keyboard—free of charge) |
| **Software for PCs** | • **TeleService:** Software for remote diagnostics, monitoring, and operation<br>• **TNCdiag:** Software for fast and easy fault diagnosis<br>• **CycleDesign:** Software for creating your own cycle structure<br>• **TNCremo:** Software for data transfer—free of charge<br>• **TNCremoPlus:** Software for data transfer with live-screen function |

# – Specifications

| Specifications | Standard | Option | |
|---|---|---|---|
| **Components** | • | | MC 62xx, or MC 63xx main computers |
| | • | | CC 61xx or UEC 11x controller units |
| | • | | TE 745 or TE 740 keyboard unit |
| | • | | BF 760 19-inch TFT color flat-panel display with soft keys |
| **Operating system** | • | | HEROS 5 real-time operating system for machine control |
| **Memory** | • | | RAM memory: *MC 62xx:* 1 GB<br>*MC 63xx:* 2 GB |
| | • | | Hard disk with at least 21 GB program memory |
| **Input resolution and display step** | • | | Linear axes: up to 0.1 µm |
| | • | | Angular axes: To 0.000 1° |
| | | 23 | Linear axes: to 0.01 µm |
| | | 23 | Angular axes: to 0.000 01° |
| **Input range** | • | | Maximum 99 999.999 mm (3937 inches) or 99 999.999° |
| **Interpolation** | • | | Line in 4 axes |
| | | 9 | Line 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 |
| **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/1 024 |
| | • | | Cycle time of position controller: 200 µs (100 µs with option 49) |
| | • | | Cycle time of speed controller: 200 µs (100 µs with option 49) |
| | • | | Cycle time of current controller: 100 µs (minimum 50 µs with option 49) |
| **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 spikes during circular movements, hysteresis, thermal expansion |
| | • | | Static friction, sliding friction |
| **Data interfaces** | • | | One each RS-232-C/V.24 max. 115 Kbps |
| | • | | Extended data interface with LSV2 protocol for remote operation of the TNC over the data interface with the HEIDENHAIN software TNCremo or TNCremoPlus |
| | • | | 2 x 100BaseT Fast Ethernet interface |
| | • | | 2 x USB (1 x front, 1 x MC) |
| | | 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: 5 °C to 40 °C |
| | • | | Storage: –20 °C to +60 °C |

# – Comparison of Controls

| Comparison of controls | TNC 620 | TNC 640 | iTNC 530 |
|---|---|---|---|
| **Area of application** | **Standard milling** | **High-end milling/turning** | **High-end milling** |
| • Basic machining centers (up to 5 axes + spindle) | ● | ● | ● |
| • Machine tools/machining centers (up to 18 axes + 2 spindles) | – | ● (Limited at present) | ● |
| • Milling/turning operation (up to 18 axes + 2 spindles) | – | ● | – |
| **Program entry** | | | |
| • In HEIDENHAIN conversational format | ● | ● | ● |
| • According to ISO | ● | ● | ● |
| • With smarT.NC | ★ | ★ | ● |
| • DXF converter | ★ | ★ | Option |
| • FK free contour programming | Option | ● | ● |
| • Extended milling and drilling cycles | Option | ● | ● |
| • Turning Cycles | – | Option | – |
| **NC program memory** | 300 MB | > 21 GB | > 21 GB |
| **5-axis and high-speed machining** | Option (limited) | 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 for screen and keyboard** (as of 2012) | 15-inch screen | 19-inch screen | 15/19-inch screen |
| **Optimized user interface** | – | ● | – |
| **AFC adaptive feed control** | – | ★ | Option |
| **DCM dynamic collision monitoring** | – | ★ | Option |
| **Global program settings (GS)** | – | ★ | Option |
| **KinematicsOpt** | Option | Option | Option |
| **Touch probe cycles** | Option | ● | ● |
| **Pallet management** | Option | ● | ● |
| **Handwheels with display** | ★ | ★ | ● |

**TNC 620**

**TNC 640**

**iTNC 530**

● Function available
★ Function planned
● Special feature of TNC 640

# HEIDENHAIN

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

**www.heidenhain.de**

**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**
08468 Heinsdorfergrund, Deutschland
☏ 03765 69544

**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

**BA** Bosnia and Herzegovina → **SL**

**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** **Belarus**
**GERTNER Service GmbH**
50354 Huerth, Germany
www.gertnergroup.com

**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**
02770 Espoo, 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

**ME** Montenegro → **SL**

**MK** Macedonia → **BG**

**MX** **HEIDENHAIN CORPORATION MEXICO**
20235 Aguascalientes, Ags., Mexico
E-mail: info@heidenhain.com

**MY** **ISOSERVE Sdn. Bhd**
56100 Kuala Lumpur, Malaysia
E-mail: isoserve@po.jaring.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**
02-489 Warszawa, Poland
www.apserwis.com.pl

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

**RO** **HEIDENHAIN Reprezentanţă Romania**
Braşov, 500338, Romania
www.heidenhain.ro

**RS** Serbia → **BG**

**RU** **OOO HEIDENHAIN**
125315 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** **Posredništvo HEIDENHAIN NAVO d.o.o.**
2000 Maribor, Slovenia
www.heidenhain-hubl.si

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

**TR** **T&M Mühendislik San. ve Tic. LTD. ŞTİ.**
34728 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.gertnergroup.com

**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!***

892 916-21 · 20 · 9/2011 · F&W · Printed in Germany